OpenSPARC™ T1 Processor Design and Verification User's Guide

Sun Microsystems, Inc.
www.sun.com

Part No. 819-5019-12
March 2009, Revision E

Submit comments about this document by clicking the Feedback[+] link at: http://docs.sun.com

Contents

Figures

Tables

Preface

The *OpenSPARC™ T1 Processor Design and Verification User's Guide* gives an overview of the design hierarchy on the OpenSPARC T1 processor. It also describes the files, procedures, and tools needed for running simulations and synthesis on the OpenSPARC T1 processor.

This book covers the following topics:

- Design and Verification implementation overview
- Design and Verification directory and files structure
- System and Electronic Design Automation (EDA) tools required to run simulations and synthesis
- Tools and scripts required to run simulation or complete regressions, including simulation flow
- Synthesis flow and scripts

How This Document Is Organized

Chapter 1 describes quick steps to run simulations after you download the design and verification files from the web site. It also includes system requirements and EDA tools requirements to run simulations and synthesis.

Chapter 2 gives an overview of the OpenSPARC T1 design hierarchy and directory structure.

Chapter 3 gives an overview of the OpenSPARC T1 verification environment implementation and directory structure. The verification environment includes test benches, tests, scripts, and Verilog Programming Language Interface (PLI).

Chapter 4 describes the synthesis flow and synthesis scripts.

Chapter 5 describes the Synplicity Pro software scripts and the XST software scripts for synthesizing field programmable gate arrays (FPGA).

Chapter 6 describes the included EDK project, which enables the user to download the synthesized OpenSPARC T1 core to a Xilinx FPGA, run diagnostic tests on it, and boot hypervisor.

Appendix A has manual pages for regression commands.

Using UNIX Commands

This document might not contain information about basic UNIX® commands and procedures such as shutting down the system, booting the system, and configuring devices. Refer to the following for this information:

- Software documentation that you received with your system
- Solaris™ Operating System documentation, which is at:

  `http://docs.sun.com`

Shell Prompts

Shell	Prompt
C shell	*machine-name*`%`
C shell superuser	*machine-name*`#`
Bourne shell and Korn shell	`$`
Bourne shell and Korn shell superuser	`#`

Typographic Conventions

Typeface*	Meaning	Examples
`AaBbCc123`	The names of commands, files, and directories; on-screen computer output	Edit your `.login` file. Use `ls -a` to list all files. `% You have mail.`
`AaBbCc123`	What you type, when contrasted with on-screen computer output	`%` **`su`** `Password:`
AaBbCc123	Book titles, new words or terms, words to be emphasized. Replace command-line variables with real names or values.	Read Chapter 6 in the *User's Guide*. These are called *class* options. You *must* be superuser to do this. To delete a file, type `rm` *filename*.

* The settings on your browser might differ from these settings.

Related Documentation

The documents listed as online or download are available at:

`http://www.opensparc.net/`

Application	Title	Part Number	Format	Location
OpenSPARC T1 instruction set	*UltraSPARC Architecture 2005 Specification*	950-4895-03	PDF	Online
OpenSPARC T1 processor's internal registers	*UltraSPARC T1 Supplement to the UltraSPARC Architecture 2005*	819-3404-02	PDF	Online
OpenSPARC T1 megacells	*OpenSPARC T1 Processor Megacell Specification*	819-5016-10	PDF	Download
OpenSPARC T1 signal pin list	*OpenSPARC T1 Processor Datasheet*	819-5015-10	PDF	Download
OpenSPARC T1 processor J-Bus and SSI interfaces	*OpenSPARC T1 Processor External Interface Specification*	819-5014-10	PDF	Download

Documentation, Support, and Training

Sun Function	URL
Documentation	http://www.sun.com/documentation/
Support	http://www.sun.com/support/
Training	http://www.sun.com/training/

Third-Party Web Sites

Sun is not responsible for the availability of third-party web sites mentioned in this document. Sun does not endorse and is not responsible or liable for any content, advertising, products, or other materials that are available on or through such sites or resources. Sun will not be responsible or liable for any actual or alleged damage or loss caused by or in connection with the use of or reliance on any such content, goods, or services that are available on or through such sites or resources.

Sun Welcomes Your Comments

Sun is interested in improving its documentation and welcomes your comments and suggestions. You can submit your comments by going to:

http://www.sun.com/hwdocs/feedback

Please include the title and part number of your document with your feedback:

OpenSPARC T1 Processor Design and Verification User's Guide,
part number 819-5019-12

CHAPTER 1

Quick Start

This chapter covers the following topics:

- System Requirements
- EDA Tool Requirements
- Running Simulations and Synthesis

Before you start running simulations or synthesis, make sure you meet system requirements and that you have the required Electronic Design Automation (EDA) tools. Once you download the OpenSPARC T1 `tar` file from the `http://www.opensparc.net` web site, follow the steps in this chapter to get started and run your first regression on the OpenSPARC T1 design.

1.1 System Requirements

OpenSPARC T1 regressions are currently supported to run only on SPARC systems running the Solaris 9 or Solaris 10 Operating System.

Disk space requirements are listed in TABLE 1-1.

TABLE 1-1 Disk Space Requirements

Disk Space required	Required for:
1.5 Gbyte	Download, unzip or uncompress, and extract from the tar file
1.6 Gbyte	Run a mini-regression
67 Gbyte	Run a full regression
0.3 Gbyte	Run synthesis
70.4 Gbyte	Total

1.2 EDA Tool Requirements

This section describes the commercial EDA tools required for running simulations for the OpenSPARC T1 processor and synthesizing OpenSPARC T1 Verilog Register Transfer Level (RTL) code.

1.2.1 EDA Simulation Tools

The following EDA tools are required to run Verilog simulations: Verilog Simulator, either VCS or NCVerilog.

- VCS from Synopsys, version 7.1.1R21 or later
- NCVerilog from Cadence, version 5.3.s2 or later

It is permissible to use a Verilog Simulator other than VCS or NCVerilog. See details in Section 3.2.3, "Running Regression With Other Simulators" on page 3-5.

The following EDA tools are optional for running Verilog simulations:

- Vera from Synopsys, version 6.2.8 or later
- Debussy from Novas, version 5.3v19 or later

1.2.2 EDA Synthesis Tools

The following EDA tool is required to perform Verilog RTL synthesis:

- Design Compiler from Synopsys, version X-2005.09 or later

One of the following EDA tool is required to perform Verilog RTL synthesis for field programmable gate arrays (FPGA):

- Synplicity Pro from Synplicity, version 8.5 or later or
- Xilinx Synthesis Technology (XST) from Xilinx, version 9.1i or later

1.2.3 FPGA Tools

The following EDA tools are required to place and route a design on a Xilinx FPGA, download the design to the Xilinx FPGA, and run tests on the FPGA system:

- Embedded Development Kit (EDK) from Xilinx, version 9.1i or later
- Integrated Synthesis Environment (ISE) from Xilinx, version 9.1i or later

- Modelsim from Mentor Graphics, version 6.1e

1.3 Running Simulations and Synthesis

This section outlines the steps needed to obtain the simulation tools, set up the simulation environment, run the simulation, and read its log file.

1.3.1 Get the Simulation Files

1. Download the file.

Download the `OpenSPARCT1.tar.bz2` file from the `http://www.opensparc.net` web site. For this procedure's examples, the destination directory is:

`/home/johndoe/OpenSPARCT1`

2. Change directories to the directory where you downloaded the file. For example:

```
% cd /home/johndoe/OpenSPARCT1
```

3. Use the `bunzip2` command to unzip the file.

```
% bunzip2 OpenSPARC_1.tar.bz2
```

4. Extract the tar file using the `tar` command.

```
% tar -xvf OpenSPARC_1.tar
```

This step creates the files and subdirectories listed in TABLE 1-2 in your current directory.

TABLE 1-2 Contents of the OpenSPARCT1 Directory

Name	Type	Description
OpenSPARCT1.cshrc	File	File to set up environment variables and paths
README	File	Instructions to set up and run simulations
lib	Directory	Verilog libraries
verif	Directory	Verification directories and files
design	Directory	Verilog RTL for OpenSPARC T1 design
tools	Directory	Tools and scripts needed to run simulations and synthesis
doc	Directory	Documentation in PDF form for the OpenSPARC T1 processor

1.3.2 Set Up Environment Variables

Edit the OpenSPARCT1.cshrc file to set the required environment variables as shown in TABLE 1-3:

TABLE 1-3 Environment Variables in .cshrc File

Environment Variable	Usage	Example value
DV_ROOT	Running simulations and synthesis	/home/johndoe/OpenSPARCT1 (Directory where you ran the tar command above)
MODEL_DIR	Running simulations	/home/johndoe/OpenSPARCT1_model (Directory where you want to run your simulations)
VERA_HOME	Running simulations	/import/EDAtools/vera/vera,v6.2.10/5.x (Directory where Vera is installed)
NOVAS_HOME	Running simulations	/import/EDAtools/debussy/debussy,v5.3v19/5.x (Directory where Debussy is installed)
VCS_HOME	Running VCS simulations	/import/EDAtools/vcs7.1.1R21 (Directory where VCS is installed)
NCV_HOME	Running NCVerilog simulations	/import/EDAtools/ncverilog/ncverilog.v5.3.s2/5.x (Directory where NCVerilog is installed)
SYN_HOME	Running synthesis	/import/EDAtools/synopsys/synopsys.vX-2005.09 (Directory where Synopsys is installed)

TABLE 1-3 Environment Variables in .cshrc File *(Continued)*

Environment Variable	Usage	Example value
CC_BIN	Compiling PLI code	/usr/dist/pkgs/devpro,v4.2/5.x-sparc/bin (Directory where C++ Compiler binaries are installed)
LM_LICENSE_FILE	Running simulations and synthesis	/import/EDAtools/licenses/synopsys_key:/import/EDAtools/licenses/ncverilog_key (EDA tool license files)
SYNP_HOME	Running Synplicity for FPGA synthesis	/import/EDAtools/synplicity/synplify.v8.5/fpga_85 (Directory where Synplicity is installed)
MODEL_HOME	Running Modelsim	/import/EDAtools/modelsim.v6.1e/modeltech

Once you set the environment variables from TABLE 1-3, the OpenSPARCT1.cshrc file sets the following environment variables:

- TRE_ENTRY
- TRE_LOG
- TRE_SEARCH
- ENVDIR
- PERL_MODULE_BASE

The OpenSPARCT1.cshrc script also adds the following directories to your PATH and path variables:

- $DV_ROOT/tools/bin
- $NCV_HOME/tools/bin
- $VCS_HOME/bin
- $VERA_HOME/bin
- $SYN_HOME/sparcOS5/syn/bin
- $CC_BIN

After completing your OpenSPARCT1.cshrc file edits, source it by using the source command:

```
% source /home/johndoe/OpenSPARCT1/OpenSPARCT1.cshrc
```

You might want to include the above command in your ~/.cshrc file so that the above environment variables are set every time you log in.

Finally, create the following symbolic link to set up the correct platform files for the verification environment. For example, if you are running the verification on a x86_64 Linux cluster, you would create the symbolic link as follows:

```
% cd $DV_ROOT/tools/env
% ln -s Makefile.Linux.x86_64 Makefile.system
```

1.3.3 Run Your First Regression

Note – OpenSPARC T1 Release 1.4 includes one more environment for single-core, single-thread implementation of OpenSPARC T1. Tests included in the `thread1` environment are a subset of the `core1` environment. The `thread1` environment does not include tests that verify multi-threaded functionality and Stream Processing Unit (SPU) related functionality.

The OpenSPARC T1 Design/Verification package comes with two test bench environments: `core1` and `chip8`.

The `core1` environment consists of:

- One SPARC CPU core
- Cache
- Memory
- Crossbar

The `core1` environment does not have an I/O subsystem.

The `chip8` environment consists of:

- A full OpenSPARC T1 chip, including all eight cores
- Cache
- Memory
- Crossbar
- I/O subsystem

Each environment can perform either a mini-regression or a full regression.

To run a regression, use the `sims` command as described in Section 1.3.3.1, "To Run a Regression" on page 1-7. The important parameters for the `sims` command are:

- `-sim_type`: Simulator type

Set this to `vcs` or `ncv`. For example: **`-sim_type=vcs`**

■ `-group`: Regression group name

The choices for `-group` are: `core1_mini`, `core1_full`, `chip8_mini`, `chip8_full`, `thread1_mini` and `thread1_full`.
For example: **`-group=core1_mini`**

1.3.3.1 To Run a Regression

1. **Create the `$MODEL_DIR` directory.**

```
% mkdir $MODEL_DIR
```

2. **Change directory to `$MODEL_DIR`.**

```
% cd $MODEL_DIR
```

This is where the simulations are run.

3. **Run a mini-regression for the `core1` environment using the VCS simulator.**

```
% sims -sim_type=vcs -group=core1_mini
```

This command creates two directories:

- A directory called `core1` under `$MODEL_DIR`. The regression compiles Vera and Verilog code under the `core1` directory. This is the Vera and Verilog "build" directory.
- A directory named with today's date and a serial number, such as `2006_01_07_0` (the format is `YYYY_MM_DD_ID`) under the current directory where simulations will run. This is the Verilog simulation's "run" directory. There is one subdirectory under this directory for each diagnostics test.

By default, the simulations are run with Vera. If you do not want to use Vera, add following option to the `sims` command:

`-novera_build -novera_run`

4. **Once simulations are completed, run the `regreport` command to generate a regression report.**

```
% cd run-directory
% regreport $PWD > report.log
```

Where *run-directory* is the “run” directory created in the above step, such as `2006_01_07_0`.

The `core1_mini` regression has 68 tests. An example of its `report.log` output is shown in CODE EXAMPLE 1-1.

CODE EXAMPLE 1-1 Example `report.log` Regression Output

```
============================================================================
Status:         core1_mini |       ALL |
----------------------------------------------------------------------------
           PASS:        68 |        68 |
           FAIL:         0 |         0 |
   Diag Problem:         0 |         0 |
License Problem:         0 |         0 |
  MaxCycles Hit:         0 |         0 |
 Socket Problem:         0 |         0 |
        Timeout:         0 |         0 |
    LessThreads:         0 |         0 |
 Simics Problem:         0 |         0 |
    Performance:         0 |         0 |
Killed By Job Q:         0 |         0 |
        Unknown:         0 |         0 |
     UnFinished:         0 |         0 |
   flexlm error:         0 |         0 |
----------------------------------------------------------------------------
     Diag Count:        68 |        68 |
----------------------------------------------------------------------------
```

If your `report.log` file displays a similar status, you have successfully completed running a mini-regression for the OpenSPARC T1 processor.

1.3.4 Run Your First Synthesis

The command to run a synthesis is `rsyn`. For example, to run a synthesis for one of the modules called `efc`, type:

```
% rsyn efc
```

This command runs a synthesis for the `efc` block and creates gate level netlists under the `$DV_ROOT/design/sys/iop/efc/synthesis/gate` directory.

The synthesis flow and scripts are described in more detail in Chapter 4.

1.3.5 Gate-Level Verification

OpenSPARC T1 depends heavily on Cross-Module References (XMRs) within the verification environment. Therefore, dropping in a `netlist` in place of the RTL core will produce a high number of XMR errors. Because of this, a simple playback support is now added. The details of the suggested methodology are described in Chapter 3.

CHAPTER 2

OpenSPARC T1 Design Implementation

This chapter gives details on the following topics:

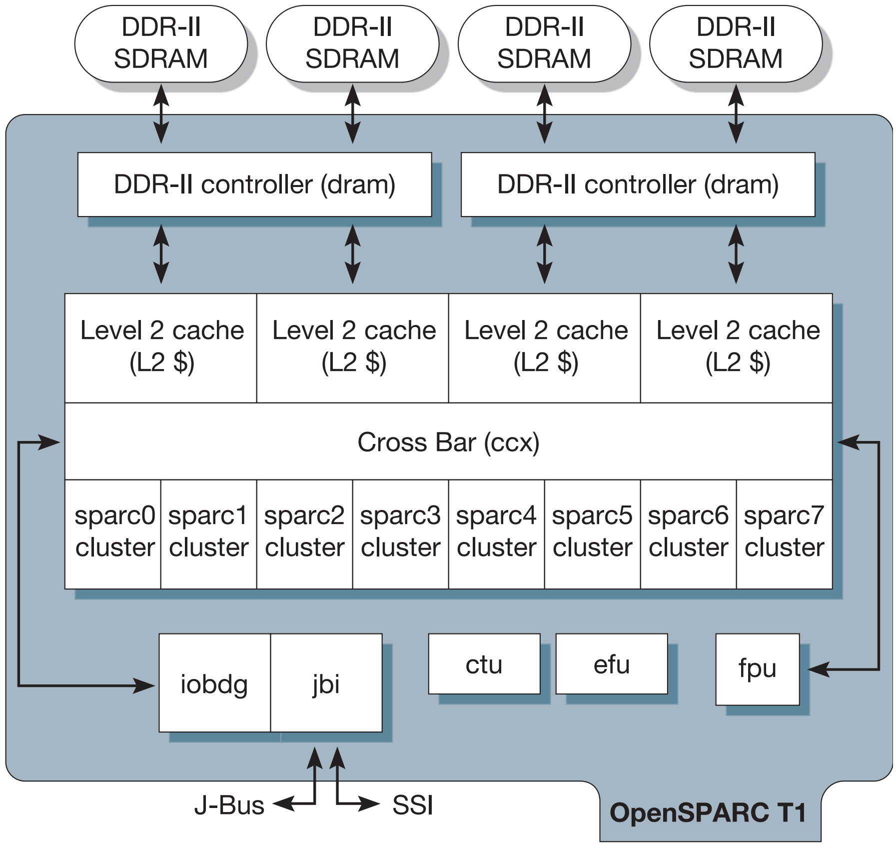
2.1 OpenSPARC T1 Design Hierarchy

The top-level Verilog module for the OpenSPARC T1 processor is called `OpenSPARCT1`. There are various types of design blocks at the top level:

- **Cluster –**A hierarchical block with one or more instances of this block in the design. For example, a SPARC CPU core is called `sparc` and has eight instances at the top level.
- **Pads –** Input, Output and Bi-directional pins on the OpenSPARC T1 processor, including input buffer, output driver, etc. For example, `pad_ddr0` contains pads for DDR bank 0.
- **Repeaters –** Many buffers and repeaters at the top level for signals going to blocks or signals with long traces in the physical implementation, such as `dram0_ddr0_rptr`.
- **Clock –** This includes global clock distribution, including buffers, drivers, repeaters, and so on.

A block diagram of the OpenSPARC T1 processor is shown in FIGURE 2-1.

FIGURE 2-1 OpenSPARC T1 Block Diagram



2.2 Module Directory Structure

The Verilog RTL for the OpenSPARC T1 processor is in the `$DV_ROOT/design/sys/iop` directory. All the top-level modules that make up that RTL, and their locations, are listed in TABLE 2-1. You can also browse the Verilog source code on the OpenSPARC web site at `http://www.opensparc.net`.

TABLE 2-1 OpenSPARC T1 Top-Level Modules

Module Name	Type	Number of Instances	Instance Names	Directory Location under `$DV_ROOT/design/sys/iop`	Description
`ccx`	Cluster	1	ccx	`ccx`	CPU-Cache Cross bar
`ctu`	Cluster	1	ctu	`ctu`	Clock and Test Unit
`dram`	Cluster	2	dram02, dram13	`dram`	DRAM controller
`efc`	Cluster	1	efc	`efc`	e-Fuse Cluster
`fpu`	Cluster	1	fpu	`fpu`	Floating Point Unit
`iobdg`	Cluster	1	iobdg	`iobdg`	I/O bridge
`jbi`	Cluster	1	jbi	`jbi`	J-Bus Interface
`scbuf`	Cluster	4	scbuf[0-3]	`scbuf`	L2 $ buffer
`scdata`	Cluster	4	scdata[0-3]	`scdata`	L2 $ data
`sctag`	Cluster	4	sctag[0-3]	`sctag`	L2 $ tag
`sparc`	Cluster	8	sparc[0-7]	`sparc`	SPARC CPU core
`pad_ddr0`	Pad	1	pad_ddr0	`pads`	DDR0 pads
`pad_ddr1`	Pad	1	pad_ddr1	`pads`	DDR1 pads
`pad_ddr2`	Pad	1	pad_ddr2	`pads`	DDR2 pads
`pad_ddr3`	Pad	1	pad_ddr3	`pads`	DDR3 pads
`pad_efc`	Pad	1	pad_efc	`pads`	efc pads
`pad_jbusr`	Pad	1	pad_jbusr	`pads`	J-Bus pads
`pad_jbusl`	Pad	2	pad_jbusl, pad_dbg	`pads`	J-Bus pads
`pad_misc`	Pad	1	pad_misc	`pads`	Miscellaneous pads

TABLE 2-1 OpenSPARC T1 Top-Level Modules *(Continued)*

Module Name	Type	Number of Instances	Instance Names	Directory Location under `$DV_ROOT/design/sys/iop`	Description
`bw_temp_diode`	Pad	2	pad_diode0, pad_diode1	`analog`	Temperature diode pads
`bw_ctu_pad_cluster`	Pad	1	pad_ctu	`analog`	CTU pads
`ccx_iob_rptr`	Repeater	1	ccx_iob_rptr	`cmp`	ccx repeater
`ccx_spc_rpt`	Repeater	8	ccx_spc_rpt[0-7]	`cmp`	ccx repeater
`ctu_top_rptr`	Repeater	1	ctu_top_rptr	`cmp`	ctu repeater
`ctu_top_rptr2`	Repeater	1	ctu_top_rptr2	`cmp`	ctu repeater
`ctu_bottom_rptr`	Repeater	1	ctu_bottom_rptr	`cmp`	ctu repeater
`ctu_bottom_rptr2`	Repeater	1	ctu_bottom_rptr2	`cmp`	ctu repeater
`dram0_ddr0_rptr`	Repeater	1	dram0_ddr0_rptr0	`cmp`	dram repeater
`dram1_ddr1_rptr`	Repeater	1	dram1_ddr1_rptr0	`cmp`	dram repeater
`dram2_ddr2_rptr`	Repeater	1	dram2_ddr2_rptr0	`cmp`	dram repeater
`dram3_ddr3_rptr`	Repeater	1	dram3_ddr3_rptr0	`cmp`	dram repeater
`dram_ddr_pad_rptr`	Repeater	2	dram0_ddr0_rptr2, dram2_ddr2_rptr2	`cmp`	dram repeater
`dram_ddr_pad_rptr_south`	Repeater	2	dram1_ddr1_rptr2, dram3_ddr3_rptr2	`cmp`	dram repeater
`dram_ddr_rptr`	Repeater	2	dram0_ddr0_rptr1, dram2_ddr2_rptr1	`cmp`	dram repeater
`dram_ddr_rptr_south`	Repeater	2	dram1_ddr1_rptr1, dram3_ddr3_rptr1	`cmp`	dram repeater
`dram_l2_buf2`	Repeater	8	dram_sc_[0-3]_rep1, dram_sc_[0-3]_rep3	`cmp`	dram repeater
`dram_sc_0_rep2`	Repeater	1	dram_sc_0_rep2	`cmp`	dram repeater
`dram_sc_1_rep2`	Repeater	1	dram_sc_1_rep2	`cmp`	dram repeater
`dram_sc_2_rep2`	Repeater	1	dram_sc_2_rep2	`cmp`	dram repeater
`dram_sc_3_rep2`	Repeater	1	dram_sc_3_rep2	`cmp`	dram repeater
`ff_dram_sc_bank0`	Repeater	1	ff_dram_sc_bank0	`cmp`	dram repeater
`ff_dram_sc_bank1`	Repeater	1	ff_dram_sc_bank1	`cmp`	dram repeater
`ff_dram_sc_bank2`	Repeater	1	ff_dram_sc_bank2	`cmp`	dram repeater

TABLE 2-1 OpenSPARC T1 Top-Level Modules *(Continued)*

Module Name	Type	Number of Instances	Instance Names	Directory Location under $DV_ROOT/design/sys/iop	Description
ff_dram_sc_bank3	Repeater	1	ff_dram_sc_bank3	cmp	dram repeater
ff_jbi_sc0_1	Repeater	1	ff_jbi_sc0_1	cmp	jbi repeater
ff_jbi_sc0_2	Repeater	1	ff_jbi_sc0_2	cmp	jbi repeater
ff_jbi_sc1_1	Repeater	1	ff_jbi_sc1_1	cmp	jbi repeater
ff_jbi_sc1_2	Repeater	1	ff_jbi_sc1_2	cmp	jbi repeater
ff_jbi_sc2_1	Repeater	1	ff_jbi_sc2_1	cmp	jbi repeater
ff_jbi_sc2_2	Repeater	1	ff_jbi_sc2_2	cmp	jbi repeater
ff_jbi_sc3_1	Repeater	1	ff_jbi_sc3_1	cmp	jbi repeater
ff_jbi_sc3_2	Repeater	1	ff_jbi_sc3_2	cmp	jbi repeater
iob_ccx_rptr	Repeater	1	iob_ccx_rptr	cmp	iob repeater
iob_jbi_rptr_0	Repeater	1	iob_jbi_rptr_0	cmp	iob repeater
iob_jbi_rptr_1	Repeater	1	iob_jbi_rptr_1	cmp	iob repeater
jbi_l2_buf2	Repeater	4	rep_jbi_sc[0-3]_1	cmp	jbi repeater
rep_jbi_sc0_2	Repeater	1	rep_jbi_sc0_2	cmp	jbi repeater
rep_jbi_sc1_2	Repeater	1	rep_jbi_sc1_2	cmp	jbi repeater
rep_jbi_sc2_2	Repeater	1	rep_jbi_sc2_2	cmp	jbi repeater
rep_jbi_sc3_2	Repeater	1	rep_jbi_sc3_2	cmp	jbi repeater
sc_0_1_dbg_rptr	Repeater	1	sc_0_1_dbg_rptr	cmp	L2 repeater
sc_2_3_dbg_rptr	Repeater	1	sc_2_3_dbg_rptr	cmp	L2 repeater
sctag_cpx_rptr_0	Repeater	1	sctag_cpx_rptr_0	cmp	L2 repeater
sctag_cpx_rptr_1	Repeater	1	sctag_cpx_rptr_1	cmp	L2 repeater
sctag_cpx_rptr_2	Repeater	1	sctag_cpx_rptr_2	cmp	L2 repeater
sctag_cpx_rptr_3	Repeater	1	sctag_cpx_rptr_3	cmp	L2 repeater
sctag_pcx_rptr_0	Repeater	1	sctag_pcx_rptr_0	cmp	L2 repeater
sctag_pcx_rptr_1	Repeater	1	sctag_pcx_rptr_1	cmp	L2 repeater
sctag_pcx_rptr_2	Repeater	1	sctag_pcx_rptr_2	cmp	L2 repeater
sctag_pcx_rptr_3	Repeater	1	sctag_pcx_rptr_3	cmp	L2 repeater
sctag_scbuf_rptr0	Repeater	1	sctag_scbuf_rptr0	cmp	L2 repeater

TABLE 2-1 OpenSPARC T1 Top-Level Modules *(Continued)*

Module Name	Type	Number of Instances	Instance Names	Directory Location under `$DV_ROOT/design/sys/iop`	Description
`sctag_scbuf_rptr1`	Repeater	1	sctag_scbuf_rptr1	`cmp`	L2 repeater
`sctag_scbuf_rptr2`	Repeater	1	sctag_scbuf_rptr2	`cmp`	L2 repeater
`sctag_scbuf_rptr3`	Repeater	1	sctag_scbuf_rptr3	`cmp`	L2 repeater
`spc_pcx_buf`	Repeater	8	buf_pcx_[0-7]	`sparc`	Buffer
`bw_clk_gl`	Clock	1	bw_clk_gl	`analog`	Global clock distribution and buffers
`bw_clk_gl_rstce_rtl`	Clock	1	flop_rptrs	`analog`	Global clock buffers and repeaters

2.3 Megacells

The OpenSPARC T1 design contains many megacells, which are custom blocks for static random access memory (SRAMs), translation lookaside buffer (TLB), TAGs, Level 2 Cache, and so on. These megacells are instantiated in the top-level clusters. The detailed descriptions of all megacells, including their function descriptions, I/O lists, block diagrams, and timing diagrams, are in the *OpenSPARC T1 Megacell Specification*.

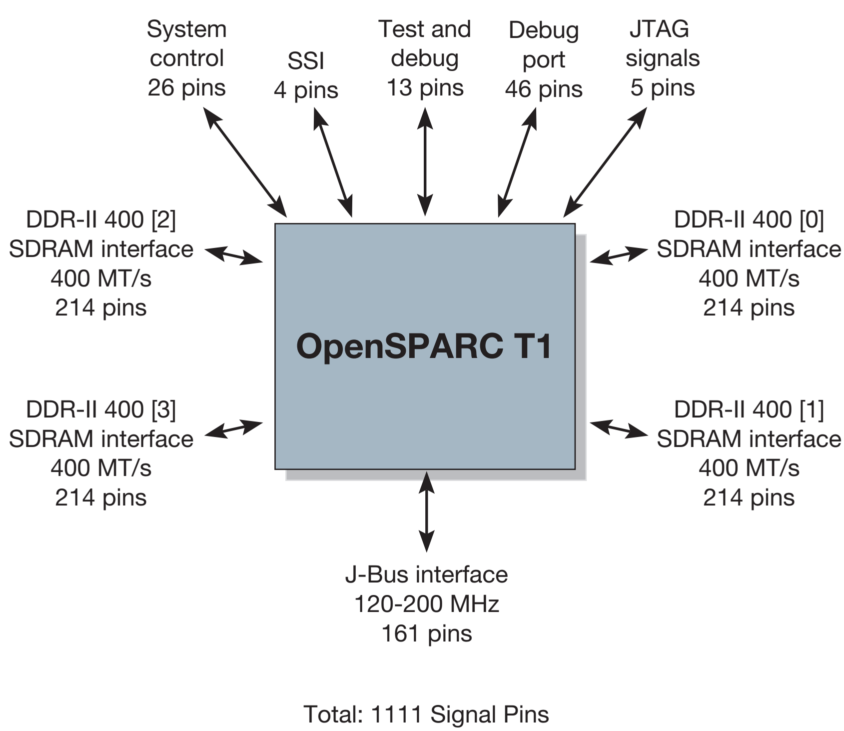
2.4 External Interfaces

The OpenSPARC T1 processor has the following external interfaces:

- Four DDR-II interfaces
- J-Bus
- SSI - Serial System Interface
- JTAG - IEEE 1149.1 interface
- System control
- Test and debug
- Debug port

The block diagram of external interfaces is shown in FIGURE 2-2.

FIGURE 2-2 OpenSPARC T1 External Interfaces



CHAPTER 3

OpenSPARC T1 Verification Environment

This chapter describes the following topics:

3.1 OpenSPARC T1 Verification Environment

The OpenSPARC T1 verification environment is a highly automated environment. With a simple command, you can run the entire regression suite for the OpenSPARC T1 processor, containing thousands of tests. With a second command, you can check the results of the regression.

The OpenSPARC T1 Design and Verification package comes with two test bench environments: `core1` and `chip8`.

The `core1` environment consists of:

- One SPARC CPU core
- Cache
- Memory

- Crossbar

The `core1` environment does not have an I/O subsystem.

The `chip8` environment consists of:

- A full OpenSPARC T1 chip, including all eight cores
- Cache
- Memory
- Crossbar
- I/O subsystem

OpenSPARC T1 Release 1.4 includes a third regression environment for single-thread implementation of the OpenSPARC T1 core. This regression environment has all the components present in `core1` except that it only supports one hardware thread and removes the Stream Processing Unit (SPU). This implementation is primarily developed to create a core with a foot-print amenable for the FPGA map. You can add back SPU into the design by disabling FPGA_SYN_NO_SPU flag during design compile time.

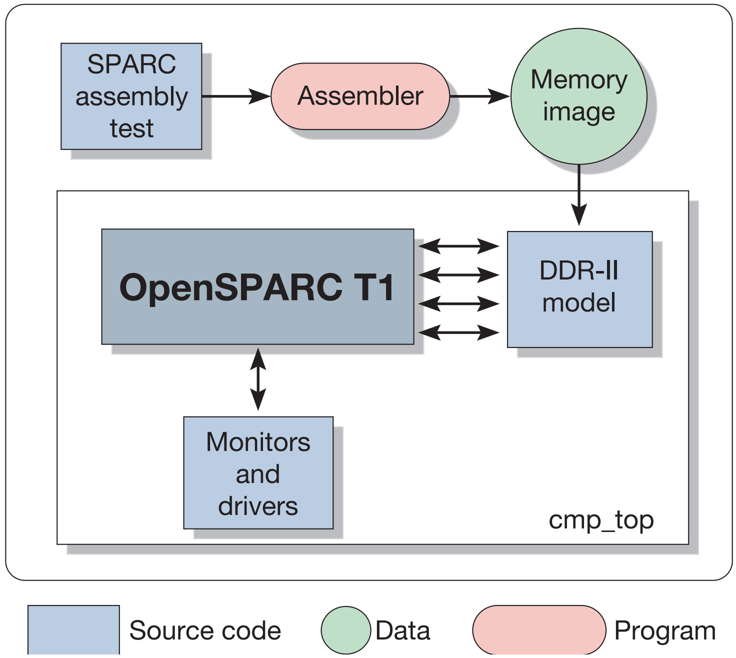
The verification environment uses source code in various languages. TABLE 3-1 shows a summary of the types of source code and their uses.

TABLE 3-1 Source Code Types in the Verification Environment

Source Code Language	Used for:
Verilog	Chip design, test bench drivers, and monitors.
Vera	Test bench drivers, monitors, and coverage objects. Use of Vera is optional.
PERL	Scripts for running simulations and regressions.
C and C++	PLI (Programming Language Interface) for Verilog.
SPARC Assembly	Verification tests.

The block diagram for the verification test bench is in FIGURE 3-1.

FIGURE 3-1 OpenSPARC T1 Verification Test Bench Overview



The top-level module for the test bench is called `cmp_top`. The same test bench is used for both the `core1` and `chip8` environments with compile-time options.

3.2 Running a Regression

For each environment, there is a mini-regression and a full regression. TABLE 3-2 describes the regression groups.

TABLE 3-2 Details of Regression Groups

Regression Group name	Environment	No. of Tests	Disk space needed to run (Mbyte)
thread1_mini	`thread1`	42	25
thread1_full	`thread1`	605	900
core1_mini	`core1`	68	41

TABLE 3-2 Details of Regression Groups *(Continued)*

Regression Group name	Environment	No. of Tests	Disk space needed to run (Mbyte)
core1_full	`core1`	900	1,680
chip8_mini	`chip8`	492	1,517
chip8_full	`chip8`	3789	29,000

3.2.1 To Run a Regression

1. **Run the `sims` command with your chosen parameters.**

   For instance, to run a mini-regression for the `core1` environment using the VCS simulator, set up the `sims` command as follows:

```
% sims -sim_type=vcs -group=core1_mini
```

   To run regressions on multiple groups at the same time, specify multiple `-group=` parameters at the same time. For a complete list of command-line options for the `sims` command, see Appendix A.

2. **Run the `regreport` command to get a summary of the regression.**

```
% regreport $PWD/2006_01_25_0 > report.log
```

3.2.2 What the `sims` Command Does

When running a simulation, the `sims` command performs the following steps:

1. Compiles the design into the `$MODEL_DIR/core1` or `$MODEL_DIR/chip8` directory, depending on which environment is being used.

2. Creates a directory for regression called *$PWD/DATE_ID*, where *$PWD* is your current directory, *DATE* is in `YYYY_MM_DD` format, and *ID* is a serial number starting with 0. For example, for the first regression on Jan 25, 2006, a directory called *$PWD*`/2006_01_26_0` is created. For the second regression run on the same day, the last ID is incremented to become *$PWD*`/2006_01_26_1`.

3. Creates a `master_diaglist.`*regression_group* file under the above directory. such as `master_diaglist.core1_mini` for the `core1_mini` regression group. This file is created based on diaglists under the `$DV_ROOT/verif/diag` directory.

4. Creates a subdirectory with the test name under the regression directory created in step 2 above.

5. Creates a `sim_command` file for the test based on the parameters in the diaglist file for the group.

6. Executes `sim_command` to run a Verilog simulation for the test. If the `-sas` option is specified for the test, it also runs the SPARC Architecture Simulator (SAS) in parallel with the Verilog simulator. The results of the Verilog simulation are compared with the SAS results after each instruction.

   The `sim_command` command creates many files in the test directory. Following are the sample files in the test directory:

```
diag.ev         diag.s          l2way.log       perf.log
sas.log.gz      sims.log        symbol.tbl      sim.perf.log
diag.exe.gz     efuse.img       midas.log       sim_command
status.log      sim.log.gz
```

The `status.log` file has a summary of the status, where the first line contains the name of the test and its status (`PASS/FAIL).`

```
Diag: xor_imm_corner:model_core1:core1_full:0    PASS
```

7. Repeats steps 4 to 6 for each test in the regression group.

3.2.3 Running Regression With Other Simulators

To use a Verilog simulator other than VCS or NCVerilog, use following options for the sims command:

`-sim_type="Your simulator name"`

`-sim_build_cmd="Your simulator command to build/compile RTL"`

`-sim_run_cmd="Your simulator command to run simulations"`

`-sim_build_args="Arguments to build/compile"`

`-sim_run_args="Arguments to run simulations"`

You only need to specify the `sim_type`, `sim_build_cmd`, and `sim_run_cmd` options once. You can specify `sim_build_args` and `sim_run_args` multiple times to specify multiple argument options.

3.3 Verification Code

This section outlines Verilog and Vera code structures and locations.

3.3.1 Verilog Code Used for Verification

There are various test bench drivers and monitors written in Verilog. A list of all Verilog modules, the location of the source code, and descriptions is in TABLE 3-3. All verification Verilog files are in the $DV_ROOT/verif/env directory.

TABLE 3-3 OpenSPARC T1 Verification Test Bench Modules

Module Name	Type	Number of instances	Instance Names	Directory Location under $DV_ROOT/verif/env	Description
OpenSPARCT1	Chip	1	iop	$DV_ROOT/design/sys/iop/rtl	OpenSPARC T1 top level
bw_sys	Driver	1	bw_sys	cmp	SSI bus driver
cmp_clk	Driver	1	cmp_clk	cmp	Clock driver
cmp_dram	Model	1	cmp_dram	cmp	DRAM modules
cmp_mem	Driver	1	cmp_mem	cmp	Memory tasks
cpx_stall	Driver	1	cpx_stall	cmp	CPX stall
dbg_port_chk	Monitor	1	dbg_port_chk	cmp	Debug port checker
dffrl_async	Driver	4	flop_ddr[0-3]_oe	$DV_ROOT/design/sys/iop/common/rtl	Flip-flop
err_inject	Driver	1	err_inject	cmp	Error Injector
jbus_monitor	Monitor	1	jbus_monitor	iss/pli/jbus_mon/rtl	J-Bus Monitor
jp_sjm	Driver	2	j_sjm_4, j_sjm_5	iss/pli/sjm/rtl	J-Bus Driver
monitor	Monitor	1	monitor	cmp	Various monitors
one_hot_mux_mon	Monitor	1	one_hot_mux_mon	cmp	Hot mux monitor
pcx_stall	Driver	1	pcx_stall	cmp	PCX stall
sas_intf	SAS	1	sas_intf	cmp	SAS interface
sas_tasks	SAS	1	sas_tasks	cmp	SAS tasks

TABLE 3-3 OpenSPARC T1 Verification Test Bench Modules *(Continued)*

Module Name	Type	Number of instances	Instance Names	Directory Location under $DV_ROOT/verif/env	Description
slam_init	Driver	1	slam_init	cmp	Initialization tasks
sparc_pipe_flow	Monitor	1	sparc_pipe_flow	cmp	SPARC pipe flow monitor
tap_stub	Driver	1	tap_stub	cmp	JTAG driver

3.3.2 Vera Code Used for Verification

Two types of Vera code are included in the OpenSPARC T1 verification environment:

- Test bench driver and Monitor Vera code
- Vera Object coverage Vera code

Vera code is in the `$DV_ROOT/verif/env/cmp/vera` directory. Each object coverage module has a corresponding subdirectory. Following is a list of Vera object coverage modules:

```
cmpmss_coverage    dram_coverage    exu_coverage    fpu_coverage
lsu_coverage       mt_coverage      tlu_coverage    coreccx_coverage
err_coverage       ffu_coverage     ifu_coverage    mmu_coverage
spu_coverage       tso_coverage
```

Object coverage Vera code for `jbi` is in the `$DV_ROOT/verif/env/iss/vera/jbi_coverage` directory. Object coverage Vera code is only used for the `chip8_cov` regression groups.

3.4 PLI Code Used For the Test Bench

Verilog's PLI (Programming Language Interface) is used to drive and monitor the simulations of the OpenSPARC T1 design. There are eight different directories for PLI source code. Some PLI code is in C language, and some is in C++ language. The object libraries for the VCS simulator and NC-Verilog simulator are included for the PLI code in the `$DV_ROOT/tools/SunOS/sparc/lib` directory. TABLE 3-4 gives the details of PLI code directories, VCS libraries, and NC-Verilog libraries.

TABLE 3-4 PLI Source Code and Object Libraries

PLI name	Source code location under `$DV_ROOT`	VCS object library name	NC-Verilog object library name	Description
`iop`	`tools/pli/iop`	`libiob.a`	`libiob_ncv.so`	Monitors and drivers
`mem`	`tools/pli/mem`	`libmem_pli.a`	`libmem_pli_ncv.so`	Memory read/write
`socket`	`tools/pli/socket`	`libsocket_pli.a`	`libsocket_pli_ncv.so`	Sockets to SAS
`utility`	`tools/pli/utility`	`libutility_pli.a`	`libutility_ncv.so`	Utility functions
`common`	`verif/env/iss/pli/common/c`	`libjpcommon.a`	`libjpcommon_ncv.so`	Common PLI functions
`jbus_mon`	`verif/env/iss/pli/jbus_mon/c`	`libjbus_mon.a`	`libjbus_mon_ncv.so`	J-Bus Monitor
`monitor`	`verif/env/iss/pli/monitor/c`	`libmonitor.a`	`libmonitor_ncv.so`	Various
`sjm`	`verif/env/iss/pli/sjm/c`	`libsjm.a`	`libsjm_ncv.so`	J-Bus Driver

VCS object libraries are statically linked libraries (`.a` files) which are linked when VCS compiles the Verilog code to generate a `simv` executable. NC-Verilog object libraries are dynamically loadable libraries (`.so` files) which are linked dynamically while running the simulations.

Makefiles are provided to compile PLI code. There is a `makefile` file under each PLI directory to create a static object library (`.a` file). There is a `makefile.ncv` file under each PLI directory to create a dynamic object library.

3.4.1 To Compile All PLI Libraries

To compile all PLI libraries, run the `mkplilib` script. This script has three options as listed in TABLE 3-5.

TABLE 3-5 Options for the `mkplilib` Script

Option	Used for
vcs	Compiling PLI libraries for VCS
ncverilog	Compiling PLI libraries for NC-Verilog
clean	Deleting all PLI libraries

- **Compile PLI libraries with your chosen option.**

  For example, to compile PLI libraries in VCS, type the following:

```
% mkplilib vcs
```

Either version of this procedure, VCS or NC_Verilog, compiles C/C++ code, creates static or dynamic libraries, and copies them to the `$DV_ROOT/tools/SunOS/sparc/lib` directory.

3.5 Verification Test File Locations

The verification or diagnostics tests (diags) for the OpenSPARC T1 processor are written in SPARC assembly language (the file names have a `.s` extension). Some diags require command files for a J-Bus Driver. Those command files are named `sjm_4.cmd` and `sjm_5.cmd`. Some diagnostics test cases in SPARC assembly are automatically generated by Perl scripts.

The main diaglist for `core1` is `core1.diaglist`. The main diaglist for `chip8` is `chip8.diaglist`. These main diaglists for each environment also include many other diaglists. The locations of various verification test files are listed in TABLE 3-6.

TABLE 3-6 Verification Test File Directories

Directory	Contents
`$DV_ROOT/verif/diag`	All diagnostics, various diagnostic list files with the extension`.diaglist`.
`$DV_ROOT/verif/diag/assembly`	Source code for SPARC assembly diagnostics. More than 2000 assembly test files.
`$DV_ROOT/verif/diag/efuse`	EFuse cluster default memory load files.

3.6 Compiling Source Code for Tools

To compile source code for some Sun tools used for the OpenSPARC T1 processor, use the `mktools` script. The tools source code is located in the `$DV_ROOT/tools/src` directory.

The `mktools` script compiles the source code and copies the binaries to `$DV_ROOT/tools/`<*Operating System*>`/`<*Processor Type*> directory, where:

- <*Operating System*> is defined by the `uname -s` command
- <*Processor Type*> is defined by the `uname -p` command

3.7 Gate-Level Verification

OpenSPARCT1 depends heavily on Cross-Module References (XMRs) within the verification environment. Therefore, dropping in a `netlist` in place of the RTL core will produce a high number of XMR errors. In order to overcome this difficulty, a simple playback support is now added.

Although we anticipate this method to be useful primarily to verify FPGA synthesized `netlists`, it could be potentially used with netlists generated by semi-custom synthesis flows as well (for example, Synopsys).

Caution – Running this vector playback mechanism on RTL, although feasible, is not recommended due to some array initialization issues. In the gate playback mode, all arrays are explicitly initialized to zero while in RTL and some arrays are initialized to random values. This may result in mismatch in playback simulation. If RTL arrays are initialized correctly (zeros) then this mechanism can be used to verify RTL netlist as well.

To verify a `netlist`, do the following:

1. **Run RTL mini or full regression to generate stimuli files for netlist verification.**

   To do this, add the `-vcs_build_args=`
   `$DV_ROOT/verif/env/cmp/playback_dump.v` option to the regression command.

   For example, `thread1_mini` regression command for the SPARC level driver (stimuli) generation would require the following:

```
% sims -sim_type=vcs -group=thread1_mini -debussy \
-vcs_build_args=$DV_ROOT/verif/env/cmp/playback_dump.v
```

   The above regression will generate `stimuli.txt` file under the run directory of each diagnostic. Sample `stimuli.txt` files are included under `$DV_ROOT/verif/gatesim` for the `thread1_mini` regression (file `thread1_mini_stim.tar.gz`). These files are generated with VCS build `args` `(-vcs_build_args)FPGA_SYN`, `FPGA_SYN_1THREAD, FPGA_SYN_8TLB` and `FPGA_SYN_NO_SPU` flags.

2. **Create a verilog file list which includes the following files:**

```
$DV_ROOT/verif/env/cmp/playback_driver.v
<SPARC level gate netlist>.v
<library used for synthesis>.v
```

   Sample `flist` is provided under `$DV_ROOT/verif/gatesim` for reference (file `flist.xilinx_unisims`)

3. **Compile the design to build the gate level model.**

   A sample compile script is provided under the `$DV_ROOT/verif/gatesim` directory. The following shows usage of the compile script:

```
% $DV_ROOT/verif/gatesim/build_gates <flist>
```

4. **Run the simulation by including** `+stim_file=<path` **to** `stim file>/stimuli.txt`

   If playback fails, the simulation will return with "`Playback FAILED with # mismatches!`" If it passes, it will return with "`Playback PASSED!`"

   Use `fsdb` generation options in the compile script to debug failing runs.

   A simple `run_gates` script is also included for reference under the `$DV_ROOT/verif/gatesim` directory. The following shows usage of the run script:

```
% $DV_ROOT/verif/gatesim/run_gates <path to stim file>
```

TABLE 3-7 Gate Netlist Files

Directory/File	Contents
`$DV_ROOT/verif/gatesim/build_gates`	Compile script to create gate level model of SPARC `netlist`
`$DV_ROOT/verif/gatesim/run_gates`	Run script to execute playback of vectors on gate `netlist`
`$DV_ROOT/verif/gatesim/flist.xilinx.unisims`	Sample verilog file list with Xilinx synthesis library
`$DV_ROOT/verif/gatesim/thread1_mini_stim.tar.gz`	FOR REFERENCE ONLY: Collection pre-packaged stimulus files for `thread1_mini` regression suit.

CHAPTER 4

OpenSPARC T1 Synthesis

This chapter describes the following topics:

- Synthesis Flow for the OpenSPARC T1 Processor
- Synthesis Output

The scripts provided in the source code are for the Synopsys Design Compiler.

4.1 Synthesis Flow for the OpenSPARC T1 Processor

There are two types of synthesis scripts:

- One set to run the Synopsys Design Compiler (`rsyn` and `syn_command`)
- One set used as input for the Design Compiler

The main script used to run Synopsys Design Compiler is called `rsyn`. This is a PERL script that calls a second script, `syn_command`, once for each module you are synthesizing. The command-line options for the `rsyn` script are described in CODE EXAMPLE 4-1.

CODE EXAMPLE 4-1 Command-Line Options for `rsyn` Script

```
rsyn : Run Synthesis for OpenSPARC T1

     -all
         to run synthesis for all blocks
     -h / -help
         to print help
     -syn_q_command='Your job Queue command'
         to specify Job queue command. e.g. specify submit command
         for LSF or GRID
     block_list :
         specify list of blocks to synthesize

     Examples:

     rsyn -all
     rsyn fpu_add
```

Synthesis scripts for most of the modules are provided in the `$DV_ROOT/design` sub-directories. There are no synthesis scripts for the following types of modules:

- Megacell modules (SRAMS, TLB, TAG, Cache, etc.)
- Top-level hierarchical modules

Synopsys scripts, their locations, and their descriptions are listed in TABLE 4-1.

TABLE 4-1 Synthesis Script Details

Script name	Location	Description
`run.scr`	`$DV_ROOT/design/sys/synopsys/script`	Main synthesis script that calls `user_cfg.scr`
`project_sparc_cfg.scr`	`$DV_ROOT/design/sys/synopsys/script`	SPARC module-specific synthesis script
`project_io_cfg.scr`	`$DV_ROOT/design/sys/synopsys/script`	I/O module-specific synthesis script
`target_lib.scr`	`$DV_ROOT/design/sys/synopsys/script`	Target library-specific script
`user_cfg.scr`	*Module directory*`/synopsys/script`	Module-specific synthesis script

The top-level Synopsys script, run.scr, calls the module-specific script named user_cfg.scr. The user_cfg.scr script calls the project_sparc_cfg.scr script or the project_io_cfg.scr script, depending on whether the module belongs to sparc or io.

The list of all modules with synthesis scripts is in the $DV_ROOT/design/sys/synopsys/block.list file. Each module has:

- A synopsys directory under the module directory
- A script directory under each synopsys directory
- The user_cfg.scr file under the script directory

For example, the efc module-specific synthesis script has the following directory path:

```
$DV_ROOT/design/sys/iop/efc/synopsys/script/user_cfg.scr
```

The target library is set to a generic library called lsi_10k.db in the target_lib.scr script. Modify this file to set your own target library and its required variables.

4.2 Synthesis Output

Running synthesis for a module creates files and directories under the *Module name*/synopsys directory, described in TABLE 4-2.

TABLE 4-2 Synthesis Output

Name	Type	Description
dc_shell.log	File	Log file from running Design Compiler
command.log	File	Command log from running Design Compiler
log	Directory	Area report files from Design Compiler
gate	Directory	Gate netlist generated by Design Compiler
.template	Directory	Template directory used by Design Compiler

CHAPTER 5

OpenSPARC T1 FPGA Synthesis

This chapter describes the following topics:

- Synplicity FPGA Synthesis Flow for the OpenSPARC T1 Processor
- Synplicity FPGA Synthesis Output
- XST Synthesis Flow for the OpenSPARC T1 Processor
- XST Synthesis Output
- Selecting OpenSPARC T1 Options for Reduced Size

The scripts provided in the OpenSPARC T1 source code are for the Synplicity Pro software, version 8.5, and for the Xilinx Synthesis Technology (XST) Software, version 9.1.

5.1 Synplicity FPGA Synthesis Flow for the OpenSPARC T1 Processor

Several scripts are required to run FPGA synthesis with Synplicity

- Shell scripts to run the Synplicity Pro software (`rsynp` and `synp_command`)
- Synplicity scripts which are read by the Synplicity Pro software

The main script used to run the Synplicity software is a PERL script called `rsynp`. `rsynp` calls a second script, `synp_command`, once for each module you are synthesizing into an FPGA. CODE EXAMPLE 5-1 describes the command-line options of the `rsynp` script.

CODE EXAMPLE 5-1 Command-Line Options for the `rsynp` Script

```
rsynp : Run Synplicity for OpenSPARC T1

Options are :
  -all
        to run Synplicity for all blocks
  -h / -help
        to print help
  -syn_q_command='Your job Queue command'
        to specify Job queue command
  -device='Target Device'
        to specify Target FPGA device
  -flat
        To run synthesis flat, must use this for Altera parts.
  -clean
        To remove all unneeded files and/or directories.
        Need to specify target device when not using default device

  block_list :
        specify list of blocks to synthesize

Examples:

        rsynp -all
        rsynp -device=XC4VLX200 sparc
        rsynp -flat -device=EP2S180 sparc
```

The OpenSPARC T1 source code provides FPGA synthesis scripts for the following modules: `sparc`, `fpu`, and `ccx`. The `rsynp` script first creates a module-specific script `proj.prj` in the <*module-directory*>`/synplicity` directory, and then it calls the Synplicity software in batch mode using the `proj.prj` script as input. TABLE 5-1 lists the FPGA Synplicity scripts, their locations, and their descriptions. The `proj.prj` script uses all the scripts listed in this table.

TABLE 5-1 Synplicity FPGA Synthesis Script Details

Script name	Location	Description
`block.list`	`$DV_ROOT/design/sys/synplicity`	List of blocks with synthesis scripts

TABLE 5-1 Synplicity FPGA Synthesis Script Details *(Continued)*

Script name	Location	Description
`pre_syn_settings.prj`	`$DV_ROOT/design/sys/synplicity`	Synplicity software settings
`env.prj`	`$DV_ROOT/design/sys/synplicity`	OpenSPARC environment-related settings
<*device*>`.prj`	`$DV_ROOT/design/sys/synplicity`	Target device-specific settings (for example, the XC4VLX200.prj script would be for the Xilinx XC4VLX200 device)

The `$DV_ROOT/design/sys/synplicity/block.list` file lists all of the modules that can be synthesized using the Synplicity software. Each module has:

- A `synplicity` directory under the module (<*module*>) directory.
- A <*module*>`.flist` file under each <*module*>`/synplicity` directory. (This file lists the Verilog files for the module.)
- Optional <*module*>`.mlist` file under <*module*>`/synplicity` directory, this is the list of SRAM modules for the module. SRAM modules are synthesized hierarchically.
- Optional <*module*>`.fmlist` file under <*module*>`/synplicity` directory, this is the flat list of SRAM Verilog files.
- Optional <*module*>`.prj` file under <*module*>`/synplicity` directory, this is the file for module specific Synplicity settings.
- Optional <*module*>`.sed` file to change names of the modules to make them unique module names in the Synplicity-generated output design `.vm` file.

5.2 Synplicity FPGA Synthesis Output

While running a FPGA synthesis for a module, the Synplicity software will create files and directories under the <*module*>/`synplicity` directory. TABLE 5-2 describes these files and directories.

TABLE 5-2 Synplicity FPGA Synthesis Output

Name	Type	Description
<*device*>	Directory	Target device-specific directory (for example, XC4VLX200 would be the directory name for the Xilinx XC4VLX200 device). The Synplicity software will create a number of files and sub-directories under this <*device*> directory.
<*device*>/<*module*>`.srr`	File	Synplicity software output file for a <*module*> using the <*device*> as a target FPGA. This file is a log of the synthesis process, and contains information about the estimated timing, area, and so on.
<*device*>/<*module*>`.edf`	File	EDIF netlist for the synthesized block.

5.3 XST Synthesis Flow for the OpenSPARC T1 Processor

Scripts are now available to allow automated synthesis of the OpenSPARC T1 using Xilinx Synthesis Technology (XST).

The main script used to run the XST software is a PERL script called rxil.

The rxil script calls XST once for each module that is being synthesized into an FPGA.

CODE EXAMPLE 5-2 describes the command-line options of the rxil script.

CODE EXAMPLE 5-2 Command-Line Options for the rxil Script

```
rxil : Run XST for OpenSPARC T1
Options are :
-all
  to run XST for all blocks
-h / -help
  to print help
-device='Target Device'
  to specify Target FPGA device
block_list :
  specify list of blocks to synthesize
Examples:
  rxil -all
  rxil -device=XC4VLX200 sparc
  rxil -device=XC4VLX200 fpu ccx
```

The OpenSPARC T1 source code provides XST synthesis scripts for the followingmodules: sparc, fpu, and ccx. The rxil script first copies a device file for the target device into the <*module-directory*>/`xst` directory, creates a <device> subdirectory, and then it calls the XST software in batch mode using the device file and the <block>.flist file as input.

TABLE 5-3 XST Synthesis Script Details

Script Name	Type	Description
`block.list`	`$DV_ROOT/design/sys/xst`	List of blocks which may be synthesized.
`XC4VFX100.xst`	`$DV_ROOT/design/sys/xst`	One of the device files for XST. This one is for a Xilinx Virtex-4 part: XC4VFX100
`xst_defines.h`	`$DV_ROOT/design/sys/iop/include`	Sets top-level defines for XST synthesis. Allows selection of different OpenSPARC T1 options.

The `$DV_ROOT/design/sys/xst/block.list` file lists all of the modules that can be synthesized using the XST software. Each module has:

- A xst directory under the module (<*module*>) directory.
- A <*module*>.flist file under each <*module*>/xst directory. (This filelists the Verilog files for the module.)

5.4 XST Synthesis Output

While running a FPGA synthesis for a module, the XST software will create files and directories under the <module>/xst directory. TABLE 5-4 describes these files and directories.

TABLE 5-4 FPGA Synthesis Output

Name	Type	Description
<device>	Directory	Target device specific directory (for example, XC4VLX200 would be the directory name for the Xilinx XC4VLX200 device). The XST software will create a number of files and subdirectories under this <device> directory.
<device>/*<block>*`.ngc`	File	Synthesized block netlist in Xilinx NGC format.
<device>/*<block>*`.v`	File	Synthesized block netlist in Verilog format.
<device>/*<device>*`.srp`	File	XST Synthesis log. Contains FPGA utilization and timing information.

5.5 Selecting OpenSPARC T1 Options for Reduced Size

OpenSPARC T1 RTL now includes four conditional compile options in the design. Each of these options create a different variant of the OpenSPARC T1 core. Although these four options are orthogonal to each other in terms of what change it brings to the design, when combined together, they produce a significantly smaller design and a very compelling solution for FPGA mapping. The following describes these options:

- `FPGA_SYN`: This option enables different implementation for some of the megacells and SRAM arrays to make it more effectively utilize FPGA resources like Block RAMs and multipliers. This option also removes all asynchronous logic (for example, latches) from the design to make it amiable for FPGA synthesis. FPGA synthesis will not complete without this option.
- `FPGA_SYN_1THREAD`: This option reduces multi-threaded overhead of logic and creates a small and clean single thread implementation of OpenSPARC T1 core. The basic functionality of the SPARC core here is the same.

- FPGA_SYN_NO_SPU: The Stream Processing Unit (SPU) in OpenSPARC T1 provides hardware acceleration for cryptographic functions. In designing general purpose FPGA based processor, this unit can be safely removed without affecting the base functionality. This option removes SPU from the design and provides further reduction in the design size.
- FPGA_SYN_8TLB: The OpenSPARC T1 TLB normally has 64 entries. To save space,a reduced-size TLB of 8 entries may be selected. The function of the TLB is remains the same. It just has fewer entries.
- FPGA_SYN_16TLB: This option can be used instead of the FPGA_SYN_8TLB option to create a design with 16 TLB entries instead of 8. This improves performance of the design, at the cost of area.
- CONNECT_SHADOW_SCAN: The shadow scan chain is used to scan out system state information for debug purposes. Normally, it is not connected in the RTL because the connection was performed by the regular scan connection tools. This option connects the shadow scan elements in the RTL. This prevents the shadow scan registers from being optimized away during FPGA synthesis, and eliminates the need for a scan connection step in an FGPA environment, where regular scan chains are not used.

Based on our experiments, combining the first four options create a design that would consume about 40,000 4-input Look-Up Tables (LUT) in the Xilinx Virtex-4 family.

To run the FPGA synthesis with Synplicity Simplify Pro,

1. **Add the following line in the file.**

```
$DV_ROOT/design/sys/synplicity/env.prj
```

```
# Set +defines
set_option -hdl_define -set "FPGA_SYN FPGA_SYN_NO_SPU
FPGA_SYN_1THREAD FPGA_SYN_8TLB"
```

2. **Then run** `rsynp`**.**

   For `rsynp` script details, see Section 5.3, "XST Synthesis Flow for the OpenSPARC T1 Processor" on page 5.

   To run the FPGA synthesis with Xilinx XST,

1. **Add the following lines in the file**

   $DV_ROOT/design/sys/iop/include/xst_defines.h

```
`define FPGA_SYN
`define FPGA_SYN_NO_SPU
`define FPGA_SYN_1THREAD
`define FPGA_SYN_8TLB
```

2. **Then run rxil.**

   For rxil script details, see Section 5.3, "XST Synthesis Flow for the OpenSPARC T1 Processor" on page 5.

Caution – The four options described above should be used *only* with the thread1_mini and thread1_full regression environments. Attempting to include these options in core1 regressions will cause the diagnostics to fail. core1 regression includes tests that verify multi-threaded and SPU related functionality which does not exist when the model is built with the above options.

CHAPTER 6

OpenSPARC T1 EDK Project

This chapter describes the following topics:

- System Description of the EDK Project
- Generation of a Bit File for a Xilinx FPGA
- Running OpenSPARC T1 Diagnostics on an Evaluation Board
- Running a Standalone Program on the OpenSPARC T1 Core on an Evaluation Board
- Booting OpenSolaris on an FPGA Evaluation Board
- Booting Ubuntu Linux on an FPGA Evaluation Board
- Dual-Core System EDK Project
- Running System-level Simulation on Legacy Projects

6.1 System Description of the EDK Project

A Xilinx Embedded Development Kit (EDK) project is included to provide a platform for running the OpenSPARC T1 core on a Xilinx ML505-V5LX110T or ML411 evaluation board. The Xilinx EDK environment allows the user to quickly put a new system together, place and route the design on a target FPGA, then download the design to the target FPGA to test and debug it.

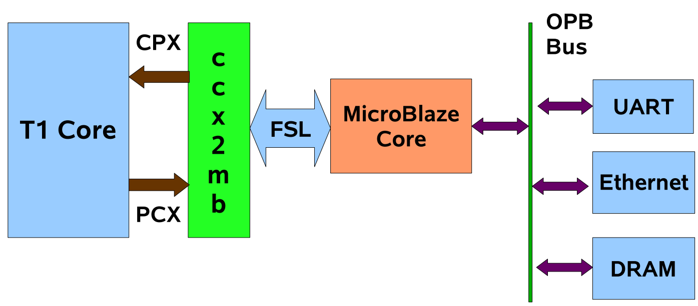
A block diagram of the system provided in the EDK project is shown in FIGURE 6-1.

The system contains the following components.

- OpenSPARC T1 core.
- An Adapter block which adapts the cache crossbar interface of the OpenSPARC T1 core to the MicroBlaze FSL interface.

- A MicroBlaze controller which will run software to service all memory and I/O requests coming from the core. The MicroBlaze core contains:
  - 16k I-cache
  - 16k D-cache
- DDR2 SDRAM controller. This controller accesses the 256 MB SODIMM on the ML505 board or the 256 MB DIMM on the ML411 board.
- UART Lite controller to communicate with a host PC over a serial interface.
- A network controller to allow the board to be connected to the network.

FIGURE 6-1 EDK OpenSPARC T1 System Block Diagram



6.1.1 Hardware Operation

The EDK system was designed for minimal overhead on top of the core. This makes it possible to fit the core into a fairly small FPGA. The system operates as follows:

Memory accesses from the core (instruction fetches, loads, and stores) are sent over the 124-bit processor-to-cache crossbar (PCX) interface. Interrupts are also communicated through this interface. The ccx2mb adapter block breaks these 124-bit requests into four 32-bit words and places those words into an FSL FIFO. Firmware running on the MicroBlaze core reads the requests out of the FSL FIFO, interprets them, and services them. Once the firmware has completed the request, it generates a 144-bit response packet, which is placed in a second FSL FIFO as 5 32-bit words. The ccx2mb block reads these words from the FIFO, reconstructs the 144-bit response, and forwards it on to the OpenSPARC T1 core.

6.1.2 MicroBlaze Firmware Operation

The firmware must perform the following functions:

- Interpret and service the memory requests coming from the OpenSPARC T1 core.
- Maintain a memory map which maps OpenSPARC T1 addresses to addresses in the MicroBlaze memory system.
- Maintain directories of the level 1 instruction and data caches of the OpenSPARC T1 core. This is required in order to keep these caches coherent.
- Send invalidations to the level 1 caches to keep them coherent.
- Send the initial wake-up interrupt to the OpenSPARC T1 after reset completes.
- Communicate device interrupts to the OpenSPARC T1 core.

The firmware operates by continuously polling the FSL FIFO connected to the OpenSPARC T1 PCX interface. When there is a valid packet, the firmware decodes it and services the request. For any kind of memory transaction, the firmware must translate the OpenSPARC T1 address to the corresponding board address. Once the correct board address is obtained, the firmware can get data from or store data to memory. Then it generates the appropriate packets to send back to the OpenSPARC T1 core.

There are two variations of the firmware: One to run OpenSPARC T1 diagnostic tests, and one to run standalone programs under hypervisor. The only difference between these variations is the memory mapping of OpenSPARC T1 addresses to board addresses.

6.2 Generation of a Bit File for a Xilinx FPGA

The OpenSPARC T1 EDK project should be ready to go. To generate a bitstream for a Xilinx FPGA, there is a simple two-step process:

1. **Copy the correct OpenSPARC T1 netlist into the pcores directory if necessary.**

   By default, there is a Synplicity netlist of a four-thread core in the netlist directory for the OpenSPARCT1. The provided netlist was synthesized with the following features:

- Four Threads
- 16-entry TLB

- No stream processing unit (SPU)

  To use an XST netlist instead, the Synplicity netlist must be removed and the XST netlist placed in the netlist directory in its place. There can only be one netlist in the netlist directory. The text box shows how to replace the Synplicity netlist with the XST netlist.

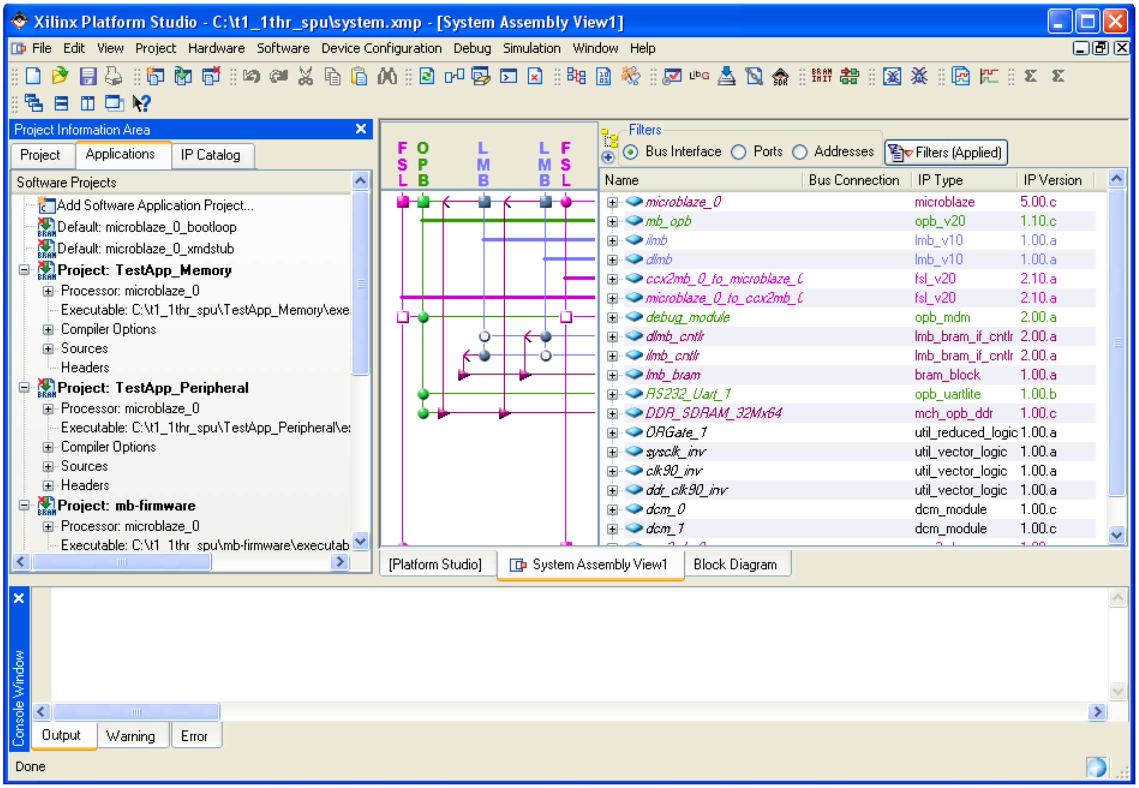
```
% cd $DV_ROOT/design/sys/edk/pcores/iop_fpga_v1_00_a
% mv netlist/sparc.edf .
% cp path-to-netlist/sparc.ngc netlist
```

  After the new netlist has been put in place, the file `data/iop_fpga_v2_1_0.bbd` must be edited to point to the new netlist.

2. **Start Xilinx Platform Studio (XPS).**

   Xilinx Platform Studio will come up as shown in FIGURE 6-2.

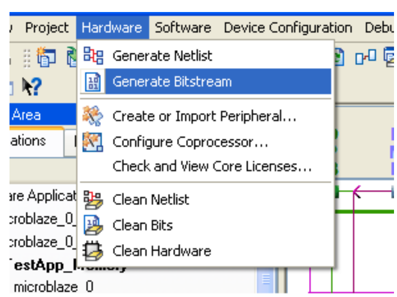
FIGURE 6-2 Xilinx Platform Studio (XPS) Window with OpenSPARC Project Loaded



3. **Open File $DV_ROOT/design/sys/edk/system.xmp.**

4. **Select the menu option to generate a bitstream.**

   Select the menu item:
   Hardware --> Generate Bitstream



6.3 Running OpenSPARC T1 Diagnostics on an Evaluation Board

Once a bitstream has been generated, it can be downloaded to the FPGA on the evaluation board, and SPARC diagnostic tests (called diags at Sun) can be run on the hardware.

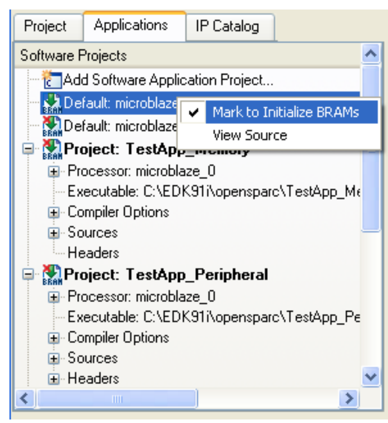
The version of the firmware to run stand-alone diagnostics requires a source file, `mbfw_diag_memimage.c`, which is a C structure representation of the memory image for the test. To run any diag, the assembly language program for that test must be assembled into an executable. Then the executable is converted to a binary file, and a perl script is run to convert the binary file into the C structure representation that is needed by the firmware. Then the firmware is re-compiled with the new C structure. The EDK project relies on the `sims` program to generate the executable programs, since it calls the assembler with all the proper options for each test.

The `ccx-firmware-diag` software project is set up to run the *bypass_win* test by default. To run this test, perform the following procedure:

6.3.1 Running the Default Diag: *bypass_win*

1. **Select the Applications tab in the Project Information Area (the left-hand window).**

   A list of software applications will be shown.

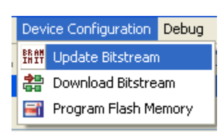


2. **Right-click on the `microblaze_0_bootloop` application and select "Mark to initialize BRAMs."**

   This will select the `microblaze_0_bootloop` program to be automatically downloaded to BRAMs with the bitstream. This will keep the MicroBlaze processor safely looping at address 0 until the `ccx-firmware-diag` program is downloaded to the DRAM. Make sure no other program is marked to initialize BRAMs

3. **Select the following menu item to update the bitstream.**

   Select menu Device Configuration --> Update Bitstream.



4. **Download the bitstream to the FPGA.**

   Select menu Device Configuration --> Download Bitstream.

5. **Make sure the Firmware executable is up to date.**

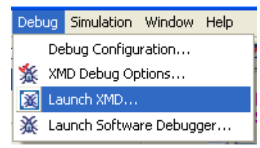
   Right-click on the `ccx-firmware-diag` software project and select `build` in the popup menu.

6. **Start any terminal window, such as Hyperterminal, and connect to the serial port that is connected to the FPGA board.**

   The communication settings should be set to 9600 baud, data 8 bits, parity none, stop bits 1, and flow control none.

7. **Launch XMD.**

   Select menu Debug --> Launch XMD or click the button to launch XMD from the XPS GUI.



8. **Download the firmware executable file.**

```
XMD% dow ccx-firmware-diag/executable.elf
```

9. **Run the diagnostic test.**

```
XMD% run
```

   The output from the test will appear in the terminal window.

```
MBFW_INFO: Running RTL diag "bypass_win"
MBFW_INFO: Microblaze firmware initialization completed.

MBFW_INFO: Powering on OpenSPARC T1
MBFW_INFO: speculative_ifill_data being returned for 0x1000144020
MBFW_INFO: received ifill request for good trap addr:  0x1000122000
MBFW_INFO: Thread 0 reached good trap.
MBFW_INFO: All threads reached good trap.
```

6.3.2 Running other Diags on the FPGA Board

To run other diags on the FPGA board, the assembly language program for the desired diag must be assembled, and the resulting executable must be converted into a C structure which is compiled into the `ccx-firmware-diag` application.

1. **Run sims to generate the desired test executable.**

   When running diags on the FPGA, the source code for the test must be compiled with a different set of compiler options than when it is run in the simulation environment. Therefore to generate the memory image, sims must be run with the following options.

   ```
   sims -sys=core1 -novcs_build -novera_build -midas_only
        -midas_args='-DFPGA_HW -DCIOP -DNO_SLAM_INIT_SPC'
   ```

   This calls the assembler to create an executable from the assembly language source of the test. The resulting executable contains some simple reset code, trap tables, as well as the main diagnostic code. The `sims` program then takes this executable and creates a memory image file from it. All the code and data sections from the executable are placed into the memory image file. Then the required page table entries and translation storage buffers for virtual memory are added.

2. **Run the genmemimage.pl script to create a C source file representation of the executable.**

   ```
   % $DV_ROOT/design/sys/edk/scripts/genmemimage.pl
     -single -f memory-image-file -name test_name
   ```

   This program takes the `mem.image` file created by `sims` and converts it into a C structure that can be compiled with the MicroBlaze firmware.

3. **Copy the output C file to the firmware source directory.**

   ```
   % cp mbfw_diag_memimage.c ccx-firmware-diag/src
   ```

4. **Recompile the ccx-firmware-diag project.**

   Right-click on the `ccx-firmware-diag` software project and select "Build Project."

5. **Follow the steps in the previous section to run the diag on the FPGA board.**

6.3.3 Running an Entire Regression on the FPGA Board

Scripts are included to allow the user to run an entire regression on the FPGA board. These scripts automatically copy the C structure representation of each executable to the `ccx-firmware-diag` source directory, re-compile the `ccx-firmware-diag` code, download the bitstream to the FPGA, and run the test. The results of each test are recorded, and a summary is generated when the regression is completed. Here is the procedure to run a complete regression:

1. **Run `sims` to generate memory image files for a complete regression group.**

   The applicable regressions that may run on a one-thread core are `thread1_mini` and `thread1_full`. For a four-thread core, the applicable regressions are `core1_mini` and `core1_full`.

   The `sims` program will generate memory image files for all the tests included in the regression. The program is run with options to simply generate the test executable, without calling the simulator. The sims program will create a directory for each test which includes the executable for that test.

```
% sims -novcs_build -novera_build -group=thread1_mini -copyall
   -midas_only -midas_args='-DFPGA_HW -DCIOP -DNO_SLAM_INIT_SPC'
```

   The above command generates memory image files for all the tests in the thread1_mini regression, compiled with the proper options to run on the FPGA board.

2. **Run the genmemimage.pl script to generate C structure representations of all the tests in the regression.**

```
% $DV_ROOT/design/sys/edk/scripts/genmemimage.pl -d
regression-dir
```

   This will create a directory called `diags` which contains all the output files. There will be one file for every test in the regression.

3. **Edit the file `thread1_full.list`, `thread1_mini.list`, `core1_mini.list`, or `core1_full.list` in the EDK project.**

   These files are lists of tests which will be run as a set. A subset of the tests in any of the above files may be selected by deleting or commenting out undesired tests in the file.

Note – Tests which access the SPU are commented out in the `core1_mini` and `core1_full` regressions. If a core is generated with an SPU, then these tests may be un-commented.

---

4. **Run the regression.**

   The following command will run the entire regression. It will re-compile the firmware code with each new memory image, then download the design to the FPGA, then run the test on the hardware. The test is most easily run from the project directory.

```
% xtclsh edk-project-dir/scripts/rundiags.tcl -edk edk-project-dir
     -d regression-path -list
     edk-project-dir/scripts/core1_mini.list -model core1
     -suite core1_mini
```

6.4 Running a Standalone Program on the OpenSPARC T1 Core on an Evaluation Board

A second firmware set-up is provided to allow the running of a stand-alone program under hypervisor. In this set-up, the OpenSPARC T1 core will boot hypervisor, which will then branch to a stand-alone program. A very simple "Hello World" program is included with the project to demonstrate this. This set-up may also be used to boot an operating system under hypervisor.

The memory map of the system is shown in FIGURE 6-3. One megabyte of the 256 MB memory is allocated for the MicroBlaze firmware code. Another megabyte is allocated for the OpenSPARC boot PROM image. The remaining 254MB is split between memory space for the OpenSPARC T1 core and a RAM disk image that contains the executable. The firmware translates the OpenSPARC T1 addresses to board addresses as shown in FIGURE 6-3.

FIGURE 6-3 Allocation of the ML505-V5LX110 256-MB DRAM with the Hypervisor Firmware Set-up

MicroBlaze Address	Function
0x8000_0000	MicroBlaze Firmware Code (1MB)
0x8010_0000 – 0x8aef_ffff	OpenSPARC Memory Space: (174 MB) Addresses: 0x00_0000_0000 – 0x00_0fdf_ffff
0x8af0_0000 – 0x8fef_ffff	RAM Disk Image (80 MB)
0x8ff0 0000	OpenSPARC boot PROM Image: 0xff_f000_0000 (1MB)

6.4.1 Running the Included "Hello World" Program

An example "Hello World" program is included in the EDK project. The program may be run under hypervisor by performing the following procedure:

1. Locate the proper boot PROM image file.

The prom.bin file contains an image of the boot PROM for the system. This file contains the reset code, hypervisor code, and the Open Boot PROM (OBP) code. Eight prom.bin files are included in the hardware package. They are located in the following directory:

```
$DV_ROOT/design/sys/edk/os/proms
```

The purpose of each file is shown in TABLE 6-1.

TABLE 6-1 OpenSPARC T1 prom.bin files

File	Purpose
1clt_prom.bin	One-core, one-thread system running stand-alone programs under hypervisor
1c4t_prom.bin	One-core, four-thread system running stand-alone programs under hypervisor
1clt_obp_prom.bin	One-core, one-thread system booting OpenSolaris from OBP

TABLE 6-1 OpenSPARC T1 prom.bin files *(Continued)*

File	Purpose
1c4t_obp_prom.bin	One-core, four-thread system booting OpenSolaris from OBP
2c1t_prom.bin	Two-core, one-thread system running stand-alone programs under hypervisor
2c4t_prom.bin	Two-core, four-thread system running stand-alone programs under hypervisor
2c1t_obp_prom.bin	Two-core, one-thread system booting OpenSolaris from OBP
2c4t_obp_prom.bin	Two-core, four-thread system booting OpenSolaris from OBP

2. **Compile the application program.**

   A pre-built memory image of the “Hello World” program is included in the EDK project. However, if it is desired to change the program, make scripts are included to make it easy to re-compile the example program. The make must be run on a SPARC machine with SunStudio compilers.

Note – The following commands will require one to download and install OpenSPARC T2 architecture bundle. You can download this bundle from:

`http://www.opensparc.net/opensparc-t2/downloads.html`

To compile the code, execute the following commands:

```
% setenv SUN_STUDIO path-to-SunStudiio-Compilers
% setenv QTOOLS $SAM_ROOT/hypervisor/src/hypervisor-tools

% cd design/sys/edk/examples/src
% make install
```

3. **Compress the application program.**

   The MicroBlaze firmware assumes that the application program has been compressed with `gzip`. This allows for faster downloading to the board, especially when the application program is quite large. To compress the application programs, run the following command:

```
% gzip design/sys/edk/examples/bin/hello_world.mem.image
% gzip design/sys/edk/examples/bin/l2_emul_test.mem.image
```

4. **Compile the MicroBlaze firmware for standalone programs.**

   In the XPS user interface right-click on the `ccx-firmware` project in the Applications window and select Build.

5. **Make sure that the `microblaze_0_bootloop` software project is set to initialize BRAMs.**

   Right-click on the `microblaze_0_bootloop` project in the Applications window and select "Mark to initialize BRAMs."

6. **Select the following menu item to update the bitstream.**

   Device Configuration --> Update Bitstream

7. **Download the bitstream to the FPGA.**

   Device Configuration --> Download Bitstream

8. **Start any terminal window, such as Hyperterminal, and connect to the serial port that is connected to the FPGA board.**

9. **Start XMD.**

   Select menu Debug --> Launch XMD or click the button on the XPS GUI.

10. **Download the firmware program to the FPGA.**

```
XMD% dow ccx-firmware/executable.elf
```

11. **Download the binary image of the code for the sample OpenSPARC T1 program to DRAM.**

```
XMD% dow -data examples/bin/hello_world.mem.image.gz 0x8af00000
```

12. **Download the reset and hypervisor code for the OpenSPARC T1 to DRAM.**

```
XMD% dow -data os/OpenSolaris/proto/1c4t_prom.bin 0x8ff00000
```

13. **Run the program.**

```
XMD% run
```

The output of the program will appear in the terminal window. CODE EXAMPLE 6-1 shows the hypervisor output, followed by the output of the "Hello World" program.

CODE EXAMPLE 6-1 Output of the "Hello World" program running under hypervisor

```
MBFW_INFO: Powering on OpenSPARC T1
``Alive and well ...
Strand start set = 0xf
Total physical mem = 0xac00000
Scrubbing the rest of memory
Number of strands = 0x4
membase = 0x0
memsize = 0x1000000
physmem = 0xac00000
 done
returned status 0x0
setup everything else
Setting remaining details
Start heart beat for control domain
Hello World

Guest stand-alone program has terminated.
Entering infinite loop.
```

6.4.2 Creating your own Stand-alone Program

Other programs can be run under hypervisor using this set-up, but there are some limitations:

- The program must be in a binary image format, because ELF format is not understood by hypervisor.
- There is no operating system, so system and library calls that are normally handled by the operating system are not available.
- Hypervisor does not dynamically link programs, so the program must be statically linked.

Solaris executable programs are in ELF format. When an executable program is run on Solaris, a memory image of the process is created from the executable program by the Solaris run-time loader. Hypervisor doesn't understand ELF format and it doesn't provide a run-time loader for ELF executable programs. The memory image of the application program is generated off-line by an ELF to memory image utility. It is this memory image that is preloaded in memory. Hypervisor doesn't support shared libraries and the executable program must be statically linked.

A minimalist Operating System for running the stand-alone program is provided by the library `libos.a`. All stand-alone programs must be linked with this OS library. The `libos.a` library provides functionality to initialize hypervisor interface, initialize privileged registers, set up a trap table, set up stack for the stand-alone program and write to console. Spill/Fill trap handlers are provided by the trap table so that the stand-alone program can execute nested and/or recursive function calls.

6.5 Booting OpenSolaris on an FPGA Evaluation Board

Booting OpenSolaris is accomplished in the same way that a stand-alone program is run. The critical OpenSolaris binaries are bundled into a RAM disk image, which is loaded into memory at the proper location. The firmware code will decompress the disk image at the start of the simulation, then will start loading the operating system.

The OpenSolaris boot setup uses the same memory allocation as a stand-alone program. Refer to FIGURE 6-3 for the memory allocation.

6.5.1 Booting from the provided OpenSolaris RAM Disk Image

To boot OpenSolaris on an FPGA evaluation board, perform the following procedure:

1. **Locate the proper boot PROM image file.**

   The boot PROM image files are found in the following location:

```
$DV_ROOT/design/sys/edk/os/proroms
```

The boot PROM image which starts OBP is required for booting an operating system.

2. **Locate the disk image file for OpenSolaris.**

A RAM disk image of a stripped-down OpenSolaris installation is included in the hardware package. It is located in the following directory:

```
$DV_ROOT/design/sys/edk/os/OpenSolaris/proto
```

3. **Compile the MicroBlaze firmware for standalone program.**

In the XPS user interface right-click on the ccx-firmware project in the Applications window and select Build.

4. **Make sure that the microblaze_0_bootloop software project is set to initialize BRAMs.**

Right-click on the microblaze_0_bootloop project in the Applications window and select "Mark to initialize BRAMs." Make sure that no other application is marked this way

5. **Select the following menu item to update the bitstream.**

Device Configuration --> Update Bitstream

6. **Download the bitstream to the FPGA.**

Device Configuration --> Download Bitstream

7. **Start any terminal window, such as Hyperterminal, and connect to the serial port that is connected to the FPGA board.**

8. **Start XMD.**

Select menu Debug --> Launch XMD or click the Launch XMD button on the XPS GUI.

9. **Download the firmware program to the FPGA.**

   In XMD, type the following command

```
XMD% dow ccx-firmware/executable.elf
```

10. **Download the RAM disk image of the OpenSolaris installation to DRAM.**

```
XMD% dow -data os/OpenSolaris/proto/ramdisk.snv-b77-nd.gz
0x8af00000
```

11. **Download the reset and hypervisor code for the OpenSPARC T1 to DRAM.**

```
XMD% dow -data os/proms/1c4t_obp_prom.bin 0x8ff00000
```

12. **Start the boot process.**

```
XMD% run
```

   Hypervisor will start up, then branch to Open Boot PROM (OBP). After a few minutes, OBP will give an OK prompt.

13. **Type "boot" at the OBP OK prompt.**

```
ok  boot -mverbose
```

   The OBP program will then start to load OpenSolaris. The boot process will take anywhere from 40 to 60 minutes, but will eventually give a login prompt. Type "`root`" at the login prompt. At this point most of the basic commands may be run.

   For details on default OpenSolaris boot process and services. Refer to the Frequently Asked Questions (FAQ) under:

```
$DV-ROOT/design/sys/edk/os/OpenSolaris/docs/t1_fpga_Opensolaris_f
aq.txt
```

6.5.2 Adding new Programs to the OpenSolaris RAM Disk Image

New programs may be added to the RAM disk image. These programs may then be run under OpenSolaris on the FPGA evaluation board. This is done by mounting the RAM disk image as a file system, making new directories, copying files to it, then unmounting it. This process requires root permissions on the machine where this process is performed. The sequence of commands to do this is shown below:

```
% su -
# mkdir ram-disk-mount-dir
# lofiadm -a ram-disk-file-name /dev/lofi/1
# mount /dev/lofi/1 ram-disk-mount-dir
# cd ram-disk-mount-dir
# cp new-file path
# cd /
# umount ram-disk-mount-dir
# lofiadm -d ram-disk-file-name
```

6.6 Booting Ubuntu Linux on an FPGA Evaluation Board

Beginning with the OpenSPARC T1 1.7 release, a RAM disk image is provided which allows the user to boot Ubuntu Linux on the FPGA evaluation platform. The procedure is very similar to the procedure used to boot OpenSolaris. Only the RAM disk image is different. Here's how to get Linux running on the OpenSPARC T1 FPGA system

1. **Follow steps 1-7 in** Section 6.5.1, "Booting from the provided OpenSolaris RAM Disk Image" on page 6-16**.**

At the end of the above seven steps, the FPGA should be programmed, and the board properly connected to the host computer.

2. **Start XMD.**

Select menu Debug --> Launch XMD or click the Launch XMD button on the XPS GUI.

3. **Download the firmware program to the FPGA.**

In XMD, type the following command:

```
XMD% dow ccx-firmware/executable.elf
```

4. **Download the RAM disk image of the Ubuntu Linux installation to DRAM.**

```
XMD% dow -data os/Ubuntu/7.10-Gutsy/proto/ramdisk.ubuntu-7.10-gutsy.gz
0x8af00000
```

5. **Download the reset and hypervisor code for the OpenSPARC T1 to DRAM.**

```
XMD% dow -data os/OpenSolaris/proto/1c4t_obp_prom.bin 0x8ff00000
```

6. **Start the boot process.**

```
XMD% run
```

7. **Type "boot" at the OBP OK prompt.**

```
ok boot
```

The OBP program will then start to load Linux. The boot process will take anywhere from 40 to 60 minutes, but will eventually give a login prompt. Type “root” at the login prompt. The password is also set to “root”. At this point most of the basic commands may be run.

For more information about running Linux on OpenSPARC please see the FAG file located in the following directory:

```
$DV_ROOT/design/sys/edk/os/Ubuntu/7.10-Gutsy/docs/t1_fpga_ubuntu_gut
sy_faq.txt
```

6.7 Running System-level Simulation with Modelsim

Environment files are provided to enable the user to run a system-level simulation of an OpenSPARC T1 diagnostic test using Xilinx Platform Studio (XPS) and the Modelsim simulator from Mentor Graphics. The simulation environment includes the following items:

- A board model which instantiates the FPGA and a DDR2 DRAM DIMM model.
- Scripts to generate the full FPGA model, compile the firmware programs and set up the simulation.

To run the system simulation using the default bypass_win test, follow the following procedure:

1. **Copy the EDK project.**

   The full-system simulation requires an edit to the system.mhs file, which would cause EDK to try to re-build the entire design. This can be prevented by creating a copy of the project just for simulation.

2. **Edit the system.mhs file.**

The following line must be added to the system.mhs line. This parameter needs to be added to the configuration of the mpmc block. This parameter must not be set when generating a bit file for an FPGA.

```
PARAMETER C_SKIP_SIM_INIT_DELAY = 1
```

3. **Compile the Xilinx libraries with Modelsim or link the project with already-compiled libraries.**

   From the XPS user interface, select the following menu item and follow the instructions from the wizard that pops up:

   Simulation --> Compile Simulation Libraries

4. **Ensure that the software project ccx-firmware-diag is set to mark BRAMs.**

   If it is not, then right-click on the ccx-firmware-diag software project and select “Mark to Initialize BRAMs.” Make sure that no other program is set to initialize BRAMs.

5. **Change the compile options for the ccx-firmware-diag project to include the `-DSIMULATION` option.**

   The -DSIMULATION compile option suppresses all print statements from the firmware. This is necessary to avoid simulating the printing of messages, which are sent by the UART at 9600 baud. To set this option, right-click on the ccx-firmware-diag software project and select Set Simulation Options. Then click on the Paths and Options tab. The best options for simulation are shown in the box below:

```
-DREGRESSION_MODE -DSIMULATION
```

6. **Next call a TCL script to generate the system model.**

```
% xps -nw -scr boardsim/setupsim.tcl system.xmp
```

   This script generates a simulation model for the entire FPGA, including the OpenSPARC T1 core, MicroBlaze core, memory controllers, and all the connecting logic. It also compiles the firmware code. When the firmware code has been compiled, the script converts the executable into a binary image, and then prepares initialization images for the DRAM models. Currently the binary image is split into four pieces because the current simulation configuration has four DRAM model instances. The script can be modified for a different memory configuration

7. **Download a DDR2 SDRAM model and modify it to allow pre-initialization using the verilog function `readmemh`.**

   The system-level simulation requires a DDR2 SDRAM model, which must be downloaded from a DRAM vendor. The model must allow for pre-initialization so that the firmware code can be placed in memory at the start of the test. The second thing to be careful about is the ordering of memory addresses within the model. The DRAM memory space is broken up into banks, rows and columns, and the code will be scrambled up if the model partitions its memory differently than the memory controller does. FIGURE 6-4 shows how the Xilinx mpmc memory controller allocates address bits between row, column and bank addresses.

FIGURE 6-4 DDR2 Memory Addressing of the MicroBlaze Memory Controller

A DDR2 SDRAM model from Micron Technology, Inc. was used in preparing the system simulation. This model did not have the capability for pre-initialization of the model. In addition, the model provided has a 184-pin DIMM model, not the 200-pin SODIMM that is on the board. This was handled by using the x16 configuration of the DIMM model, and adapting the board model to fit it. A script is provided to hack this memory model to work with the system simulation set-up. To run this script do the following:

a. **Download the Micron DDR2 memory model from the following web site:**

   `http://download.micron.com/downloads/models/verilog/sdram/ddr2/512Mb_ddr2.zip`

b. **Copy the models to the edk directory and run the patch script.**

```
% mkdir design/sys/edk/boardsim/dram_model
% cp ddr2.v ddr2_parameters.v design/sys/edk/boardsim/dram_model
% cp ddr2_dimm.v design/sys/edk/boardsim/dram_model
% cd design/sys/edk/boardsim/dram_model
% ../microndddr2_patch
```

If different DDR2 SDRAM models are used, the procedure to get them working in the system simulation will be different.

8. **Run the simulation.**

Change to the behavioral directory and run the simulation.

```
% cd simulation/behavioral
% vsim -do ../../do_sim_mb.do
```

The simulation should start. The first 3 ms of the simulation are required for the MicroBlaze processor to initialize its instruction and data caches. When this is complete, the OpenSPARC T1 core is started and begins to execute code.

A monitor model prints diagnostic messages to the log every time the OpenSPARC T1 core fetches instructions from memory. Sample output from the code monitor is shown in CODE EXAMPLE 6-2.

CODE EXAMPLE 6-2 Output from the PCX Monitor

```
# PCX:  3082235.000 ns : I-fetch from address  0xfff0000020
# PCX:  3143915.000 ns : I-fetch from address  0xfff0000024
# PCX:  3166855.000 ns : I-fetch from address  0xfff0000028
# PCX:  3188215.000 ns : I-fetch from address  0xfff000002c
# PCX:  3209655.000 ns : I-fetch from address  0xfff0000030

. . .

# PCX:  6191955.000 ns : I-fetch from address  0x1130000140
# PCX:  6223335.000 ns : I-fetch from address  0x1130000160
# PCX:  6255515.000 ns : I-fetch from address  0x1000122000
# PCX:  6255515.000 ns : Reached good trap:  Diag Passed
```

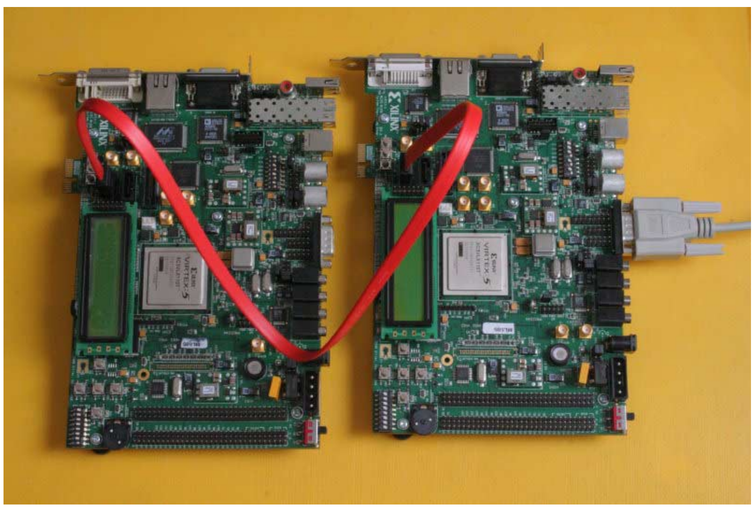
At the end of the test, the code will branch to one of two locations. The *good trap* location indicates a successful test, while the *bad trap* location indicates a problem. The PCX monitor will stop the simulation shortly after the *good trap* or *bad trap* location is reached.

The default diagnostic test that is simulated is *bypass_win*. Other diagnostic tests may be simulated by following the same procedure used to run these tests on the FPGA board. This is discussed in Section 6.3.2, "Running other Diags on the FPGA Board" on page 6-7.

6.8 Dual-Core System EDK Project

A new EDK project is provided which allows the user to create a two-core system by connecting two OpenSPARC evaluation boards together. This project is supported only on the OpenSPARC Evaluation Kit, which contains a Xilinx Virtex 5 XC5VLX110T FPGA, although another project is available for the BEE3 board. Implementation of the project requires two boards. A single four-thread core is implemented on the FPGA on each board, and the two cores communicate over an Aurora interface. One board is the master board. It maintains the entire OpenSPARC memory space and processes all memory transactions from the two cores. Transactions from the other board are simply forwarded to the master board for processing. The connection of the system is shown in FIGURE 6-5.

FIGURE 6-5 EDK Dual-Core System Connection



6.8.1 Generation of a Bit File for the Dual-Core Project

A bit file for the dual-core project can be generated in the same way it is done for the single-core project (see Section 6.2, "Generation of a Bit File for a Xilinx FPGA" on page 6-3). The Xilinix Platform Studio (XPS) is started, and the project file for the dual-core project is loaded. The project file for the dual-core project in the same directory as the single-core project. The name of the file is shown in the following box.

```
$DV_ROOT/design/sys/edk/dual_core.xmp
```

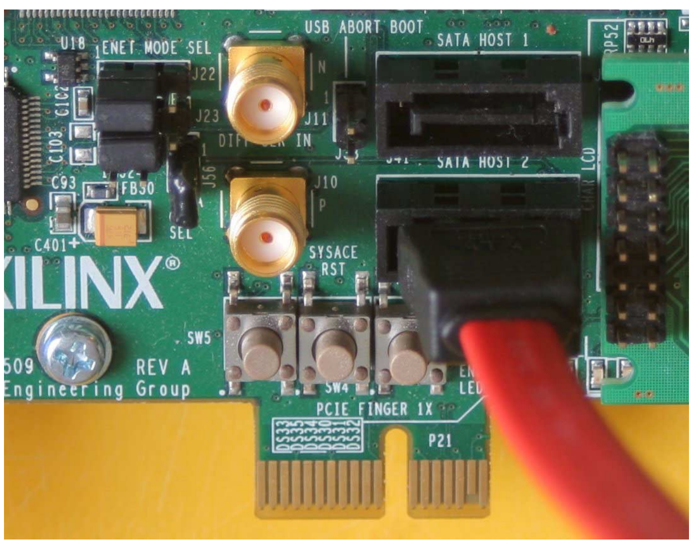
6.8.2 Running OpenSPARC Diagnostics on the Dual-Core Design

Running on the dual-core design is more complicated because there are two FPGAs on two different boards to program. Here are the steps required to run a diagnostic test on the dual-core design.

1. **Connect the boards together.**

   The boards are connected together using the red SATA crossover cable supplied with the OpenSPARC Development Kit. The cable must connect to each board using SATA connector #2 on the board, see FIGURE 6-6.

FIGURE 6-6 Dual-Core System Cable Connection



A null-modem serial cable should be used to connect from the master board to a host computer.

2. **Start any terminal window on the host computer, such as Hyperterminal, and connect to the serial port that is connected to the Master FPGA board.**

   The communication settings should be set to 9600 baud, data 8 bits, parity none, stop bits 1,and flow control none.

3. **Select the Applications tab in the Project Information Area (the left-hand window).**

   A list of software applications will be shown.

4. **Right-click on the micro_0_bootloop application and select "Mark to initialize BRAMs".**

   This program should already be set to initialize BRAMs, so this step may not be necessary.

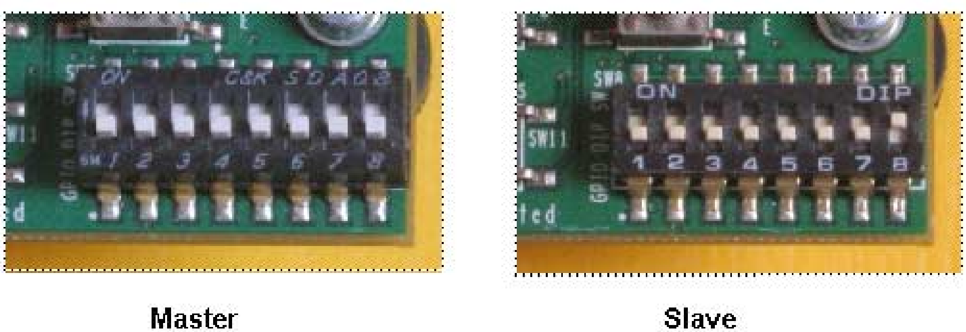
5. **Select the following menu item to update the bitstream.**

   Select Menu Device Configuration - - > Update Bitstream

6. **Set the DIP switches on each board to indicate the core ID.**

   The core ID is set by the SW8 DIP switches on the board. Switches 6, 7, and 8 of this switch bank are used, with switch 6 being the most-significant bit of the core ID, and switch 8 being the least-significant bit. The core ID for the master board should be 0, and the core ID for the slave board should be 1. See FIGURE 6-7.

FIGURE 6-7 DIP Switch Settings for Master and Slave Boards



7. **Make sure the firmware executables are up to date.**

   The firmware for the master board will be ccx-firmware-diag. The firmware for the slave board should be ccx-firmware-slave. Each executable can be re-compiled by right-clicking on the software project name and selecting build in the popup menu.

8. **Download the bit file to the slave board.**

   Connect the download cable to the slave board. Then select menu Device Configuration - - > Download Bitstream

9. **Download the firmware code to the slave board.**

   Select menu Debug - -> Launch XMD or click the button to launch XMS from the XPS GUI. Download and start the slave firmware program.

   Change to the behavioral directory and run the simulation.

```
XMD% dow ccx-firmware-slave/executable.elf
XMD% run
XMD% exit
```

10. **Download the bit file to the master board.**

    Connect the download cable to the master board. Then select menu Device Configuration - - > Download Bitstream

11. **Download the firmware code to the master board.**

    Select menu Debug - - > Launch XMD or click the button to launch XMS from the XPS GUI. Download and start the master firmware program.

```
XMD% dow ccx-firmware-diag/executable.elf
```

12. **Run the diagnostic test.**

```
XMD% run
```

The output from the test will appear in the terminal window.

```
MBFW_INFO: Running RTL diag "bypass_win"
MBFW_INFO: Microblaze firmware initialization completed.
MBFW_INFO: Powering on OpenSPARC T1
MBFW_INFO: speculative_ifill_data being returned for 0x1000144020
MBFW_INFO: received ifill request for good trap addr: 0x1000122000
MBFW_INFO: Thread 0 reached good trap.
MBFW_INFO: All threads reached good trap
```

Note that tests in the thread1 or core1 regressions are not that interesting for dual-core system because they only use one core to begin with. More interesting tests that exercise more than one core may be found in the fullchip regression.

6.8.3 Running other Tests on the Dual-Core System

The procedure above runs a single default test on the dual-core system. To run different tests, the procedure described in Section 6.3.2, "Running other Diags on the FPGA Board" on page 6-7 should be followed to generate the memory image file for the desired test. Once that memory image file is created, it can be copied to the source directory for the ccx-firmware-diag software project and the project can be re-compiled to link in this new memory image. Once this has been done the procedure in the previous section can be followed to run the test on the dual-core system.

6.8.4 Running an Entire Regression on the Dual-Core System

Because running on the dual-core system requires programming multiple boards and manually moving the download cable between the two boards, there is no way provided to run an entire regression of tests on the two-core system.

6.8.5 Booting an Operating System on the Dual-Core System

The procedure to boot an operating system on the dual-core system is very similar to the procedure in Section 6.5.1, "Booting from the provided OpenSolaris RAM Disk Image" on page 6-16. However it is more complicated because there are two boards to program. Following is the complete procedure:

1. **Connect the boards together.**

The boards are connected together using the red SATA crossover cable supplied with the OpenSPARC Development Kit. The cable must connect to each board using SATA connecter #2 on the board (see FIGURE 6-5).

A null-modem serial cable should be used to connect from the master board to a host computer.

Set the DIP switches on each board correctly (see FIGURE 6-7).

2. **Start any terminal window on the host computer, such as Hyperterminal, and connect to the serial port that is connected to the master FPGA board.**

The communication settings should be set to 9600 baud, data 8 bits, parity none, stop bits 1, and flow control none.

3. **Locate the proper boot PROM image file.**

   The default OpenSPARC core netlist supplied with the EDK project is a four-thread netlist. The PROM file required to boot an operating system would be 2c4t_obp_prom.bin. If a single-thread netlist was substituted, the proper PROM image file would be 2c1t_obp_prom.bin. The boot PROM images may be found in the following location:

```
$DV_ROOT/design/sys/edk/os/proms
```

4. **Locate the disk image file for the Operating System.**

   As before, the RAM disk image of the OpenSolaris installation is located in the following directory:

```
$DV_ROOT/design/sys/edk/os/OpenSolaris/proto
```

5. **Compile the MicroBlaze firmware for standalone program.**

   In the XPS user interface right-click on the ccx-firmware project in the Applications window and select Build.

6. **Make sure that the microblaze_0_bootloop software project is set to initialize BRAMs.**

   Right-click on the microblaze_0_bootloop project in the Applications window and select "Mark to initialize BRAMs". Make sure that no other application is marked this way.

7. **Program the slave board.**

   Connect the download cable to the slave board. Download the bit file, then start XMD and download and start the slave firmware:

```
XMD% dow ccx-firmware-slave/executable.elf
XMD% run
XMD% exit
```

8. **Program the master board.**

Move the download cable to the master board. Download the bit file again, then start XMD and download the master firmware:

```
XMD% dow ccx-firmware/executable.elf
```

9. **Download the Operating system RAM disk image and the PROM image to the master board DRAM.**

For Solaris:

```
XMD% dow -data
os/OpenSolaris/proto/ramdisk.snv-b77-nd-no-boot-time-network.gz
0x8af00000
XMD% dow -data os/OpenSolaris/proto/2c4t_obp_prom.bin 0x8ff00000
```

Linux does not currently work on the dual-core design.

10. **Start the boot process.**

```
XMD% run
```

Hypervisor will start up, then branch to Open Boot PROM (OBP). After a few minutes, OBP will give an OK prompt.

11. **Type a boot command at the OBP OK prompt.**

```
ok boot
```

The OBP program will then start to load the operating system. The boot process will take up to two hours, but will eventually give a login prompt. Type "root" at the login prompt to log in. There is no root password. At this point, most basic operating system commands may be run.

6.9 EDK Project for the ML411 Evaluation Board

An EDK project file is provided for the ML410/ML411 evaluation board. This project file contains the same peripherals as the default project for the ML505-V5LX110T board, and has the same address map. All the procedures outlined in the previous sections can be run on the ML411 board with the following changes:

1. The script for downloading the design to the ML411 board is different from the one used on the ML505 board. Before attempting to download the design to the FPGA, change to the edk/etc. directory, and rename the ML411 script.

```
% cd design/sys/edk/etc
% mv download.cmd download_bak.cmd
% mv download_ml411.cmd download.cmd
```

2. When opening the project in EDK, open the system_ml411.xmp file instead of the system.xmp file.
3. The OpenSPARC T1 netlist supplied in the pcores directory of the EDK project is too large to route easily on the XC4VFX100 FPGA. Try generating a netlist with one of the following options:
   a. Reduce the number of TLB entries to 8.
   b. Use a single-thread core.
4. For an ML410 board with an XC4VFX60 FPGA, only a single-thread core will fit.

6.10 Running System-level Simulation on Legacy Projects

Release 1.5 of OpenSPARC T1 featured an EDK project which used the on-board 64 MB DDR DRAM chip instead of the DDR2 DIMM. This section preserves the instruction for running full-system simulation on the OpenSPARC T1 release 1.5 project.

To run the system simulation using the default bypass_win test, follow the following procedure:

1. **Compile the Xilinx libraries with Modelsim or link the project with already-compiled libraries.**

   From the XPS user interface, select the following menu item and follow the instructions from the wizard that pops up:

   Simulation --> Compile Simulation Libraries

2. **Ensure that the software project mb-firmware is set to mark BRAMs.**

   This should be set by default in the project. If it is not then, right-click on the mb-firmware software project and select “Mark to Initialize BRAMs”.

3. **Change the compile options to include the** `-DSIMULATION` **option.**

   The -DSIMULATION compile option suppresses all print statements from the firmware. This is necessary to avoid simulating the printing of messages, which are sent by the UART at 9600 baud. To set this option, right-click on the `mb-firmware` software project and select `Set Simulation Options`. Then click on the `Paths and Options` tab. The best options for simulation are shown in the box below:

```
-DREGRESSION_MODE -DSIMULATION
```

4. **Next call a TCL script to generate the system model.**

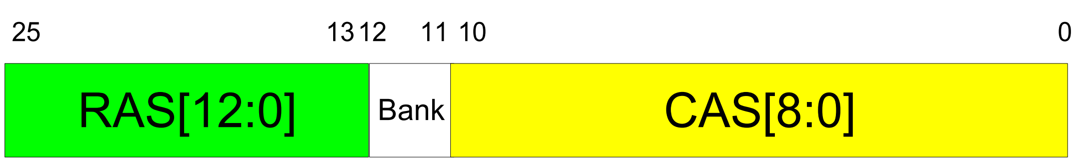
```
% xps -nw -scr boardsim/setupsim.tcl system.xmp
```

   This script generates a simulation model for the entire FPGA, including the OpenSPARC T1 core, MicroBlaze core, memory controllers, and all the connecting logic. It also compiles the firmware code.

5. **Download a DDR SDRAM model and modify it to allow pre-initialization using the verilog function** `readmemh`**.**

   The system-level simulation requires a DDR SDRAM model, which must be downloaded from a DRAM vendor. The model must allow for pre-initialization so that the firmware code can be placed in memory at the start of the test. The second thing to be careful about is the ordering of memory addresses within the model. The DRAM memory space is broken up into banks, rows and columns, and the code will be scrambled up if the model partitions its memory differently than the memory controller does. FIGURE 6-8 shows how the address of the DRAM is split between row, column and bank addresses.

FIGURE 6-8 DDR Memory Addressing of the MicroBlaze Memory Controller



   A DDR SDRAM model from Micron Technology, Inc. was used in preparing the system simulation. This model did not have the capability for pre-initialization of the model. A script is provided to hack this memory model to work with the system simulation set-up. To run this script do the following:

a. **Download the Micron DDR memory model from the following web site:**

http://download.micron.com/downloads/models/verilog/sdram/ddr/256meg/256Mb_ddr.zip

b. **Copy the models to the edk directory and run the patch script.**

```
% cp ddr.v ddr_parameters.v design/sys/edk/boardsim
% cd design/sys/edk/boardsim
% ./microndddr_patch
```

If different DDR SDRAM models are used, the procedure to get them working in the system simulation will be different.

6. **Run the simulation.**

Change to the behavioral directory and run the simulation.

```
% cd simulation/behavioral
% vsim -do ../../do_sim_mb.do
```

The simulation should start. The first 3 ms of the simulation are required for the MicroBlaze processor to initialize its instruction and data caches. When this is complete, the OpenSPARC T1 core is started and begins to execute code. A monitor model prints diagnostic messages to the log every time the OpenSPARC T1 core fetches instructions from memory. Sample output from the code monitor is shown in CODE EXAMPLE 6-3.

CODE EXAMPLE 6-3 Output from the PCX Monitor

```
# PCX:  3082235.000 ns : I-fetch from address  0xfff0000020
# PCX:  3143915.000 ns : I-fetch from address  0xfff0000024
# PCX:  3166855.000 ns : I-fetch from address  0xfff0000028
# PCX:  3188215.000 ns : I-fetch from address  0xfff000002c
# PCX:  3209655.000 ns : I-fetch from address  0xfff0000030

. . .

# PCX:  6191955.000 ns : I-fetch from address  0x1130000140
# PCX:  6223335.000 ns : I-fetch from address  0x1130000160
# PCX:  6255515.000 ns : I-fetch from address  0x1000122000
# PCX:  6255515.000 ns : Reached good trap:  Diag Passed
```

At the end of the test, the code will branch to one of two locations. The *good trap* location indicates a successful test, while the *bad trap* location indicates a problem. The PCX monitor will stop the simulation shortly after the *good trap* or *bad trap* location is reached.

The default diagnostic test that is simulated is *bypass_win*. Other diagnostic tests may be simulated by following the same procedure used to run these tests on the FPGA board. This is discussed in Section 6.3.2, “Running other Diags on the FPGA Board” on page 6-7.

APPENDIX **A**

Design and Verification Manual Pages

This appendix provides the manual pages for commands used for OpenSPARC T1 design and verification.

A.1 sims

NAME

```
sims - Verilog rtl simulation environment and regression script
```

SYNOPSIS

```
sims [args ...]

NOTE: Use "=" instead of "space" to separate args and their options.

where args are:
```

SIMULATION ENV

```
    -sys=NAME
       sys is a pointer to a specific testbench configuration
       to be built and run. A config file is used to associate
       the sys with a set of default options to build the
       testbench and run diagnostics on it. The arguments
       in the config file are the same as the arguments passed
       on the command line.

    -group=NAME
       group name identifies a set of diags to run in a
```

```
        regression. The presence of this argument indicates
        that this is a regession run. The group must be found
        in the diaglist. Multiple groups may be specified to be
        run within the same regression.

     -group=NAME -alias=ALIAS
        This combination of options gets the diag run time options
        from the diaglist based on the given group and alias.
        The group must be found in the diaglist. The alias is
        made up of diag_alias:name_tag. Only one group should be
        specified when using this command format.
```

VERILOG COMPILATION RELATED

```
    -sim_type=vcs/ncv
        Defines which simulator to use, vcs or ncverilog, Defaults to vcs.

    -sim_q_command="command"
        Defines which job queue manager command to use to launch jobs.
        Defaults to /bin/sh and runs simulation jobs on the local machine.

    -ncv_build/-noncv_build
        Builds a ncverilog model and the vera testbench. Defaults to off.

    -ncv_build_args=OPTION
        ncverilog compile options. Multiple options can be specified using
        multiple such arguments.

    -ncv_use_vera/-noncv_use_vera
        Compiles in the vera libraries. Defaults to off.

    -vcs_build/-novcs_build
        Builds a vcs model and the vera testbench. Defaults to off.

    -vcs_build_args=OPTION
        vcs compile options. Multiple options can be specified using
        multiple such arguments.

    -vcs_clean/-novcs_clean
        Wipes out the model directory and rebuilds it from scratch.
        Defaults to off.

    -vcs_use_2state/-novcs_use_2state
        Builds a two-state model instead of the default four-state model.
        This defaults to off.

    -vcs_use_initreg/-novcs_use_initreg
        Initializes all registers to a valid state (1/0).
        This feature works with -tg_seed to set the seed of the random
```

```
    initialization. Defaults to off.

-vcs_use_fsdb/-novcs_use_fsdb
    Uses the debussy fsdb pli and include the dump calls in the
    testbench. this defaults to on.

-vcs_use_vcsd/-novcs_use_vcsd
    uses the vcs direct kernel interface to dump out debussy files.
    Defaults to on.

-vcs_use_vera/-novcs_use_vera
    Compiles in the vera libraries. If -vcs_use_ntb and -vcs_use_vera are
    used, -vcs_use_ntb wins. Defaults to off.

-vcs_use_ntb/-novcs_use_ntb
    Enables the use of NTB when building model (simv) and running simv.
    If -vcs_use_ntb and -vcs_use_vera are used, -vcs_use_ntb wins.
    Defaults to off.

-vcs_use_rad/-novcs_use_rad
    Uses the +rad option when building a vcs model (simv).
    Defaults to off.

-vcs_use_sdf/-novcs_use_sdf
    Builds vcs model (simv) with an sdf file. Defaults to off.

-vcs_use_cli/-novcs_use_cli
    Uses the +cli -line options when building a vcs model (simv).
    Defaults to off.

-flist=FLIST
    Full path to flist to be appended together to generate the
    final verilog flist. Multiple such arguments may be used and
    each flist will be concatenated into the final verilog flist
    used to build the model.

-graft_flist=GRAFTFILE
    GRAFTFILE is the full path to a file that lists each verilog
    file that will be grafted into the design. The full path to
    the verilog files must also be given in the GRAFTFILE.

-vfile=FILE
    Verilog file to be included into the flist

-config_rtl=DEFINE
    Places each such parameter as a ‘define’ in config.v to
    configure the model being built properly. This allows
    each testbench to select only the rtl code that it needs
    from the top-level rtl file.
```

```
  -model=NAME
       The name of a model to be built. The full path to a model
       is $MODEL_DIR/$model/$vcs_rel_name.

  -vcs_rel_name=NAME
       Specifies the release of the model to be built. The full path
       to a model is $MODEL_DIR/$model/$vcs_rel_name.
```

VERA COMPILATION RELATED

```
  VERA and NTB share all of the vera options except a few. See NTB RELATED.

  -vera_build/-novera_build
   Builds the vera/ntb testbench. Defaults to on.

  -vera_clean/-novera_clean
   Performs a make clean on the vera/ntb testbench before building
   the model. Defaults to off.

  -vera_build_args=OPTION
   Vera testbench compile time options. Multiple options can be
   specified using multiple such commands. These are passed as
   arguments to the gmake call when building the vera testbench.

  -vera_diag_args=OPTION
   Vera/ntb diag compile-time options.
   Multiple options can be specified using multiple such arguments.

  -vera_dummy_diag=NAME
   Provides a dummy vera diag name that will be
   overridden if a vera diag is specified, else used for vera
   diag compilation.

-vera_pal_diag_args=OPTION
   Vera/ntb pal diag expansion options.
   (i.e., "pal OPTIONS -o diag.vr diag.vrpal")
   Multiple options can be specified using multiple such arguments.

  -vera_proj_args=OPTION
   Vera proj file-generation options. Multiple options can be
   specified using multiple such arguments.

  -vera_vcon_file=ARG
   Name of the vera vcon file that is used when running the simulation.

  -vera_cov_obj=OBJ
   This argument is passed to the vera Makefile as a OBJ=1 and to
   vera as -DOBJ to enable a given vera coverage object. Multiple
```

```
   such arguments can be specified for multiple coverage objects.
```

NTB RELATED

```
  NTB and VERA share all of the vera options except these:

-vcs_use_ntb/-novcs_use_ntb
   Enables the use of NTB when building model (simv).
   If -vcs_use_ntb and -vcs_use_vera are used, -vcs_use_ntb wins.
   Defaults to off.

-ntb_lib/-nontb_lib
  Enables the NTB two-part compile where the Vera/NTB files get
  compiled first into a libtb.so file which is dynamically
  loaded by vcs at runtime. The libtb.so file is built by
  the Vera Makefile, not sims. Use the Makefile to affect the
  build. If not using -ntb_lib, sims will build VCS and NTB
  together in one pass (use Makefile to affect that build as
  well). Defaults to off.
```

VERILOG RUNTIME RELATED

```
  -vera_run/-novera_run
   Runs the vcs simulation and loads in the vera proj file
   or the ntb libtb.so file. Defaults to on.

  -vcd/-novcd
      Signals the bench to dump in VCD format.

  -vcdfile=filename
   The name of the vcd dump file. If the file name starts with
   a "/", that is the file dumped to; otherwise, the actual file is
   created under $tmp_dir/$vcdfile and copied back to the current
   directory when the simulation ends. Use "-vcdfile=`pwd`/filename"
   to force the file to be written in the current directory directly
   (not efficient since dumping is done over network instead of to
   a local disk).

  -vcs_run/-novcs_run
   Runs the vcs simulation (simv). Defaults to off.

  -vcs_run_args=OPTION
   vcs (simv) runtime options. Multiple options can be specified
   using multiple such arguments.

  -vcs_finish=TIMESTAMP
   Forces vcs to finish and exit at the specified timestamp.
```

```
-fast_boot/-nofast_boot
   Speeds up booting when using the ciop model. Passes the
   +fast_boot switch to the simv run and the -sas_run_args=-DFAST_BOOT
   and -midas_args=-DFAST_BOOT to sas and midas. Also sends
   -DFAST_BOOT to the diaglist and config file preprocessors.

  -debussy/-nodebussy
   Enables debussy dump. This must be implemented in the testbench
   to work properly. Defaults to off.

  -start_dump=START
   Starts dumping out a waveform after START number of units.

  -stop_dump=STOP
   Stops dumping out a waveform after STOP number of units.

  -fsdb2vcd
       Runs fsdb2vcd after the simulation has completed to generate a vcd file.

  -fsdbfile=filename
   The name of the debussy dump file.
   If the file name starts with a "/", that is the file dumped to.
   Otherwise, the actual file is created under $tmp_dir/$fsdbfile
   and copied back to the current directory when the simulation ends.
   Use "-fsdbfile=`pwd`/filename" to force the file to be
   written in the current directory directly (not efficient since
   dumping is done over network instead of to a local disk).

  -fsdbDumplimit=SIZE_IN_MB
   Max size of Debussy dump file.  Minimum value is 32MB.
   Latest values of signal values making up that size is saved.

  -fsdb_glitch
     Turns on glitch and sequence dumping in fsdb file. This will collect
   glitches and sequence of events within time in the fsdb waveform.
   beware that this will cause the fsdb file size to grow significantly.
   This option effectively does this:
   setenv FSDB_ENV_DUMP_SEQ_NUM 1
   setenv FSDB_ENV_MAX_GLITCH_NUM 0
   Defaults to off.

  -rerun
     Reruns the simulation from an existing regression run directory.

  -post_process_cmd=COMMAND
   Post-processing command to be run after vcs (simv) run completes.

  -pre_process_cmd=COMMAND
```

Pre-processing command to be run before vcs (simv) run starts.

-use_denalirc=FILE
Uses FILE as the .denalirc in the run area. Default copies
$env_base/.denalirc

ZEROIN RELATED

-zeroIn_checklist
Runs 0in checklist

-zeroIn_build
Builds 0In pli for simulation into vcs model.

-zeroInSearch_build
Builds 0in search pli for simulation into vcs model.

-zeroIn_build_args
Additional arguments to be passed to the 0in command.

-zeroIn_dbg_args
Additional debug arguments to be passed to the 0in shell.

SAS RELATED

-sas/-nosas
Runs architecture-simulator. If vcs_run option is OFF,
simulation is sas-only. If vcs_run option is ON, sas
runs in lock-step with rtl. Defaults to off.

-sas_run_args=DARGS
Defines arguments for sas.

MIDAS RELATED

midas is the diag assembler.

-midas_args=DARGS
Arguments for midas. midas creates memory image and user-event
files from the assembly diag.

-midas_only
Compiles the diag using midas and exit without running it.

-midas_use_tgseed
Adds -DTG_SEED=tg_seed to midas command line. Use -tg_seed to
set the value passed to midas or use a random value from /dev/random.

SJM RELATED

```
sjm is the J-Bus bus functional model

  -sjm_args
       Arguments to be passed in to sjm_tstgen.pl for generation of an sjm
       random diagnostic.

  -sjm/-nosjm
       Generates a random sjm diagnostic using the -tg_seed if provided.
       Defaults to off.

  -tg_seed
       Random generator seed for sjm random test generators.
       Also the value passed to +initreg+ to randomly initialize registers
       when -vcs_use_initreg is used.
```

VCS COVERMETER RELATED

```
  -vcs_use_cm/-novcs_use_cmd
   Passes in the -cm switch to vcs at build time and simv at runtime
   Defaults to off.

  -vcs_cm_args=ARGS
   Argument to be given to the -cm switch.

  -vcs_cm_cond=ARGS
   Argument to be given to the -cm_cond switch.

  -vcs_cm_config=ARGS
   Argument to be given to the -cm_hier switch.

  -vcs_cm_fsmcfg=ARGS
   Argument to be given to the -cm_fsmcfg switch.
   Specifies an FSM coverage configuration file.

  -vcs_cm_name=ARGS
   Argument to be given to the -cm_name switch. defaults to cm_data.
```

MISC

```
  -nobuild
    Master switch to disable all building options.
  There is no such thing as -build to enable all build options.

  -copyall/-nocopyall
  Copies back all files to launch directory after passing
  regression run.  Normally, only failing runs cause a
  copy back of files. Defaults to off.
```

```
  -copydump/-nocopydump
    Copies back dump file to launch directory after passing
    regression run.  Normally, only failing runs cause a copy
    back of non-log files.  The file copied back is sim.fsdb,
    or sim.vcd if -fsdb2vcd option is set.
    Default is off.

  -tarcopy/-notarcopy
    Copies back files using 'tar'. This only works in copyall or
    in the case the simulations 'fails' (per sims' determination).
    Default is to use 'cp'.

  -diag_pl_args=ARGS
    If the assembly diag has a Perl portion at the end, it
    is put into diag.pl and is run as a Perl script.
    This allows you to give arguments to that Perl script.
    The arguments accumulate, if the option is used multiple
    times.

  -pal_use_tgseed
   Sends '-seed=<tg_seed_value>' to pal diags.  Adds
   -pal_diag_args=-seed=tg_seed to midas command line, and
   -seed=tg_seed to pal options (vrpal diags). Use -tg_seed to set
   the value passed to midas or use a random value from /dev/random.

  -parallel
      When specifying multiple groups for regressions, this switch will
      submit each group to Job Q manager to be executed as a separate regression.
      This has the effect of speeding up regression submissions.
      NOTE: This switch must not be used with -injobq

  -reg_count=COUNT
      Runs the specified group multiple times in regression mode. This
      is useful when we want to run the same diag multiple times using
      a different random generator seed each time or some such.

  -regress_id=ID
      Specifies the name of the regression.

  -report
      Used to produce a report of a an old or running
      regression. With -group options, sims produces the report
      after the regression run. Report for the previous
      regression run can be produced using -regress_id=ID
      option along with this option.

-finish_mask=MASK
   Masks for vcs simulation termination. Simulation terminates
   when it hits 'good_trap' or 'bad_trap'. For multithread
```

```
 simulation, simulation terminates when any of the thread
 hits bad_trap, or all the threads specified by the finish_mask
 hits the good_trap.
 example: -finish_mask=0xe
 Simulation will be terminated by good_trap, if threads 1, 2 and
 3 hit the good_trap.

-stub_mask=MASK
 Mask for vcs simulation termination. Simulation ends when the
 stub driving the relevant bit in the mask is asserted. This
 is a hexadecimal value similar to -finish_mask.

-wait_cycle_to_kill=VAL
 Passes a +wait_cycle_to_kill to the simv run. a testbench
 may chose to implement this plusarg to delay killing a
 simulation by a number of clock cycles to allow collection
 of some more data before exiting (e.g. waveform).

-rtl_timeout
   Passes a +TIMEOUT to the simv run.
 Sets the number of clock cycles after all threads have become
 inactive for the diag to exit with an error. If all threads hit
 good trap on their own the diag exits right away. If any of the
 threads is inactive without hitting good trap/bad trap the
 rtl_timeout will be reached and the diag fails. Defaults to 1000.
 This is only implemented in the cmp based testbenches.

-max_cycle
   Passes a +max_cycle to the simv run.
 Sets the maximum number of clock cycle that the diag will take
 to complete. Defaults to 30000. If max_cycle is hit the diag
 exits with a failure. Not all testbenches implement this
 feature.

-norun_diag_pl
 Does not run diag.pl (if it exists) after simv (vcs) run.
 Use this option if, for some reason, you want to run an
 existing assembly diag without the Perl part that is in
 the original diag.

-nosaslog
    Turns off redirection of sas stdout to the sas.log file.
    Use this option when doing interactive runs with sas.

-nosimslog
    Turns off redirection of stdout and stderr to the sims.log file.
    Use this option to get to the cli prompt when using vcs or to
    see a truncated sim.log file that exited with an error. This
    must be used if you want control-c to work while vcs is running.
```

```
-nogzip
    Turns off compression of log files before they are copied over
    during regressions.

-version
    Print version number.

-help
    Prints this man page.
```

IT SYSTEM RELATED

```
-use_iver=FILE
    Full path to iver file for frozen tools.

-use_sims_iver
    For reruns of regression tests only, use sims.iver to choose
    TRE tool versions saved during original regression run.

-dv_root=PATH
    Absolute path to design root directory. This overrides $DV_ROOT.

-model_dir=PATH
    Absolute path to model root directory. This overrides $MODEL_DIR.

-tmp_dir=PATH
    Path where temporary files such as debussy dumps will be created.

-sims_config=FILE
    Full path to sims config file.

-env_base=PATH
    Specifies the root directory for the bench environment.
    It is typically defined in the bench config file. It has no
    default.

-config_cpp_args=OPTION
    Allows the user to provide CPP arguments (defines/undefines)
    that will be used when the testbench configuration file is
    processed through cpp. Multiple options are concatenated
    together.

-result_dir=PATH
    Allows the regression run to be launched from a different
    directory than the one sims was launced from. Defaults to
    $ENV{PWD}.

-diaglist=FILE
```

```
        Full path to diaglist file.

    -diaglist_cpp_args=OPTION
        Allows the user to provide CPP arguments (defines/undefines)
        that will be used when the diaglist file is processed through
        cpp. Multiple options are concatenated together.

  -asm_diag_name=NAME
    -tpt_diag_name=NAME
    -tap_diag_name=NAME
    -vera_diag_name=NAME
    -vera_config_name=NAME
    -efuse_image_name=NAME
    -image_diag_name=NAME
    -sjm_diag_name=NAME
    -pci_diag_name=NAME
        Name of the diagnostic to be run.

    -asm_diag_root=PATH
    -tpt_diag_root=PATH
    -tap_diag_root=PATH
    -vera_diag_root=PATH
    -vera_config_root=PATH
    -efuse_image_root=PATH
    -image_diag_root=PATH
    -sjm_diag_root=PATH
    -pci_diag_root=PATH
     Absolute path to diag root directory. sims will perform a find
     from here to find the specified type of diag. If more than one
     instance of the diag name is found under root, sims exits with
     an error. this option can be specified multiple times to allow
     multiple roots to be searched for the diag.

    -asm_diag_path=PATH
    -tpt_diag_path=PATH
    -tap_diag_path=PATH
    -vera_diag_path=PATH
    -vera_config_path=PATH
    -efuse_image_path=PATH
    -image_diag_path=PATH
    -sjm_diag_path=PATH
    -pci_diag_path=PATH
     Absolute path to diag directory. sims expects the specified
     diag to be in this directory. The last value of this option
     is the one used as the path.
```

ENV VARIABLES

sims sets or uses the following ENV variables that may be used with pre/post processing scripts, and other internal tools:

TABLE A-1 Environment Variables

Environment Variable	Description
SIMS_LAUNCH_DIR	Path to launch directory where sims is running the job
ASM_DIAG_NAME	Contains the assembly diag name
VERA_LIBDIR	Dir where Vera files are compiled
DV_ROOT	Overwrite by -dv_root if specified
MODEL_DIR	Overwrite by -model_dir if specified
TRE_SEARCH	Based on -use_iver, -use_sims_iver
DENALI	User defined
VCS_HOME	User defined
VERA_HOME	User defined
NCV_HOME	User defined

PLUSARGS

+args are not implemented in sims. They are passed directly to simulator at compile time and at runtime.

DESCRIPTION

sims is the frontend for vcs to run single simulations and regressions.

How To Build Models

Build a vcs model using $DV_ROOT as design root:

```
sims -sys=cmp -vcs_build
```

Build a ncverilog model using $DV_ROOT as design root:

```
sims -sys=cmp -ncv_build
```

Build the vera testbench only using $DV_ROOT as design root:

```
sims -sys=cmp -vera_build
```

Build a model from any design root:

```
sims -sys=cmp -vcs_build -dv_root=/home/regress/2002_06_03
```

Build a graft model from any design root:

```
  sims -sys=cmp -vcs_build -dv_root=/model/2002_06_03 \
  -graft_flist=/regress/graftfile
```

Build a model and re-build the vera:

```
  sims -sys=cmp -vcs_build -vera_clean
```

Build a model and turn off incremental compile:

```
  sims -sys=cmp -vcs_build -vcs_clean
```

Build a model with a given name:

```
  sims -sys=cmp -vcs_build -vcs_rel_name=mymodel
```

How to Run Models

Run a diag with default model:

```
  sims -sys=cmp -vcs_run diag.s
```

Run a diag with a specified model:

```
  sims -sys=cmp -vcs_rel_name=mymodel -vcs_run diag.s
```

Run a diag with debussy dump with default model:

```
  sims -sys=cmp -debussy +dump=cmp_top:0 -vcs_run diag.s
```

=head2 Run regressions

Run a regression using $DV_ROOT as design root:

```
  sims -group=mini
```

Run a regression using $DV_ROOT as design root and specify the diaglist:

```
  sims -group=mini -diaglist=/home/user/my_dialist
```

Run a regression using any design root:

```
  sims -group=mini -dv_root=/afara/design/regress/model/2002_06_03
```

A.2 midas

NAME

midas - assembles diags (Midas Is a Diag Assembler)

SYNOPSIS

midas [options] <diag_name>

DESCRIPTION

This program builds assembly diags. It is substantially more involved than simply assembling the diag because it also has to link the diag, program the MMU, and generate several output files.

The diag specified on the command line will be built. Pretty much everything else is configurable.

Options

The following are the options you need to get started:

-h Display man page.

-verbose [level] / -noverbose (abbreviated -v / -nov)
    Sets verbosity level (default=2). -noverbose (or -nov) is a synonym for -verbose 0, which means to generate no output in the absence of errors. The highest level of verbosity currently defined is 3.

-version
    Returns version information and exit.

-format
    Displays help on the diag format and exit.

-config <file>
    Uses this file as the config file instead of the one that is distributed with Midas.

-project <project>
    Uses this project for project-specific configuration. Default is the environment variable $PROJECT. Legal value is OpenSPARCT1.

Common Options

The following are the commonly used options:

```
-diag_root <path>
    Uses the specified path as a base for finding standard
    include files.  Default is $DV_ROOT.

-build_dir <path>
    Path (absolute or relative to where command is invoked)
  to directory where temporary files are generated and the
    build is done.  Default is './build'.

-dest_dir <path>
    Path (absolute or relative to where command is invoked)
    of where to store output files.  Default is '.'.

-find_root <dir>
    Interprets the diag on the command line as the name of a
    diag to search for.  It does a breadth-first search
    under the specified directory.  The default behavior is
    not to do any search, but to assume that the specified
    diag is a full or relative path to the file.

-find
    This is a shortcut for "-find_root
    <diag_root>/verif/diag".

-mmu <mmu_type>
    Generates programming for the specified MMU.  Recognized
    options are "ultra2", "OpenSPARCT1".

-ttefmt <tte_format>
    Specifies TTE format for those MMUs that require it.
    May be "sun4u" or "sun4v".  Default is project specific:
    "sun4v" for OpenSPARC T1.

-tsbtagfmt <tsbtagfmt>
    Specifies the format of the TSB tag.  Legal values are
    'tagaccess' and 'tagtarget'.  Default is
    project-specific: 'tagaccess' OpenSPARC T1.

-force_build or -f
    Builds the diag, even if it looks like it has the same
    input as before and the same args as before.

-copy_products / -nocopy_products
    By default, the product files generated in the build
    directory are hard linked to the destination directory.
```

```
    The reason they are hard linked and not copied is for
    speed.  If the hard link fails, it will fall back to a
    copy in case the directories are on different physical
    disks.  If -copy_products is given, however, it will
    always do a copy, not a hard link.  Default is
    project specific:  -nocopy_products for OpenSPARC T1.

-E  Stops after the preprocessing stage.

-cleanup / -nocleanup
  If -cleanup is enabled, then after a successful build,
    the build directory is erased if and only if the build
    directory was created by this invocation of midas.
    Default is project specific: -cleanup for OpenSPARC T1.

-force_cleanup / -noforce_cleanup
    If -cleanup is enabled, but this invocation of midas did
    not create the build directory, -force_cleanup will
    remove the build directory anyway.  Default is
    project specific: -noforce_cleanup for OpenSPARC T1.

-D<symbol> or -D<symbol>=<value>
    Adds a define to the preprocessing line.  Option may be
    repeated.

-stddef / -nostddef
    Includes standard preprocessor definitions on
    command line.  -nostddef disables these.  Default is
    -stddef, but no standard symbols are currently defined.

-I<dir>
    Adds a directory to the include path used by cpp and m4.
    Path should be absolute or relative to the directory
    where midas was invoked.  Option may be repeated.

-stdinc / -nostdinc
    With -stdinc, the standard include paths are used during
    preprocessing (both cpp and m4).  -nostdinc disables
    these. The standard include directories are the directory
  where midas was invoked,the build directory and
    <diag_root>/verif/diag/assembly/include (keep in mind
    that <diag_root> defaults to $DV_ROOT). Default is -stdinc.

-include_build / -noinclude_build
    This option is only meaningful with -nostdinc.  If
    standard includes are switched off, -include_build will
    add the build directory back to the include path.
    Default is -noinclude_build.
```

```
-include_start / -noinclude_start
    This option is only meaningful with -nostdinc.  If
    standard includes are switched off, -include_start will
    add the start directory (the directory where midas was
    invoked) back to the include path.  Default is
    -noinclude_start.

-L<dir>
    Adds a directory to the search path when looking for
    object files in a MIDAS_OBJ directive.  Option may be
    repeated.

-C<dir>
    Adds a directory to the search path when looking for C
    source files in a MIDAS_CC directive.  Option may be
    repeated.

-pal_diag_args <args>
    If the diag is run through pal, gives these arguments to
    the pal diag.  Option may be repeated.  Note that these
    arguments are given to the diag, not pal itself.  For
    instance, "midas -pal_args -abc mydiag.pal
    -pal_diag_args def -pal_diag_args ghi" will run the pal
    command sline "pal -abc mydiag.pal def ghi".

-build_threads <num_threads>
    When doing work that can be done in parallel (such as
    assembling a bunch of files), use <num_threads> to do
    it.  Default is project specific: 3 for OpenSPARC T1.

-print_errors / -noprint_errors
    If -noprint_errors is defined, then generation of error
    messages is turned off.  When used with -verbose 0,
    midas is completly silent.  This is probably only useful
    for the test harness (which is why the switch is there).

-copy_products / -nocopy_products
    If this is set, then copies files from the build directory
    to the starting directory.  With -nocopy_products, the
    files are hard linked instead.  If it tries to create a
    hard link and fails, it will fall back to a copy.
    Default is -nocopy_products.

-compress_image / -nocompress_image
    If -compress_image is enabled (as it is by default),
    then allows compressed mem.images to be generated.  By
    default, all MMU-generated blocks are compressed when
    written to mem.image, meaning that instead of
    initializing unused sections to zero, they are simply
```

```
        uninitialized.  The -nocompress_image is equivalent to
        explicitly putting a 'compressimage=0' in all
        attr_text/attr_data blocks.

    -env_zero / -noenv_zero
        When compressing blocks, if -env_zero is enabled the
        blocks will contain '// zero_image' directives to the
        environment.  These directives are supported only by
        OpenSPARC T1, and they are used to backdoor initialize large
        tracts of memory to zero.  If -noenv_zero is used, then
        compression will simply leave the data uninitialized.

    -default_radix <decimal|hex>
        Radix to assume for all parameters that do not
        explicitly start with '0x'.  Default is 'decimal'.


    -gen_all_tsbs / -nogen_all_tsbs
        If -gen_all_tsbs is given, then all TSBs that are
        defined are written to the memory image.  If
        -nogen_all_tsbs, then generate only the TSBs that are
        used.  Default is project specific: -nogen_all_tsbs for
        OpenSPARC T1.

    -allow_tsb_conflicts / -noallow_tsb_conflicts
        If -allow_tsb_conflicts is enabled, then it is legal to
        have multiple virtual addresses map to the same entry in a
        TSB.  A linked list will be created to hold all entries.
        With -noallow_tsb_conflicts (which is the default for
        N1), collisions in the TSB can only happen with the same
        VA but different contexts.  Default is project specific.

    -allow_empty_sections / -noallow_empty_sections
        If TEXT_VA is specified, then at least one attr_text
        block for the section has to be specified, and the same
        is true for DATA_VA and attr_data blocks.  If
        -allow_empty_sections is specified, then midas will
        allow you to specify a TEXT_VA(DATA_VA) for the section,
        even if the section has no attr_text(attr_data) blocks.
        Of course, any text(data) in such a section will be
        ignored.  Default is project specific:
        -noallow_empty_sections for OpenSPARC T1.

    -allow_duplicate_tags / -noallow_duplicate_tags
        When adding to a TSB link list, it is an error to add
        the same tag twice.  -allow_duplicate_tags suspends the
        error check.  Default is project specific:
        -noallow_duplicate_tags for OpenSPARC T1.
```

```
-allow_illegal_page_sizes / -noallow_illegal_page_sizes
    If -allow_illegal_page_sizes, then tte_size attributes
    are not checked for valid values, though they are still
    checked against the width of the field.  For instance,
    in the OpenSPARC T1 MMU, there are 3 page bits, so values can
    be specified 0-7.  However, the only legal values for
    OpenSPARC T1 are 0, 1, 3, and 5, and unless
    -allow_illegal_page_sizes is in effect, setting page
    bits of 2, 4, 6, or 7 will cause an error.  The default
    is project specific: -noallow_illegal_page_sizes for
    OpenSPARC T1.

-allow_misalgined_tsb_base / -noallow_misaligned_tsb_base
    If -allow_misaligned_tsb_base is set, then a TSB base
    address need not be aligned with the TSB size.
  If an unalgined address is specified
    as the base and -allow_misaligned_tsb_base is specified,
    then midas will forcibly align the address.  Default
    should be -noallow_misaligned_tsb_base for all projects.

 -errcode <error_code>
    Prints a one-line description for the midas error code,
    then exits with status 0.
```

Configuring Commands

midas runs several commands in the course of its operation. Several of these can be configured. The configurable commands are: pal, cpp, m4, gcc, as, and ld. Each configurable command has 3 associated options:

```
-std_<command>_args / -nostd_<command>_args
    When -std_<command>_args is enabled, the standard set of
    arguments for <command> is used.  Default is
    -std_<command>_args

-<command>_args <args>
    Adds <args> to the argument list for the specified
    <command>.

-<command>_cmd <custom_command>
    Uses <custom_command> to run the specifed <command>
    instead of the standard version.
```

Example

For instance, to add -foo to the link line, use my_cpp to preprocess, and not use any standard assembler options, use:

```
midas -ld_args -foo -cpp_cmd my_cpp -nostd_as_args mydiag.s
```

Configuring Filenames

There are several generated files, and they all have default names. You can configure the names of many of the files with the following option:

-file <tag>=<name>
    Causes midas to name the file whose tag is <tag> to be named <name> instead of the default. <name> is treated as the name of a file in the build directory.

Valid tags for the -file option are:

src Local version of the original source code for the diag. Default is diag.src.

s Assembly portion of diag before any preprocessing. Default is diag.s.

pl Perl portion of the diag. Default is diag.pl.

cpp Output of the C preprocessor. Default is diag.cpp.

m4 Output of the m4 preprocessor. Default is diag.m4.

ldscr
    Linker script. Default is diag.ls_scr.

exe Linked executable. Default is diag*.exe where * is application name.

image
    Verilog memory image. Default is mem.image.

events
    Events file Default is diag.ev.

symtab
    Symbol table. Default is symbol.tbl.

goldfinger
    Specification to goldfinger on how to create memory image. Default is diag.goldfinger.

directives
    File to contain midas directives after section splitting. Default is diag.midas.

```
cmdfile
    File to stash the midas command-line.  Default is .midas_args.

oldcmdfile
    File to move old command-line options.  Default is .midas_args.old.

oldm4
    File to stash m4 output of previous run.  Default is
  .midas.diag.m4.old.
```

Running Specific Phases

```
The build process is broken into phases: setup, preprocess,
sectioning, assemble, link, postprocess, copydest, cleanup.
The default behavior is to run all phases.  You can,
however, restrict operation to a selected set of phases.

-start_phase <phase_name>
    Starts with the named phase and run all subsequent phases.

-phase <phase_name>
    Runs the specified phase.  If any -phase or -start_phase
    option exists, then by default all phases are off
    (except for the ones that -phase and -start_phase switch
    on).  You can have multiple -phase options.

-E  This option (mentioned above, which runs the
    preprocessor only) is just a shortcut for "-phase setup
    -phase preprocess").

Keep in mind that running selected phases is caveat emptor.
There are cases where phases expect data or files from
previous phases.
```

Errors

```
When midas is unable to run correctly it will exit with one
of the following error codes.

M_NOERROR (#0): No error.
M_MISC (#1): Miscellaneous error
M_CODE (#2): Error in midas code.
M_DIR (#3): Directory error.
M_FILE (#4): File error.
M_CMDFAIL (#5): Command failed.
M_SECSYNTAX (#6): Error in section syntax.
M_ATTRSYNTAX (#7): Error in attr syntax.
M_MISSINGPARAM (#8): Missing parameter.
```

```
M_ILLEGALPARAM (#9): Illegal parameter.
M_OUTOFRANGE (#10): Out of range.
M_NOTNUM (#11): Not a number.
M_VACOLLIDE (#12): VA collision.
M_PACOLLIDE (#13): PA collision.
M_DIRECTIVESYNTAX (#14): Directive syntax error.
M_GENFAIL (#15): File generation failed.
M_ASMFAIL (#16): Assembler failed.
M_CCFAIL (#17): C compiler failed.
M_LINKFAIL (#18): Linker failed.
M_CPPFAIL (#19): CPP failed.
M_M4FAIL (#20): M4 preprocessor failed.
M_BADCONFIG (#21): Bad configuration.
M_EVENTERR (#22): Event parsing error.
M_ARGERR (#23): Argument error.
M_NOSEC (#24): Undefined section.
M_BADTSB (#25): Bad TSB.
M_BADALIGN (#26): Bad Alignment.
M_EMPTYSECTION (#27): Empty section.
M_TSBSYNTAX (#28): Error in tsb syntax.
M_APPSYNTAX (#29): Error in app syntax.
M_MEMORY (#30): Memory error.
M_GOLDFINGERPARSE (#31): Goldfinger parse error.
M_GOLDFINGERARG (#32): Goldfinger arg error.
M_ELF (#33): ELF error.
M_BADLABEL (#34): Bad label.
M_GOLDFINGERMISC (#35): Uncategorized goldfinger error.
M_GOLDFINGERVERSION (#36): Bad version of goldfinger
M_DUPLICATETAG (#37): Duplicate tags in TSB
M_BLOCKSYNTAX (#38): Error defining goldfinger BLOCK
```

A.3 goldfinger

NAME

goldfinger - Midas' partner for building diags

SYNOPSIS

goldfinger [options]

DESCRIPTION

```
Goldfinger is midas' partner. Goldfinger is implemented in
C and uses libelf for efficient
analysis of ELF files.  In the new regime, midas builds a
linked executable and a command file (i.e., a .goldfinger
file), which are then processed by goldfinger. The final
output files are produced by goldfinger.  It is the
intention that end users never invoke goldfinger directly,
but only through midas.  Nevertheless, users may find a case
whey they need to build a diag in a very non-standard way,
and goldfinger provides a lower-level interface.

Goldfinger is typically used twice in a normal build
process:

Section splitting
    "goldfinger -splitsec <diag_file>" is used to split a
    diag into multiple assembly files, one per section.  All
    embedded midas directives are written to a separate
    file.

Extracting from executable file
    After the executable file is linked, midas needs to
    extract a memory image and a symbol table.  The options
    "goldfinger -in <cmd_file> -genimage -gentsbs
    -gensymtab" will generate these files based on the
    directives in <cmd_file>.
```

Options

```
The options recognized by goldfinger are:

-h  Show usage.

-version
    Print version number and exit.
```

```
-v or -verbose
    Make it more chatty.

-d or -debug
    Make it very chatty.

-silent
    Say nothing unless there's an error.

-n or -nooutput
    Do not write any output files (for debugging only).

-noprint_errors
    Don't print any error messages (usually used with
    -silent).  You can still tell there was an error by the
    exit status.

-prefix <string>
    Prepend <string> to each line of normal output.

-destdir <dir>
    All created files go in this directory (or a relative
    path from it).  The directory specified can be absolute
    or relative from where goldfinger is invoked.

-srcdir <dir>
    If any of the command files specify filenames with
    relative paths, start searching from this directory.
    Note that the command files themselves are always
    specified absolutely or relative to where goldfinger is
    run.
```

Section splitting options

```
The following functions are meaningful when splitting
sections.

-splitsec <file>
    Splits the specified file into sections and writes an
    assembly file for each.  Writes all midas directives
    into a file that must be specified by the -midasfile
    option.

-midasfile <file>
    When doing section splitting, write all midas directives
    into this file.
```

Linked executable options

```
The following options are meaningful when analyzing linked
executables.

-in <command_file>
    Analyzes linked executables based in the directives in
    <command_file> (also referred to as a .goldfinger file).

-genimage
    Generates a memory image based on the linked executable.
    Goes to stdout unless -imagefile is also specified.

-imagefile <file>
    If -genimage is also specified, then redirects output here
    instead of stdout.

-gensymtab
    Generates a symbol table from the linked executable.
    Goes to stdout unless -symtabfile is also specified.

-symtabfile <file>
    If -gensymtab is also specified, then writes the symbol
    table here instead of stdout.

-gentsbs
    Generates TSB programming based on the object files.  It
    is in mem.image format.  It will go to stdout unless
    -imagefile is also specified.

-allow_tsb_conflicts
    If -tsbgen is also provided, then doesn't cause a fatal
    error if there is a collision in the TSB.  Adds to the
    TSB_LINK area instead.

-allow_duplicate_tags
    If -allow_tsb_conflicts is enabled, you are adding
    elements to a TSB_LINK area, and you try to add the same
    tag more than once, it is normally an error.  This
    option disables the error check.  This option is not
    recommended, since the duplicate tag defines a
    translation that can never be used.

-nocompress
    Does not do compression of mem.image sections for any
    sections, regardless of what is in the imagespec file.

-noenvzero
    Does not use the backdoor environment initialization to
```

zero during image compression.

COMMAND FILE SYNTAX

In the command file (i.e., the .goldfinger file), all keywords can be either all uppercase or all lowercase, but not mixed. All numbers are 64-bit numbers. They can be written as decimal (first digit is 1-9), octal (first digit is 0), or hex (begins with 0x). A boolean option can be set to a nonzero number (true) or 0 (false). If a boolean option is named, but not assigned to (e.g, "COMPRESS;" instead of "COMPRESS = 1;"), then it is assigned to 1.

The attrs file is a list of four types of objects at the top level. They can appear on any order:

```
PA_SIZE = num;

APP <name>
  app_lines
END APP

TSB <name>
   tsb_lines
END TSB

TSB_LINK <name>
   tsb_link_lines
END TSB_LINK
```

The PA_SIZE field is the only top-level attribute. It defines the size of a physical address in bits. The default is 40.

All types of block contain two attributes:

SRC_FILE = "file";
    File name where this block is originally defined. Used for error and debugging output.

SRC_LINE = num;
    Line number in SRC_FILE where this block is originally defined. Used for error and debugging output.

APP

An APP object contains a few parameters and a list of block objects. An APP names one linked executable (see ELF_FILE) and a list of blocks that describe what to do with that

```
file.  The APP syntax is:

  APP <appname>

    SRC_FILE = "source file";
    SRC_LINE = <num>;
    ELF_FILE = "executable file";


    BLOCK <name>
       block_attrs
    END BLOCK

    BLOCK <another_name>
       block_attrs
       BLOCK_TSB <name>
          block_tsb_attrs
       END BLOCK_TSB
    END BLOCK

    ...

  END APP

ELF_FILE = "executable file";
    Names the linked executable file (relative to
    srcdir) that will be processed by this APP object.

BLOCK

A BLOCK defines a section of a linked executable that should
be treated the same way.  It can take the following
parameters:

SECTION_NAME = "name";
    Name of the section (e.g., ".MAIN") where this block is
    defined (used for debugging and error reporting).  Used
    only for error reporting.

SEGMENT_NAME = "name";
    Name of segment within the section (e.g., "text") for
    which this block is defined.  Used only for error
    reporting.

LINK_SECTION = "name";
    ELF section name where this block should look in the
    executable.

VA = <num>;
```

```
START_LABEL = "label";
    Optionally specifies that the block should start at a
    particular address or label.  You can specify one or the
    other, but not both.  If neither is specified, then the
    starting VA for the elf section is used.  The starting
    address, however it is specified, must be page aligned
    if it is to be added to a TSB.

END_VA = <num>;
END_LABEL = "label";
    Optionally specifies that the block should end at a
    particular address or label.  You can specify one or the
    other, but not both.  If neither is specified, the the
    ending VA for the elf section is used.

COMPRESS = num;
    Boolean.  If set, then compresses the output of this block
    in the image.  Compression means that if an entire line
    (i.e., aligned 32 bytes) is zero, the line is skipped.
    If -noenvzero is enabled, the 32 bytes are simply
    uninitialized.  Otherwise, the backdoor '// zero_bytes'
    syntax is used to initialize the memory in the
    environment.  The backdoor syntax is specific to
    Niagara, so other projects should either adopt it or use
    -noenvzero.

IN_IMAGE = <boolean>;
    If this is defined and num is zero, then doesn't write
    this block to the memory image.  It is still included in
    the symbol table.

PA = <num>;
    Physical address to write to the image file.  Also used
    in symbol table.

RA = <num>;
    Real address.  Used to write into the TSB and for the
    symbol table.  Written as 'X' in the symbol table if
    this is not specified.

RA_EQ_VA = <boolean>;
    Boolean.  If set, then sets RA to VA (perhaps after VA is
    computed from a label).  It is illegal to set both RA
    and RA_EQ_VA.

PA_EQ_VA = <boolean>;
    Boolean.  If set, then sets PA to VA (perhaps after VA is
    computed from a label).  It is illegal to set both RA
    and PA_EQ_VA.
```

```
NO_END_RANGE_CHECK =  <boolean>;
    If this is set to a nonzero value, then does not do an
    error check to make sure that end_va is not off the end
    of the segment.

BLOCK_TSB <name>
    A BLOCK may contain one or more BLOCK_TSB blocks
    (delimted by "BLOCK_TSB <name>" and "END BLOCK_TSB".  A
    BLOCK_TSB definition names a TSB (see TSB objects below)
    that the block shoudl add itself to.  It also defines
    parameters about how the block should add itself.

BLOCK_TSB
  A BLOCK_TSB object defines how to add its containing block
  to a TSB.  The name of the BLOCK_TSB object is the name of
  the TSB object (see below) that the block should add itself
  to.

TAG_BASE = num;
    Number to use as the basis for TSB tags in this
    attr block.  The virtual address is OR'd into the proper
    bit range in this number (using TAG_ADDR_BITS) to form
    the TSB tags.

DATA_BASE = num;
    Basis for the TSB data entries in this attr
    block.  The real addres is OR'd into the proper bit
    range in this number (using DATA_ADDR_BITS) to form the
    TSB data entries.

START_RA = num;
    Starting real address for this attr block.  Must be page
    aligned.

PAGE_SIZE = num;
    Page size used for computing number of TSB entries and
    for alignment checks.

VA_INDEX_BITS = hi : lo;
    Bits of the virtual address used to index a TSB.  This
    is independant of the TSB size.  If the TSBs being used
    have non-zero size_bits, they will add the size_bits to
    the 'hi' value specified.

TAG_ADDR_BITS = hi : lo;
    Bits of the TSB tag that should contain a
    portion of the virtual address.
```

```
TTE_TAG_ADDR_BITS = hi : lo;
    Bits in the TSB tag that contain the VA.

DATA_ADDR_BITS = hi : lo;
    Bits of the TSB data that should contain a
    portion of the real address.

TSB
  Defines a TSB.  The ATTR blocks define which TSBs they
  want to write to use, and this holds the address
  translations for them.

START_ADDR = num;
    Physical address of where this TSB should live in
    memory.

NUM_ENTRIES = num;
    Number of entries in the TSB.  This can be computed from
    SIZE_BITS for a particular MMU, but goldfinger doesn't
    want to get in the business of interpreting prcoessor-
    specific bit fields.

SIZE_BITS = num;
    Sizes bits from the config register.  It is used in the
    index calculation.

SPLIT = binary;
    If set and num is non-zero, then makes this a split TSB.

LINKAREA = name;
    If there is a collision at any entry in the TSB, it can
    create a linked list.  This parameter names a TSB_LINK
    object (see below) that should contain the linked list.
    If a collision occurs when -allow_tsb_conflicts is not
    set, however, a collision is a fatal error.

TSB_LINK
    This defines a link area.  If there is a collision in the TSB, a
    linked list can be used to track the multiple entries.  This is an
    object for containing that linked list.

START_ADDR = num;
    Physical address where the linked list should live.
```

EXIT STATUS

The exit status will be 0 if the command succeeds. If it fails, it will exit with a positive exit status. The error codes are identical between goldfinger and midas. See "midas -h" for the most up to date description of the errors.

A.4 regreport

NAME

regreport - regression report generator

SYNOPSIS

```
regreport <options> [<directory> [<list>]]
```

DESCRIPTION

regreport examines all regression *.log files for diags under regression directory and prints report. It is called by sims for each diag. User typically calls regreport to generate summary of regression by typing following :

```
regreport <regression_direcotry>

<options>: [default]

  -1 [<diag_dir>]:
      Prints report for the specified or current-directory diag

  -regress <output_file> <directory>:
      In regression mode, regreport writes summary status for finished
      diags to a file until all diags are finished. NOTE: if
      some diag doesn't produce status, regreport,1.73 will wait forever.

  -sas_only
      Verilog simulator will not run, sas only.

  -regenerate
      Regenerates the status.log files in the diag directories.
      Call it from the parent dir of all diag runs e.g. 2004_04_04/

  <directory> [<list>]
      Prints report for all diags under <directory>. <list> is 0 or more
      of simulation system (testbench) names, such as 'core1', 'chip8', etc.
      When nothing specified, all systems are included.

  -simline
      Typically only 1000 last lines of sim.log will be examined.
  -simline=NNN can increase or decrease this number

  -full
```

    The whole file will be processed in regreport -1 mode

  -[no]printpassed
    Does not print passed diags in detailed summary

  -[no]vlog
    Disables vlog run on failing diags. Enabled by default
    If a diag fails we run vlog on it. This is good for automation.

  -debug
    Runs with debug on.

  -summary
    Prints only summary

  -emailaddr=<e-mail address>
    Gives an email address where regression status will be sent.

A.5 vlog

NAME

vlog - post process verilog log file

SYNOPSIS

vlog [logfilename|path_to_sim.log] [-debug -h -ccx -l2 -dram -cycles -[no]sort] [-perf]

DESCRIPTION

vlog is called by regreport and user does not need call it directly. Supported command options are:

  -ccx
    Prints ccx related messages

  -l2
    Prints l2 related messages

  -dram
    Prints dram related messages

  -h
    Prints out this screen

```
  -debug
      Script debug.

  -cycles
      Prints the cycles and not the time

  -sort
      Sorts sim.log according to time stamps first [default is on]

  -perf
      Prints all kinds of performance data - I, D miss e.t.c.
```

Examples:

```
  vlog -ccx -l2 -dram >! vlog.log
  vlog <my_path>/sim.log >! vlog.log
```