

Verification Continuum™

Synopsys Synplify Pro for Lattice

User Guide

October 2024

SYNOPSYS®

Copyright Notice and Proprietary Information

© 2024 Synopsys, Inc. All rights reserved. This software and documentation contain confidential and proprietary information that is the property of Synopsys, Inc. The software and documentation are furnished under a license agreement and may be used or copied only in accordance with the terms of the license agreement. No part of the software and documentation may be reproduced, transmitted, or translated, in any form or by any means, electronic, mechanical, manual, optical, or otherwise, without prior written permission of Synopsys, Inc., or as expressly provided by the license agreement.

Free and Open-Source Licensing Notices

If applicable, Free and Open-Source Software (FOSS) licensing notices are available in the product installation.

Destination Control Statement

All technical data contained in this publication is subject to the export control laws of the United States of America. Disclosure to nationals of other countries contrary to United States law is prohibited. It is the reader's responsibility to determine the applicable regulations and to comply with them.

Disclaimer

SYNOPSYS, INC., AND ITS LICENSORS MAKE NO WARRANTY OF ANY KIND, EXPRESS OR IMPLIED, WITH REGARD TO THIS MATERIAL, INCLUDING, BUT NOT LIMITED TO, THE IMPLIED WARRANTIES OF MERCHANTABILITY AND FITNESS FOR A PARTICULAR PURPOSE.

Trademarks

Synopsys and certain Synopsys product names are trademarks of Synopsys, as set forth at

<http://www.synopsys.com/Company/Pages/Trademarks.aspx>.

All other product or company names may be trademarks of their respective owners.

Third-Party Links

Any links to third-party websites included in this document are for your convenience only. Synopsys does not endorse and is not responsible for such websites and their practices, including privacy practices, availability, and content.

Synopsys, Inc.
690 East Middlefield Road
Mountain View, CA 94043
www.synopsys.com

October 2024

Synopsys Statement on Inclusivity and Diversity

Synopsys is committed to creating an inclusive environment where every employee, customer, and partner feels welcomed. We are reviewing and removing exclusionary language from our products and supporting customer-facing collateral. Our effort also includes internal initiatives to remove biased language from our engineering and working environment, including terms that are embedded in our software and IPs. At the same time, we are working to ensure that our web content and software applications are usable to people of varying abilities. You may still find examples of non-inclusive language in our software or documentation as our IPs implement industry-standard specifications that are currently under review to remove exclusionary language.

Contents

1:: Introduction

Synopsys FPGA and Prototyping Products	14
FPGA Implementation Tools	14
Symphony Model Compiler	16
Rapid Prototyping	17
Synopsys FPGA Tool Features	18
User Interface Overview	21

2:: FPGA Synthesis Design Flows

Logic Synthesis Design Flow	24

3:: Preparing the Input

Setting Up HDL Source Files	28
Creating HDL Source Files	28
Using the Context Help Editor	30
Checking HDL Source Files	32
Editing HDL Source Files with the Built-in Text Editor	33
Setting Editing Window Preferences	36
Using an External Text Editor	38
Using Library Extensions for Verilog Library Files	40
Using Mixed Language Source Files	43
Working with Constraint Files	48
When to Use Constraint Files over Source Code	48
Tcl Syntax Guidelines for Constraint Files	48
Checking Constraint Files	50
Using a Text Editor for Constraint Files (Legacy)	50
Generating Constraint Files for Forward Annotation (Legacy)	51

4:: Setting Up a Logic Synthesis Project

Setting Up Project Files	54

Creating a Project File	54
Opening an Existing Project File	58
Making Changes to a Project	58
Setting Project View Display Preferences	59
Updating Verilog Include Paths in Older Project Files	63
Managing Project File Hierarchy	64
Creating Custom Folders	64
Manipulating Custom Project Folders	67
Manipulating Custom Files	68
Setting Up Implementations	70
Setting Logic Synthesis Implementation Options	72
Setting Device Options	72
Setting Optimization Options	74
Specifying Global Frequency and Constraint Files	76
Specifying Result Options	78
Specifying Timing Report Output	80
Setting Verilog and VHDL Options	80
Specifying Attributes and Directives	86
Specifying Attributes and Directives in VHDL	87
Specifying Attributes and Directives in Verilog	88
Specifying Attributes Using the SCOPE Editor	89
Specifying Attributes in the Constraints File	93
Handling Properties with Attributes or Directives	93
Working with Hierarchical Projects	95
Block-Based Subprojects	95
Creating Hierarchical Subprojects by Exporting Blocks	95
Creating Nested Subprojects	98
Working with Multiple Implementations in Hierarchical Projects	104
Working with Multiple Instances and Parameterized Modules	105
Configuring Synthesis Runs for Hierarchical Projects	108
Analyzing Synthesis Results for Hierarchical Projects	112
Searching Files	114
Identifying the Files to Search	115
Filtering the Files to Search	115
Initiating the Search	116
Search Results	116
Archiving Files and Projects	117
Archive a Project	117
Un-Archive a Project	121

Copy a Project	125
Support for Hierarchical Include Paths	128

5:: Specifying Constraints

Using the SCOPE Editor	132
Creating Constraints in the SCOPE Editor	132
Creating Constraints With the FDC Template Command	136
Specifying SCOPE Constraints	138
Entering and Editing SCOPE Constraints	138
Setting Clock and Path Constraints	140
Defining Input and Output Constraints	142
Specifying Standard I/O Pad Types	143
Specifying Limits for FDC Wildcard Reporting	144
Using the TCL View of SCOPE GUI	145
Guidelines for Entering and Editing Constraints	147
Specifying Timing Exceptions	150
Defining From/To/Through Points for Timing Exceptions	150
Defining Multicycle Paths	154
Defining False Paths	155
Finding Objects with Tcl find and expand	156
Specifying Search Patterns for Tcl find	156
Refining Tcl Find Results with -filter	158
Using the Tcl Find Command to Define Collections	160
Using the Tcl expand Command to Define Collections	161
Checking Tcl find and expand Results	162
Using Tcl find and expand in Batch Mode	164
Using Collections	165
Creating and Using SCOPE Collections	166
Creating Collections using Tcl Commands	168
Viewing and Manipulating Collections with Tcl Commands	171
Converting SDC to FDC	175
Using the SCOPE Editor (Legacy)	176
Entering and Editing SCOPE Constraints (Legacy)	178
Specifying SCOPE Timing Constraints (Legacy)	179
Defining Input and Output Constraints (Legacy)	193
Defining False Paths (Legacy)	194
Generating Constraint Files for Technologies Supported in Radiant Software ...	196

6:: Synthesizing and Analyzing the Results

Synthesizing Your Design	198
Running Logic Synthesis	198
Using Up-to-date Checking for Job Management	198
Checking Log File Results	203
Viewing and Working with the Log File	203
Accessing Specific Reports Quickly	207
Accessing Results Remotely	210
Analyzing Results Using the Log File Reports	213
Using the Watch Window	214
Checking Resource Usage	215
Querying Metrics for a Design	216
Handling Messages	218
Checking Results in the Message Viewer	218
Filtering Messages in the Message Viewer	220
Filtering Messages from the Command Line	222
Automating Message Filtering with a Tcl Script	223
Log File Message Controls	225
Working with Downgradable Errors and Critical Warnings	228

7:: Analyzing with HDL Analyst

Working in the Schematic	232
Opening the Views	233
Cloning Schematics	235
Viewing Object Properties	236
Selecting Objects in the Schematic	239
Grouping Objects in the Schematic	241
Moving Between Views in a Schematic Window	247
Setting Schematic Preferences	247
Exploring Design Hierarchy	249
Traversing Design Hierarchy with the Hierarchy Browser	249
Exploring Object Hierarchy with Push/Pop Commands	252
Finding Objects	257
Browsing to Find Objects in HDL Analyst Views	257
Using Wildcards with the Find Command	265
Crossprobing	266
Crossprobing within a View	266
Crossprobing from an HDL Analyst View	267
Crossprobing from the Text Editor Window	272

Analyzing With the HDL Analyst Tool	274
Viewing Design Hierarchy and Context	274
Filtering Schematics	279
Expanding Pin and Net Logic	281
Dissolving and Partial Dissolving of Buses and Pins	286
Flattening Schematic Hierarchy	289
Using the FSM Viewer	292
Working in the Standard Schematic	296
Differentiating Between the HDL Analyst Views	297
Opening the Views	297
Viewing Object Properties	298
Selecting Objects in the RTL/Technology Views	303
Working with Multisheet Schematics	304
Moving Between Views in a Schematic Window	305
Setting Schematic Preferences	306
Managing Windows	307
Exploring Design Hierarchy	309
Traversing Design Hierarchy with the Hierarchy Browser	309
Exploring Object Hierarchy by Pushing/Poppping	310
Exploring Object Hierarchy of Transparent Instances	316
Finding Objects (Standard)	317
Browsing to Find Objects in HDL Analyst Views	317
Using Find for Hierarchical and Restricted Searches	319
Using Wildcards with the Find Command	322
Combining Find with Filtering to Refine Searches	326
Using Find to Search the Output Netlist	327
Crossprobing (Standard)	330
Crossprobing within an RTL/Technology View	330
Crossprobing from the RTL/Technology View	331
Crossprobing from the Text Editor Window	333
Crossprobing from the Tcl Script Window	336
Crossprobing from the FSM Viewer	336
Analyzing With the Standard HDL Analyst Tool	338
Viewing Design Hierarchy and Context	339
Filtering Schematics	342
Expanding Pin and Net Logic	344
Expanding and Viewing Connections	348
Flattening Schematic Hierarchy	349
Minimizing Memory Usage While Analyzing Designs	353
Using the FSM Viewer (Standard)	355

8:: Analyzing Timing

Analyzing Timing in Schematic Views	360
Viewing Timing Information	360
Annotating Timing Information in the Schematic Views	361
Analyzing Clock Trees in the RTL View	363
Viewing Critical Paths	363
Handling Negative Slack	366
Generating Custom Timing Reports with STA	368
Using Analysis Design Constraints	371
Scenarios for Using Analysis Design Constraints	372
Creating an ADC File	373
Using Object Names Correctly in the adc File	377
Using Auto Constraints	378
Results of Auto Constraints	380

9:: Inferring High-Level Objects

Defining Black Boxes for Synthesis	384
Instantiating Black Boxes and I/Os in Verilog	384
Instantiating Black Boxes and I/Os in VHDL	386
Adding Black Box Timing Constraints	388
Adding Other Black Box Attributes	392
Defining State Machines for Synthesis	393
Defining State Machines in Verilog	393
Defining State Machines in VHDL	394
Specifying FSMs with Attributes and Directives	395
Automatic RAM Inference	398
Block RAM	398
RAM Attributes	399
Inferring Block RAM	401
Initializing RAMs	408
Initializing RAMs in Verilog	408
Initializing RAMs in VHDL	409

10:: Specifying Design-Level Optimizations

Tips for Optimization	414
General Optimization Tips	414
Optimizing for Area	415
Optimizing for Timing	416

Retiming	419
Controlling Retiming	420
Retiming Example	422
Retiming Report	422
How Retiming Works	423
Preserving Objects from Being Optimized Away	426
Using syn_keep for Preservation or Replication	427
Controlling Hierarchy Flattening	429
Preserving Hierarchy	430
Optimizing Fanout	431
Setting Fanout Limits	431
Controlling Buffering and Replication	433
Sharing Resources	435
Inserting I/Os	436
Optimizing State Machines	438
Deciding when to Optimize State Machines	438
Running the FSM Compiler	439
Inserting Probes	443
Specifying Probes in the Source Code	443
Adding Probe Attributes Interactively	444

11:: Working with Compile Points

Compile Point Basics	448
Advantages of Compile Point Design	448
Manual Compile Points	450
Nested Compile Points	451
Compile Point Types	452
Compile Point Synthesis Basics	456
Compile Point Constraint Files	456
Interface Logic Models	459
Interface Timing for Compile Points	459
Compile Point Synthesis	463
Incremental Compile Point Synthesis	464
Forward-annotation of Compile Point Timing Constraints	467
Synthesizing Compile Points	469
The Manual Compile Point Flow	469
Creating a Top-Level Constraints File for Compile Points	472
Defining Manual Compile Points	473

Setting Constraints at the Compile Point Level	476
Analyzing Compile Point Results	478
Resynthesizing Incrementally	480
Resynthesizing Compile Points Incrementally	480

12:: Working with IP Input

Generating IP with SYNCore	484
Specifying FIFOs with SYNCore	484
Specifying RAMs with SYNCore	491
Specifying Byte-Enable RAMs with SYNCore	500
Specifying ROMs with SYNCore	507
Specifying Adder/Subtractors with SYNCore	512
Specifying Counters with SYNCore	520
The Synopsys FPGA IP Encryption Flow	529
Overview of the Synopsys FPGA IP Flow	529
Encryption and Decryption	530
Working with Encrypted IP	534
Encrypting Your IP	534
Encrypting IP with the encryptP1735.pl Script	535
Encrypting IP with the encryptIP Script	541
Specifying the Script Output Method	542
Preparing the IP Package	545
Using Hyper Source	548
Using Hyper Source for Prototyping	548
Using Hyper Source for IP Designs	548
Threading Signals Through the Design Hierarchy of an IP	549
Working with Lattice IP	556

13:: Optimizing Processes for Productivity

Using Batch Mode	558
Running Batch Mode on a Project File	558
Running Batch Mode with a Tcl Script	560
Queuing Licenses	561
Working with Tcl Scripts and Commands	564
Using Tcl Commands and Scripts	564
Generating a Job Script	565
Setting Number of Parallel Jobs	565
Creating a Tcl Synthesis Script	566
Using Tcl Variables to Try Different Clock Frequencies	568

Using Tcl Variables to Try Several Target Technologies	569
Running Bottom-up Synthesis with a Script	570
Automating Flows with synhooks.tcl	571

14:: Improving Runtime

Multiprocessing With Compile Points	576
Setting Maximum Parallel Jobs	576
Specifying Licenses for Multiprocessing	577

15:: Clock Conversion

Working with Gated Clocks	582
Obstacles to Conversion	584
Prerequisites for Gated Clock Conversion	585
Defining Clocks Properly	587
Synthesizing a Gated-Clock Design	592
Accessing the Clock Conversion Report	593
Analyzing the Clock Conversion Report	594
Interpreting Gated Clock Error Messages	596
Disabling Individual Gated Clock Conversions	600
Integrated Clock Gating Cells	600
Using Gated Clocks for Black Boxes	605
OR Gates Driving Latches	606
Obstructions to Optimization	607
Unsupported Constructs	608
Other Potential Workarounds to Solve Clock-Conversion Issues	609
Restrictions on Using Gated Clocks	609
Optimizing Generated Clocks	613
Enabling Generated-Clock Optimization	613
Conditions for Generated-Clock Optimization	613
Generated-Clock Optimization Examples	614

16:: Optimizing for Lattice Designs

Optimizing Lattice Designs	618
Instantiating Lattice Macros	618
Using Lattice GSR Resources	619
Inferring Lattice PIC Latches	620
Controlling I/O Insertion in Lattice Designs	627
Forward-Annotating Lattice Constraints	628

17:: Working with Synthesis Output

Passing Information to the P&R Tools	632
Specifying Pin Locations	632
Generating Vendor-Specific Output	633
Targeting Output to Your Vendor	633
Invoking Third-Party Vendor Tools	634
Configuring Tool Tags	634
Invoking a Third-Party Tool	635

18:: Running Post-Synthesis Operations

Running P&R Automatically after Synthesis	640
Integrating Synthesis and Place-and-Route in One Run	640
Running Place and Route Using a Custom Script	640
Releasing the Synthesis License During Place and Route	641
Simulating with the VCS Tool	643

CHAPTER 1

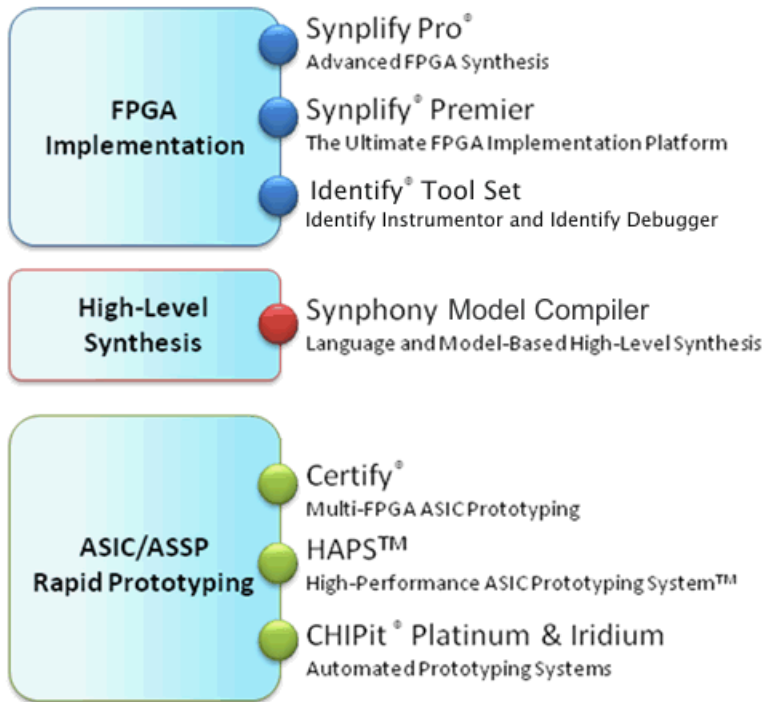
Introduction

This introduction to the Synplify Pro[®] tool describes the following:

- [Synopsys FPGA and Prototyping Products](#), on page 14
- [Scope of the Document](#), on page 21
- [Getting Started](#), on page 22
- [User Interface Overview](#), on page 24

Synopsys FPGA and Prototyping Products

The following figure displays the Synopsys FPGA and Prototyping family of products.



FPGA Implementation Tools

The Synplify Pro and Synplify Premier products are RTL synthesis tools especially designed for FPGAs (field programmable gate arrays) and CPLDs (complex programmable logic devices).

Synplify Pro Synthesis Software

The Synplify Pro FPGA synthesis software is the de facto industry standard for producing high-performance, cost-effective FPGA designs. Its unique Behavior Extracting Synthesis Technology® (B.E.S.T.™) algorithms, perform high-level optimizations before synthesizing the RTL code into specific FPGA logic. This approach allows for superior optimizations across the FPGA, fast runtimes, and the ability to handle very large designs. The software supports the latest VHDL and Verilog language constructs, including SystemVerilog and VHDL 2008. The tool is technology independent allowing quick and easy retargeting between FPGA devices and vendors from a single design project.

Synplify Premier Synthesis Software

The Synplify Premier functionality is a superset of the Synplify Pro tool, providing the ultimate FPGA implementation and debug environment. It includes a comprehensive suite of tools and technologies for advanced FPGA designers, and also serves as the synthesis engine for ASIC prototypers targeting single FPGA-based prototypes.

The Synplify Premier product offers both FPGA designers and ASIC prototypers targeting single FPGAs with the most efficient method of design implementation and debug. On the design implementation side, it includes functionality for timing closure, logic verification, IP usage, ASIC compatibility, and DSP implementation, as well as a tight integration with FPGA vendor back-end tools. On the debug side, it provides for in-system verification of FPGAs which dramatically accelerates the debug process, and also includes a rapid and incremental method for finding elusive design problems.

The Synplify Premier product offers FPGA designers and ASIC prototypers, targeting single FPGA-based prototypes, with the most efficient method of design implementation and debug. The Synplify Premier software provides in-system verification of FPGAs, dramatically accelerates the debug process, and provides a rapid and incremental method for finding elusive design problems. Features exclusively supported in the Synplify Premier tool are the following:

- Fast and Advanced Synthesis Modes
- Design Planning (Optional)
- DesignWare Support
- Unified Power Format (UPF)

Identify Tool Set

The Identify® tool set allows you to instrument and debug an operating FPGA directly in the source RTL code. The Identify software is used to verify your design in hardware as you would in simulation, however much faster and with in-system stimulus. Designers and verification engineers are able to navigate the design graphically and instrument signals directly in RTL with which they are familiar, as probes or sample triggers. After synthesis, results are viewed embedded in the RTL source code or in a waveform. Design iterations are rapidly performed using incremental place and route. Identify software is closely integrated with synthesis and routing tools to create a seamless development environment.

Synphony Model Compiler

Synphony Model Compiler is a language and model-based high-level synthesis technology that provides an efficient path from algorithm concept to silicon. Designers can construct high-level algorithm models from math languages and IP model libraries, then use the Synphony Model Compiler engine to synthesize optimized RTL implementations for FPGA and ASIC architectural exploration and rapid prototyping. In addition, Synphony Model Compiler generates high performance C-models for system validation and early software development in virtual platforms. Key features for this product include:

- MATLAB Language Synthesis
- Automated Fixed-point Conversion Tools
- Synthesizable Fixed-point High Level IP Model Library
- High Level Synthesis Optimizations and Transformations
- Integrated FPGA and ASIC Design Flows
- RTL Testbench Generation
- C-model Generation for Software Development and System Validation

Rapid Prototyping

The Certify® and Identify products are tightly integrated with the HAPS™ and ChipIT® hardware tools.

Certify Product

The Certify software is the leading implementation and partitioning tool for ASIC designers using FPGA-based prototypes to verify their designs. The tool provides a quick and easy method for partitioning large ASIC designs into multi-FPGA prototyping boards. Powerful features allow the tool to adapt easily to existing device flows, therefore, speeding up the verification process and helping with the time-to-market challenges. Key features include the following:

- Graphical User Interface (GUI) Flow Guide
- Manual Partitioning
- Synopsys Design Constraints Support for Timing Management
- Multi-core Parallel Processing Support for Faster Runtimes
- Support for Most Current FPGA Devices
- Industry Standard Synplify Premier Synthesis Support
- Compatible with HAPS Boards Including HSTDM

Synopsys FPGA Tool Features

This table distinguishes between major functionality for the Synopsys FPGA products.

	Synplify	Synplify Pro	Synplify Premier	Synplify Premier DP
Performance				
Behavior Extracting Synthesis Technology® (BEST™)	x	x	x	x
Vendor-Generated Core/IP Support (certain technologies)		x	x	x
FSM Compiler	x	x	x	x
FSM Explorer		x	x	x
Gated Clock Conversion		x	x	x
Register Pipelining		x	x	x
Register Retiming		x	x	x
SCOPE® Constraint Entry	x	x	x	x
High Reliability Features		x	x	x
Integrated Place-and-Route	x	x	x	x
Analysis				
HDL Analyst®	Option	x	x	x
Timing Analyzer – Point-to-point		x	x	x
Timing Report View			x	x
FSM Viewer		x	x	x
Crossprobing		x	x	x
Probe Point Creation		x	x	x
Identify® Instrumentor		x	x	x
Identify Debugger			x	x
Power Analysis (SAIF)			x	x

	Synplify	Synplify Pro	Synplify Premier	Synplify Premier DP
Physical Design				
Design Plan File				x
Logic Assignment to Regions				x
Area Estimation and Region Capacity				x
Pin Assignment				x
Physical Optimizations				x
Physical Analyst			x	x
Synopsys DesignWare® Foundation Library			x	x
Runtime				
Hierarchical Design		x	x	x
Advanced Synthesis			x	x
Fast Synthesis			x	x
Multiprocessing			x	x
Compile on Error			x	x
Team Design				
Mixed Language Design		x	x	x
Compile Points		x	x	x
Hierarchical Design		x	x	x
True Batch Mode (Floating licenses only)		x	x	x
GUI Batch Mode (Floating licenses)	x	x	x	x
Batch Mode P&R	-	x	x	x
Back-annotation of P&R Data	-	-	-	x

	Synplify	Synplify Pro	Synplify Premier	Synplify Premier DP
Formal Verification		x	Logic synthesis mode	Logic synthesis mode
Identify Integration	Limited	x	x	x
Back-annotation of P&R Data				x
Design Environment				
Text Editor View	x	x	x	x
Watch Window		x	x	x
Message Window		x	x	x
Tcl Window		x	x	x
Multiple Implementations		x	x	x
Vendor Technology Support	x	x	Selected	Selected
Prototyping Features				
Runtime features			x	x
Compile Points		x	x	x
Gated Clock Conversion			x	x
Compile on Error			x	x

Scope of the Document

The following explain the scope of this document and the intended audience.

The Document Set

This user guide is part of a document set that includes reference manuals. It is intended for use with the other documents in the set. It concentrates on describing how to use the Synopsys FPGA software to accomplish typical tasks. This implies the following:

- The user guide only explains the options needed to do the typical tasks described in the manual. It does not describe every available command and option. For complete descriptions of all the command options and syntax, refer to the [Tcl Synthesis Commands](#) chapter in the *Synopsys FPGA Synthesis Command Reference Manual*.
- The user guide contains task-based information. For a breakdown of how information is organized, see [Getting Help, on page 22](#).

Audience

The Synplify Pro software tool is targeted towards the FPGA system developer. It is assumed that you are knowledgeable about the following:

- Design synthesis
- RTL
- FPGAs
- Verilog/VHDL

Getting Started

This section shows you how to get started with the Synopsys FPGA synthesis software. It describes the following topics, but does not supersede the information in the installation instructions about licensing and installation:

- [Starting the Software](#), on page 22
- [Getting Help](#), on page 22

Starting the Software

1. If you have not already done so, install the Synopsys FPGA synthesis software according to the installation instructions.
2. Start the software.
 - If you are working on a Windows platform, select Programs->Synopsys->*product version* from the Start button.
 - If you are working on a UNIX platform, type the appropriate command at the command line:

```
synplify_pro
```

The command starts the synthesis tool, and opens the Project window. If you have run the software before, the window displays the previous project. For more information about the interface, see the [User Interface Overview](#) chapter of the *Reference Manual*.

Getting Help

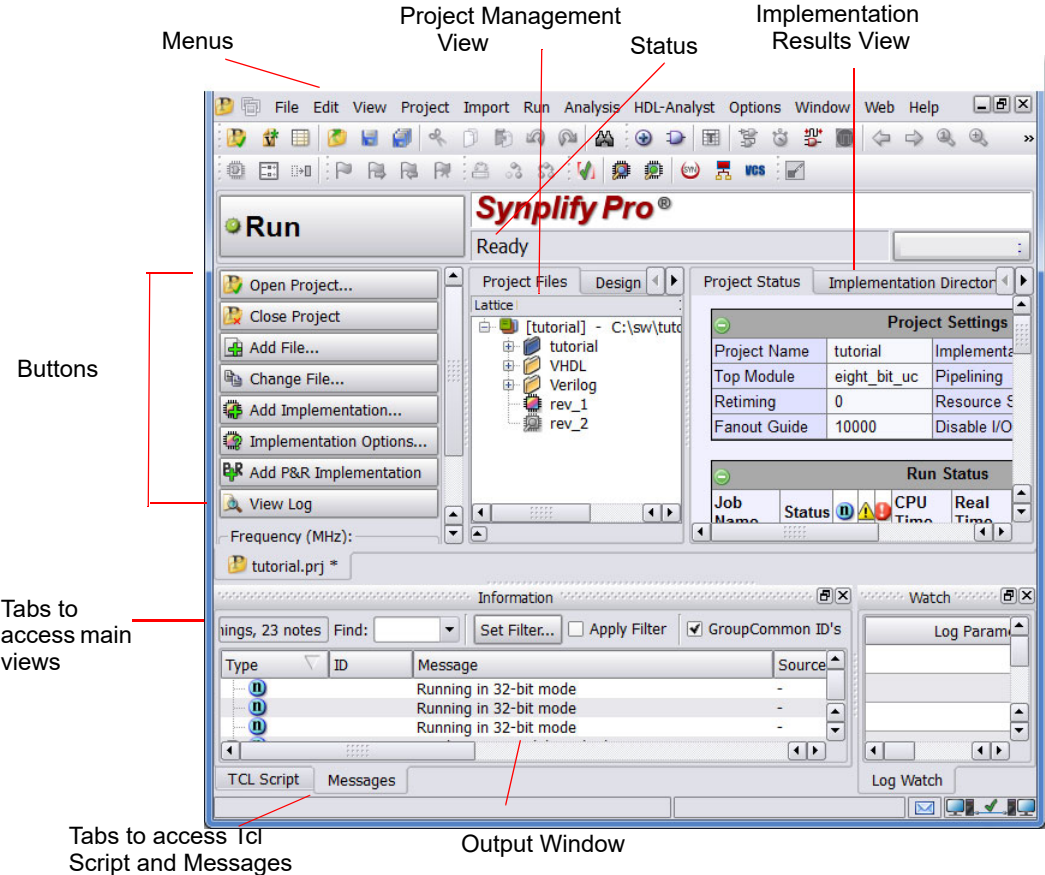
Before you call Synopsys Support, look through the documented information. You can access the information online from the Help menu, or refer to the PDF version. The following table shows you how the information is organized.

For help with...	Refer to the...
Using software features	<i>Synopsys FPGA Synthesis User Guide</i>
How to...	<i>Synopsys FPGA Synthesis User Guide</i> , application notes on the support web site
Flow information	<i>Synopsys FPGA Synthesis User Guide</i> , application notes on the support web site
Error messages	Online help (select Help->Error Messages)
Licensing	Synopsys Website
Attributes and directives	<i>Synopsys FPGA Synthesis Attribute Reference Manual</i>
Synthesis features	<i>Synopsys FPGA Synthesis Reference Manual</i>
Language and syntax	<i>Synopsys FPGA Synthesis Reference Manual</i>
Tcl syntax	Online help (select Help->Tcl Help)
Tcl synthesis commands	<i>Synopsys FPGA Synthesis Command Reference Manual</i>
Product updates	<i>Synopsys FPGA Synthesis Reference Manual</i> (Web menu commands)

User Interface Overview

The graphical user interface (GUI) consists of a main window, called the Project view, and specialized windows or views for different tasks. For details about each of the features, see [Chapter 2, User Interface Overview](#) of the *Synplify Pro FPGA Synthesis Reference Manual*.

Synplify Pro Interface



CHAPTER 2

FPGA Synthesis Design Flows

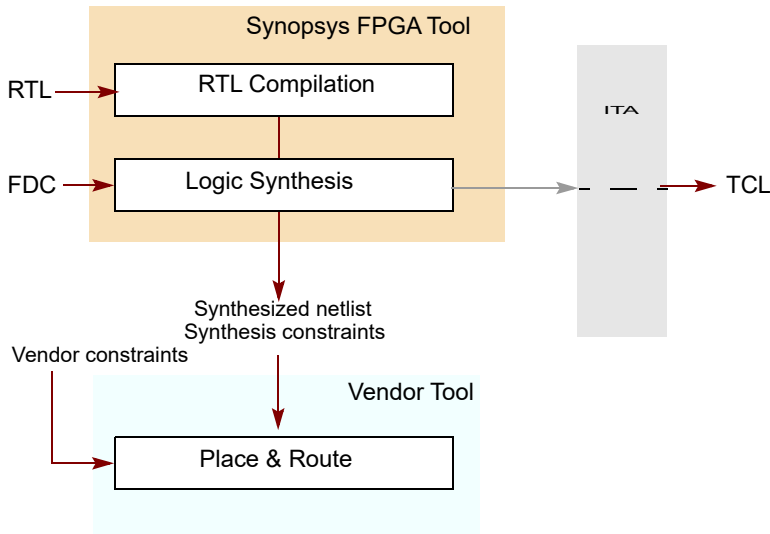
This describes the following tool flows:

- [Logic Synthesis Design Flow](#), on page 24

Logic Synthesis Design Flow

The Synopsys FPGA tools synthesize logic by first compiling the RTL source into technology-independent logic structures, and then optimizing and mapping the logic to technology-specific resources. After logic synthesis, the tool generates a vendor-specific netlist and constraint file that you can use as inputs to the place-and-route (P&R) tool.

The following figure shows the phases and the tools used for logic synthesis and some of the major inputs and outputs. The interactive timing analysis is optional. Although the flow shows the vendor constraint files as direct inputs to the P&R tool, you should add these files to the synthesis project for timing black boxes.



Logic Synthesis Procedure

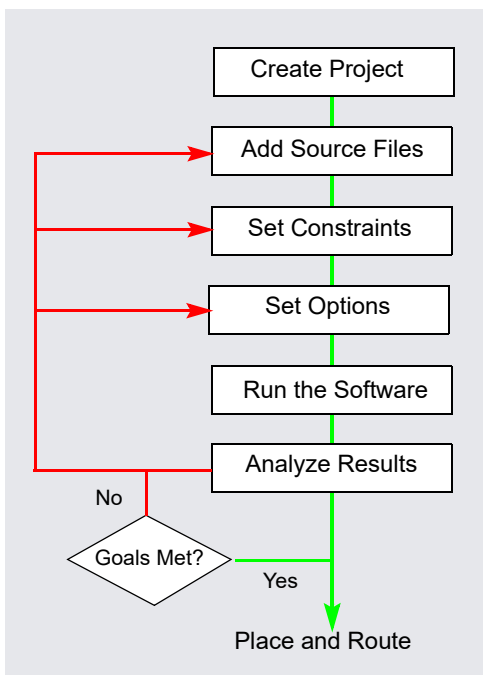
The following steps summarize the procedure for synthesizing the design, which is also illustrated in the figure that follows.

1. Create a project.
2. Add the source files to the project.
3. Set attributes and constraints for the design.

4. Set options for the implementation in the Implementation Options dialog box.
5. Click Run to run logic synthesis.
6. Analyze the results, using tools like the log file, the HDL Analyst schematic views, the Message window and the Watch Window.

After you have completed the design, you can use the output files to run place-and-route with the vendor tool and implement the FPGA.

The following figure lists the main steps in the flow:



CHAPTER 3

Preparing the Input

When you synthesize a design, you need to set up two kinds of files: HDL files that describe your design, and project files to manage the design. This chapter describes the procedures to set up these files and the project. It covers the following:

- [Setting Up HDL Source Files](#), on page 28
- [Using Mixed Language Source Files](#), on page 42
- [Working with Constraint Files](#), on page 47

Setting Up HDL Source Files

This section describes how to set up your source files; project file setup is described in [Setting Up Project Files, on page 54](#). Source files can be in Verilog or VHDL. For information about structuring the files for synthesis, refer to the *Reference Manual*. This section discusses the following topics:

- [Creating HDL Source Files, on page 28](#)
- [Using the Context Help Editor, on page 30](#)
- [Checking HDL Source Files, on page 31](#)
- [Editing HDL Source Files with the Built-in Text Editor, on page 32](#)
- [Setting Editing Window Preferences, on page 35](#)
- [Using an External Text Editor, on page 37](#)
- [Using Library Extensions for Verilog Library Files, on page 39](#)

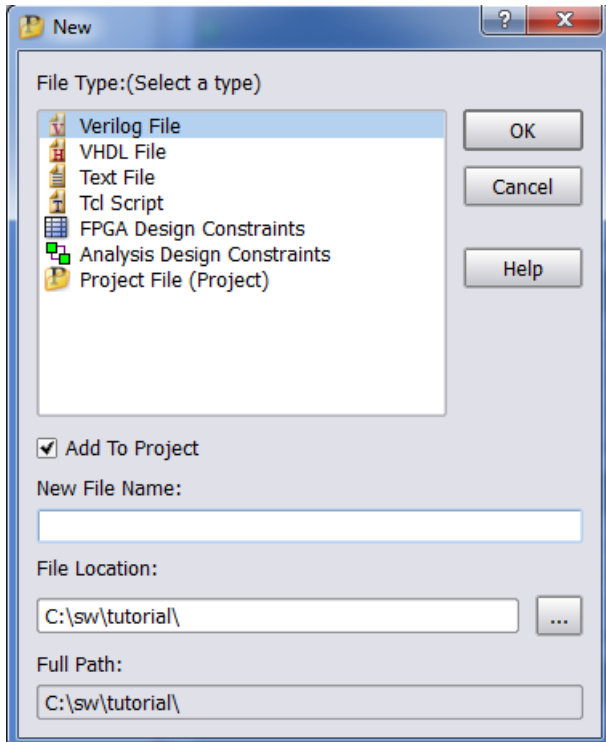
Creating HDL Source Files

This section describes how to use the built-in text editor to create source files, but does not go into details of what the files contain. For details of what you can and cannot include, as well as vendor-specific information, see the *Reference Manual*. If you already have source files, you can use the text editor to check the syntax or edit the file (see [Checking HDL Source Files, on page 31](#) and [Editing HDL Source Files with the Built-in Text Editor, on page 32](#)).

You can use Verilog or VHDL for your source files. The files have `.v` (Verilog) or `.vhd` (VHDL) file extensions, respectively. You can use Verilog and VHDL files in the same design. For information about using a mixture of Verilog and VHDL input files, see [Using Mixed Language Source Files, on page 42](#).

1. To create a new source file either click the HDL file icon () or do the following:
 - Select File->New or press Ctrl-n.
 - In the New dialog box, select the kind of source file you want to create, Verilog or VHDL. Note that you can use the Context Help Editor for designs that contain Verilog, SystemVerilog, or VHDL constructs in the source file. For more information, see [Using the Context Help Editor, on page 30](#).

If you are using Verilog 2001 format or SystemVerilog, make sure to enable the Verilog 2001 or System Verilog option before you run synthesis (Project->Implementation Options->Verilog tab). The default Verilog file format for new projects is SystemVerilog.



- Type a name and location for the file and Click OK. A blank editing window opens with line numbers on the left.
- 2. Type the source information in the window, or cut and paste it. See [Editing HDL Source Files with the Built-in Text Editor, on page 32](#) for more information on working in the Editing window.

For the best synthesis results, check the *Reference Manual* and ensure that you are using the available constructs and vendor-specific attributes and directives effectively.

- 3. Save the file by selecting File->Save or the Save icon ().

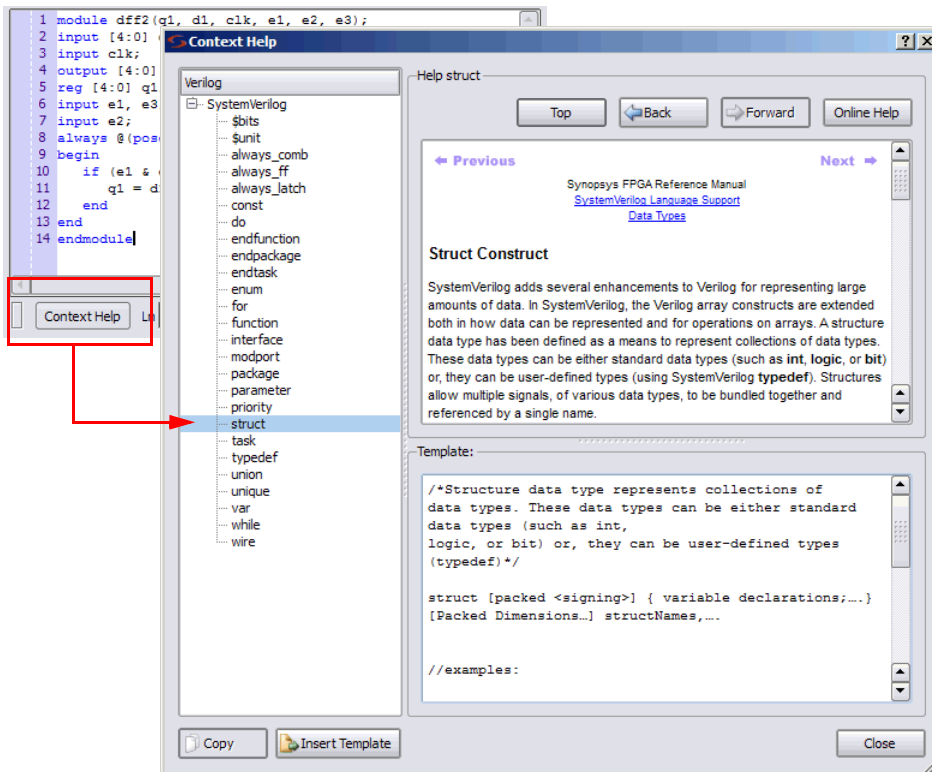
Once you have created a source file, you can check that you have the right syntax, as described in [Checking HDL Source Files, on page 31](#).

Using the Context Help Editor

When you create or open a design file, use the Context Help button displayed at the bottom of the window to help you code with Verilog/SystemVerilog/VHDL constructs in the source file or Tcl constraint commands into your Tcl file.

To use the Context Help Editor:

1. Click on the Context Help button to display this text editor.



2. When you select a construct in the left-side of the window, the online help description for the construct is displayed. If the selected construct has this feature enabled, the online help topic is displayed on the top of the window and a generic code or command template for that construct is displayed at the bottom.
3. The Insert Template button is also enabled. When you click the Insert Template button, the code or command shown in the template window is

inserted into your file at the location of the cursor. This allows you to easily insert the code or command and modify it for the design that you are going to synthesize.

4. If you want to copy only parts of the template, select the code or command you want to insert and click Copy. You can then paste it into your file.

Checking HDL Source Files

The software automatically checks your HDL source files when it compiles them, but if you want to check your source code before synthesis, use the following procedure. There are two kinds of checks you do in the synthesis software: syntax and synthesis.

1. Select the source files you want to check.
 - To check all the source files in a project, deselect all files in the project list, and make sure that none of the files are open in an active window. If you have an active source file, the software only checks the active file.
 - To check a single file, open the file with File->Open or double-click the file in the Project window. If you have more than one file open and want to check only one of them, put your cursor in the appropriate file window to make sure that it is the active window.
2. To check the syntax, select Run->Syntax Check or press Shift+F7.

The software detects syntax errors such as incorrect keywords and punctuation and reports any errors in a separate log file (syntax.log). If no errors are detected, a successful syntax check is reported at the bottom of this file.

3. To run a synthesis check, select Run->Synthesis Check or press Shift+F8.

The software detects hardware-related errors such as incorrectly coded flip-flops and reports any errors in a separate log file (syntax.log). If there are no errors, a successful syntax check is reported at the bottom of this file.

4. Review the errors by opening the syntax.log file when prompted and use Find to locate the error message (search for @E). Double-click on the 5-character error code or click on the message text and push F1 to display online error message help.

5. Locate the portion of code responsible for the error by double-clicking on the message text in the syntax.log file. The Text Editor window opens the appropriate source file and highlights the code that caused the error.
6. Repeat steps 4 and 5 until all syntax and synthesis errors are corrected.

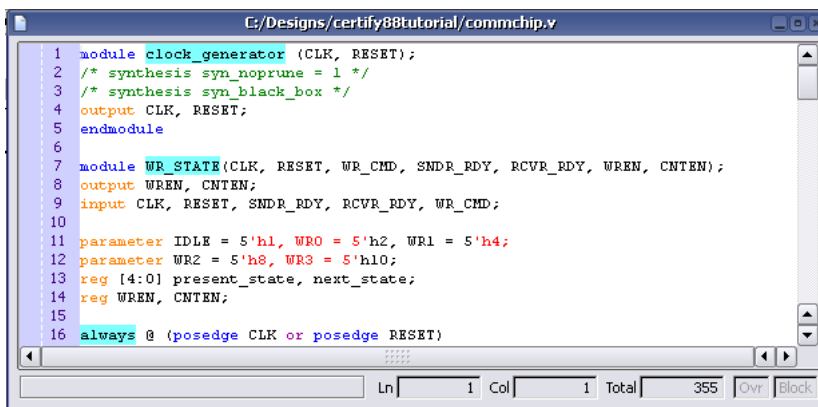
Messages can be categorized as errors, warnings, or notes. Review all messages and resolve any errors. Warnings are less serious than errors, but you must read through and understand them even if you do not resolve all of them. Notes are informative and do not need to be resolved.

Editing HDL Source Files with the Built-in Text Editor

The built-in text editor makes it easy to create your HDL source code, view it, or edit it when you need to fix errors. If you want to use an external text editor, see [Using an External Text Editor, on page 37](#).

1. Do one of the following to open a source file for viewing or editing:
 - To automatically open the first file in the list with errors, press F5.
 - To open a specific file, double-click the file in the Project window or use File->Open (Ctrl-o) and specify the source file.

The Text Editor window opens and displays the source file. Lines are numbered. Keywords are in blue, and comments in green. String values are in red. If you want to change these colors, see [Setting Editing Window Preferences, on page 35](#).



2. To edit a file, type directly in the window.

This table summarizes common editing operations you might use. You can also use the keyboard shortcuts instead of the commands.

To ...	Do ...
Cut, copy, and paste; undo, or redo an action	Select the command from the popup (hold down the right mouse button) or Edit menu.
Go to a specific line	Press Ctrl-g or select Edit->Go To, type the line number, and click OK.
Find text	Press Ctrl-f or select Edit ->Find. Type the text you want to find, and click OK.
Replace text	Press Ctrl-h or select Edit->Replace. Type the text you want to find, and the text you want to replace it with. Click OK.
Complete a keyword	Type enough characters to uniquely identify the keyword, and press Esc.
Indent text to the right	Select the block, and press Tab.
Indent text to the left	Select the block, and press Shift-Tab.
Change to upper case	Select the text, and then select Edit->Advanced ->Uppercase or press Ctrl-Shift-u.
Change to lower case	Select the text, and then select Edit->Advanced ->Lowercase or press Ctrl-u.
Add block comments	Put the cursor at the beginning of the comment text, and select Edit->Advanced->Comment Code or press Alt-c.
Edit columns	Press Alt, and use the left mouse button to select the column. On some platforms, you have to use the key to which the Alt functionality is mapped, like the Meta or diamond key.

3. To cut and paste a section of a PDF document, select the T-shaped Text Select icon, highlight the text you need and copy and paste it into your file. The Text Select icon lets you select parts of the document.
4. To create and work with bookmarks in your file, see the following table.

Bookmarks are a convenient way to navigate long files or to jump to points in the code that you refer to often. You can use the icons in the

Edit toolbar for these operations. If you cannot see the Edit toolbar on the far right of your window, resize some of the other toolbars.

To ...	Do ...
Insert a bookmark	Click anywhere in the line you want to bookmark. Select Edit->Toggle Bookmarks, press Ctrl-F2, or select the first icon in the Edit toolbar. The line number is highlighted to indicate that there is a bookmark at the beginning of that line.
Delete a bookmark	Click anywhere in the line with the bookmark. Select Edit->Toggle Bookmarks, press Ctrl-F2, or select the first icon in the Edit toolbar. The line number is no longer highlighted after the bookmark is deleted.
Delete all bookmarks	Select Edit->Delete all Bookmarks, press Ctrl-Shift-F2, or select the last icon in the Edit toolbar. The line numbers are no longer highlighted after the bookmarks are deleted.
Navigate a file using bookmarks	Use the Next Bookmark (F2) and Previous Bookmark (Shift-F2) commands from the Edit menu or the corresponding icons from the Edit toolbar to navigate to the bookmark you want.

5. To fix errors or review warnings in the source code, do the following:
 - Open the HDL file with the error or warning by double-clicking the file in the project list.
 - Press F5 to go to the first error, warning, or note in the file. At the bottom of the Editing window, you see the message text.
 - To go to the next error, warning, or note, select Run->Next Error/Warning or press F5. If there are no more messages in the file, you see the message “No More Errors/Warnings/Notes” at the bottom of the Editing window. Select Run->Next Error/Warning or press F5 to go to the the error, warning, or note in the next file.
 - To navigate back to a previous error, warning, or note, select Run->Previous Error/Warning or press Shift-F5.

6. To bring up error message help for a full description of the error, warning, or note:
 - Open the text-format log file (click View Log) and either double click on the 5-character error code or click on the message text and press F1.
 - Open the HTML log file and click on the 5-character error code.
 - In the Tcl window, click the Messages tab and click on the 5-character error code in the ID column.
7. To cross probe from the source code window to other views, open the view and select the piece of code. See [Crossprobing from the Text Editor Window, on page 331](#) for details.
8. When you have fixed all the errors, select File->Save or click the Save icon to save the file.

Setting Editing Window Preferences

You can customize the fonts and colors used in a Text Editing window.

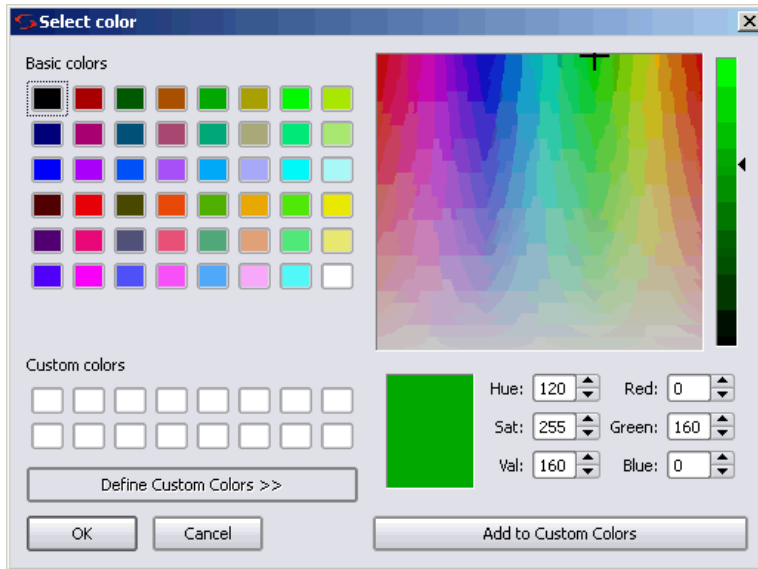
1. Select Options->Editor Options and either Synopsys Editor or External Editor. For more information about the external editor, see [Using an External Text Editor, on page 37](#).
2. Then depending on the type of file you open, you can to set the background, syntax coloring, and font preferences to use with the text editor.

Note: Thereafter, text editing preferences you set for this file will apply to all files of this file type.

The Text Editing window can be used to set preferences for project files, source files (Verilog/VHDL), log files, Tcl files, constraint files, or other default files from the Editor Options dialog box.

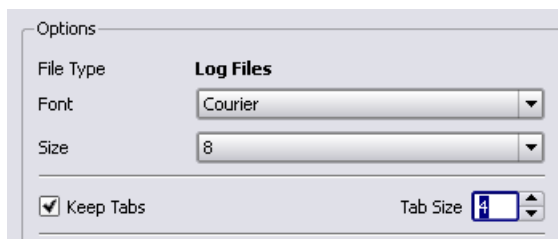
3. You can set syntax colors for some common syntax options, such as keywords, strings, and comments. For example in the log file, warnings and errors can be color-coded for easy recognition.

Click in the Foreground or Background field for the corresponding object in the Syntax Coloring field to display the color palette.



You can select basic colors or define custom colors and add them to your custom color palette. To select your desired color click OK.

4. To set font and font size for the text editor, use the pull-down menus.
5. Check **Keep Tabs** to enable tab settings, then set the tab spacing using the up or down arrow for **Tab Size**.

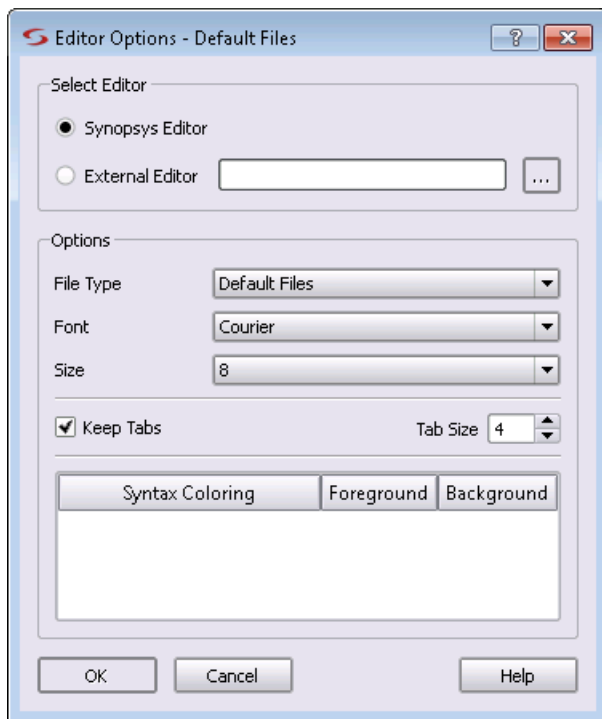


6. Click OK on the Editor Options form.

Using an External Text Editor

You can use an external text editor like `vi` or `emacs` instead of the built-in text editor. Do the following to enable an external text editor. For information about using the built-in text editor, see [Editing HDL Source Files with the Built-in Text Editor](#), on page 32.

1. Select Options->Editor Options and turn on the External Editor option.
2. Select the external editor, using the method appropriate to your operating system.
 - If you are working on a Windows platform, click the ... (Browse) button and select the external text editor executable.
 - From a UNIX or Linux platform for a text editor that creates its own window, click the ... Browse button and select the external text editor executable.
 - From a UNIX platform for a text editor that does not create its own window, do not use the ... Browse button. Instead type `xterm -e editor`. The following figure shows `VI` specified as the external editor.



- From a Linux platform, for a text editor that does not create its own window, do not use the ... Browse button. Instead, type `gnome-terminal -x editor`. To use emacs for example, type `gnome-terminal -x emacs`.

The software has been tested with the emacs and vi text editors.

3. Click OK.

Using Library Extensions for Verilog Library Files

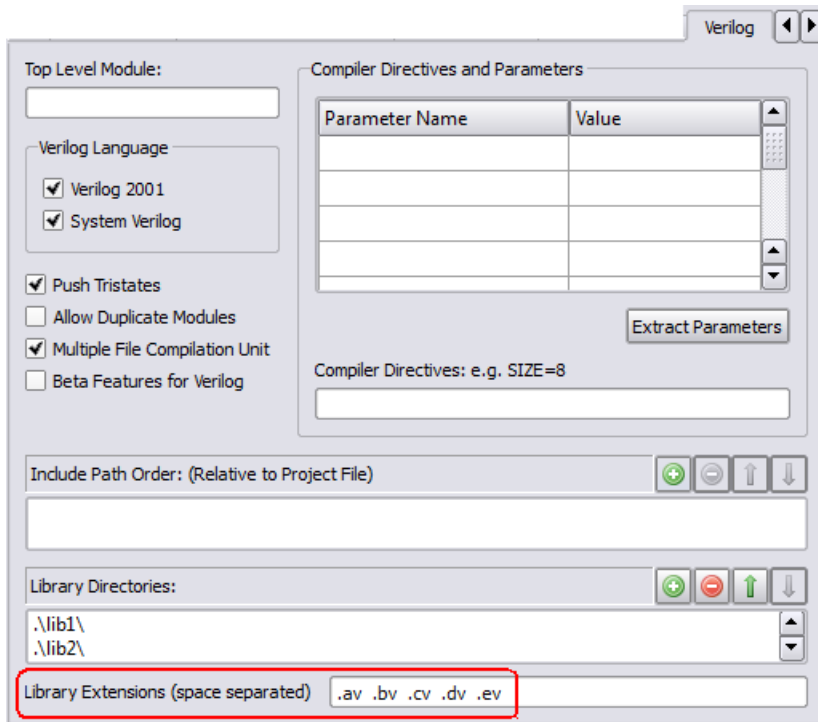
Library extensions can be added to Verilog library files included in your design for the project. When you provide search paths to the directories that contain the Verilog library files, you can specify these new library extensions as well as the Verilog and SystemVerilog (.v and .sv) file extensions.

To do this:

1. Select the Verilog tab of the Implementation Options panel.
2. Specify the locations of the Library Directories for the Verilog library files to be included in your design for the project.
3. Specify the Library Extensions.

Any library extensions can be specified, such as .av, .bv, .cv, .xxx, .va, .vas (separate library extensions with a space).

The following figure shows you where to enter the library extensions on the dialog box.



The Tcl equivalent for this example is the following command:

```
set_option -libext .av .bv .cv .dv .ev
```

For details, see [libext](#), on page 113 in the *Command Reference*.

- After you compile the design, you can verify in the log file that the library files with these extensions were loaded and read. For example:

```
@N: Running Verilog Compiler in SystemVerilog mode
@I: "C:\dir\top.v"
@N: CG1180 : "C:\dir\top.v":8:0:8:3|Loading file
C:\dir\lib1\sub1.av from specified library directory
C:\dir\lib1
@I: "C:\dir\lib1\sub1.av"
@N: CG1180 : "C:\dir\top.v":10:0:10:3|Loading file
C:\dir\lib2\sub2.bv from specified library directory
C:\dir\lib2
@I: "C:\dir\lib2\sub2.bv"
@N: CG1180 : "C:\dir\top.v":12:0:12:3|Loading file
C:\dir\lib3\sub3.cv from specified library directory
```



```
C:\dir\lib3
@I::"C:\dir\lib3\sub3.cv"
@N: CG1180 : "C:\dir\top.v":14:0:14:3|Loading file
C:\dir\lib4\sub4.dv from specified library directory
C:\dir\lib4
@I::"C:\dir\lib4\sub4.dv"
@N: CG1180 : "C:\dir\top.v":16:0:16:3|Loading file
C:\dir\lib5\sub5.ev from specified library directory
C:\dir\lib5
@I::"C:\dir\lib5\sub5.ev"
Verilog syntax check successful!
```

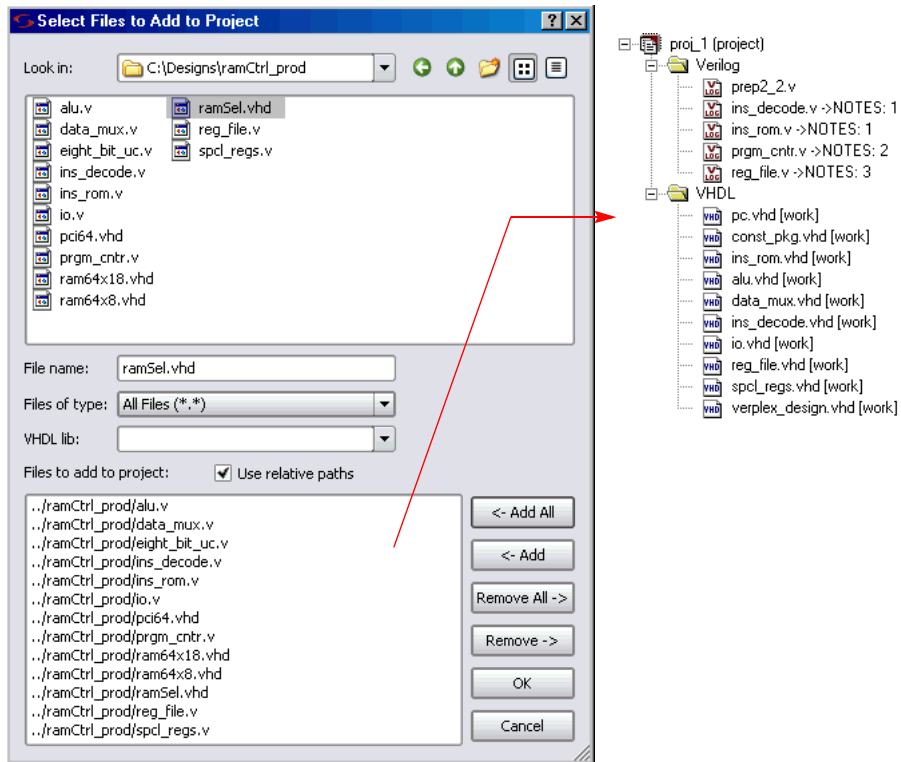
Using Mixed Language Source Files

With the synthesis software, you can use a mixture of VHDL and Verilog input files in your project. For examples of the VHDL and Verilog files, see the *Reference Manual*. You cannot use Verilog and VHDL files together in the same design with the Synplify tool.

1. Remember that Verilog does not support unconstrained VHDL ports and set up the mixed language design files accordingly.
2. If you want to organize the Verilog and VHDL files in different folders, select Options->Project View Options and toggle on the View Project Files in Folders option.

When you add the files to the project, the Verilog and VHDL files are in separate folders in the Project view.

3. When you open a project or create a new one, add the Verilog and VHDL files as follows:
 - Select the Project->Add Source File command or click the Add File button.
 - On the form, set Files of Type to HDL Files (*.vhd, *.vhd, *.v).
 - Select the Verilog and VHDL files you want and add them to your project. Click OK. For details about adding files to a project, see [Making Changes to a Project, on page 58](#).



The files you added are displayed in the Project view. This figure shows the files arranged in separate folders.

4. When you set device options (Implementation Options button), specify the top-level module. For more information about setting device options, see [Setting Logic Synthesis Implementation Options, on page 72](#).
 - If the top-level module is Verilog, click the Verilog tab and type the name of the top-level module.
 - If the top-level module is VHDL, click the VHDL tab and type the name of the top-level entity. If the top-level module is not located in the default work library, you must specify the library where the compiler can find the module. For information on how to do this, see [VHDL Panel, on page 374](#).

The image shows two tabs from a configuration window: 'Verilog' and 'VHDL'.

Verilog Tab:

- Top Level Module:** eight_bit_uc
- Verilog Language:**
 - ☒ Verilog 2001
 - ☐ System Verilog
- ☒ Push Tristates
- ☐ Allow Duplicate Modules
- ☐ Multiple File Compilation Unit
- ☐ Beta Features for Verilog

Compiler Directives and Parameters:

Parameter Name	Value

Extract Parameters

Compiler Directives: e.g. SIZE=8

VHDL Tab:

- Top Level Entity:** eight_bit_uc
- Default Enum Encoding:** default
- ☒ Push Tristates
- ☐ Synthesis On/Off Implemented as Translate On/Off
- ☐ VHDL 2008

You must explicitly specify the top-level module, because it is the starting point from which the mapper generates a merged netlist.

5. Select the Implementation Results tab on the same form and select one output HDL format for the output files generated by the software. For more information about setting device options, see [Setting Logic Synthesis Implementation Options, on page 72](#).
 - For a Verilog output netlist, select Write Verilog Netlist.
 - For a VHDL output netlist, select Write VHDL Netlist.
 - Set any other device options and click OK.

You can now synthesize your design. The software reads in the mixed formats of the source files and generates a single `vm` file that is used for synthesis.

6. If you run into problems, see [Troubleshooting Mixed Language Designs, on page 45](#) for additional information and tips.

Troubleshooting Mixed Language Designs

This section provides tips on handling specific situations that might come up with mixed language designs.

VHDL File Order

For VHDL-only designs or mixed designs where the top level is not specified, the FPGA synthesis tools automatically re-arrange the VHDL files so that the VHDL packages are compiled in the correct order.

However, if you have a mixed-language design where you have specified the top level, you must specify the VHDL file order for the tool. You only need to do this once, by selecting the Run->Arrange VHDL files command. If you do not do this, you get an error message

VHDL Global Signals

Currently, you cannot have VHDL global signals in mixed language designs, because the tool only implements these signals in VHDL-only designs.

Passing VHDL Boolean Generics to Verilog Parameters

The tool infers a black box for a VHDL component with Boolean generics, if that component is instantiated in a Verilog design. This is because Verilog does not recognize Boolean data types, so the Boolean value must be represented correctly. If the value of the VHDL Boolean generic is TRUE and the Verilog literal is represented by a 1, the Verilog compiler interprets this as a black box.

To avoid inferring a black box, the Verilog literal for the VHDL Boolean generic set to TRUE must be 1'b1, not 1. Similarly, if the VHDL Boolean generic is FALSE, the corresponding Verilog literal must be 1'b0, not 0. The following example shows how to represent Boolean generics so that they correctly pass the VHDL-Verilog boundary, without inferring a black box.

VHDL Entity Declaration

```
Entity abc is
Generic
(
  Number_Bits : integer := 0;
  Divide_Bit  : boolean := False;
);
```

Verilog Instantiation

```
abc #(
  .Number_Bits (16),
  .Divide_Bit  (1'b0)
)
```

Passing VHDL Generics Without Inferring a Black Box

In the case where a Verilog component parameter, (for example [0:0] RSR = 1'b0) does not match the size of the corresponding VHDL component generic (RSR : integer := 0), the tool infers a black box.

You can work around this by removing the bus width notation of [0:0] in the Verilog files. You must use a VHDL generic of type integer because the other types do not allow for the proper binding of the Verilog component.

Instantiating Verilog Components

When instantiating Verilog modules in VHDL, you must use component instantiation. Direct/entity instantiation is not supported.

Working with Constraint Files

Constraint files are text files that are automatically generated by the SCOPE interface (see [Specifying SCOPE Constraints, on page 139](#)), or which you create manually with a text editor. They contain Tcl commands or attributes that constrain the synthesis run. Alternatively, you can set constraints in the source code, but this is not the preferred method.

This section contains information about

- [When to Use Constraint Files over Source Code](#), on page 47
- [Tcl Syntax Guidelines for Constraint Files](#), on page 47
- [Checking Constraint Files](#), on page 49
- [Using a Text Editor for Constraint Files \(Legacy\)](#), on page 49
- [Generating Constraint Files for Forward Annotation \(Legacy\)](#), on page 50

When to Use Constraint Files over Source Code

You can add constraints in constraint files (generated by SCOPE interface or entered in a text editor) or in the source code. In general, it is better to use constraint files, because you do not have to recompile for the constraints to take effect. It also makes your source code more portable. See [Using the SCOPE Editor, on page 132](#) for more information.

However, if you have black box timing constraints like `syn_tco`, `syn_tpd`, and `syn_tsu`, you must enter them as directives in the source code. Unlike attributes, directives can only be added to the source code, not to constraint files. See [Specifying Attributes and Directives, on page 86](#) for more information on adding directives to source code.

Tcl Syntax Guidelines for Constraint Files

This section covers general guidelines for using Tcl for constraint files:

- Tcl is case-sensitive.
- For naming objects:
 - The object name must match the name in the HDL code.
 - Enclose instance and port names within curly braces {}.

- Do not use spaces in names.
- Use the dot (.) to separate hierarchical names.
- In Verilog modules, use the following syntax for instance, port, and net names:

v:cell[*prefix*:]*objectName*

Where *cell* is the name of the design entity, *prefix* is a prefix to identify objects with the same name, *objectName* is an instance path with the dot (.) separator. The prefix can be any of the following:

Prefix (Lower-case)	Object
i:	Instance names
p:	Port names (entire port)
b:	Bit slice of a port
n:	Net names

- In VHDL modules, use the following syntax for instance, port, and net names in VHDL modules:

v:cell[.*view*] [*prefix*:]*objectName*

Where *v*: identifies it as a view object, *lib* is the name of the library, *cell* is the name of the design entity, *view* is a name for the architecture, *prefix* is a prefix to identify objects with the same name, and *objectName* is an instance path with the dot (.) separator. *View* is only needed if there is more than one architecture for the design. See the table above for the prefixes of objects.

- Name matching wildcards are * (asterisk matches any number of characters) and ? (question mark matches a single character). These characters do not match dots used as hierarchy separators. For example, the following string identifies all bits of the *statereg* instance in the *statemod* module:

```
i:statemod.statereg[*]
```


Checking Constraint Files

You can check syntax and other pertinent information on your constraint files using the Constraint Check command. To generate a constraint report, do the following:

1. Create a constraint file and add it to your project.
2. Select Run->Constraint Check.

This command generates a report that checks the syntax and applicability of the timing constraints in the FPGA synthesis constraint files for your project. The report is written to the *projectName_cck.rpt* file and lists the following information:

- Constraints that are not applied
- Constraints that are valid and applicable to the design
- Wildcard expansion on the constraints
- Constraints on objects that do not exist

For details on this report, see [Constraint Checking Report, on page 172](#) of the *Reference Manual*.

Using a Text Editor for Constraint Files (Legacy)

You can use the Legacy SCOPE editor for the SDC constraint files created before release version G-2012.09. However, it is recommended that you translate your SDC files to FDC files to enable the latest version of the SCOPE editor and to utilize the enhanced timing constraint handling in the tool.

If you choose to use the legacy SCOPE editor, this section shows you how to manually create a Tcl constraint file. The software automatically creates this file if you use the legacy SCOPE editor to enter the constraints. The Tcl constraint file only contains general timing constraints. Black box constraints must be entered in the source code. For additional information, see [When to Use Constraint Files over Source Code, on page 47](#).

1. Open a file for editing.
 - Make sure you have closed the SCOPE window, or you could overwrite previous constraints.
 - To create a new file, select File->New, and select the Constraints File (SCOPE) option. Type a name for the file and click OK.

- To edit an existing file, select File->Open, set the Files of Type filter to Constraint Files (.sdc) and open the file you want.
- 2. Follow the syntax guidelines in [Tcl Syntax Guidelines for Constraint Files, on page 47](#).
- 3. Enter the timing constraints you need. For the syntax, see the *Reference Manual*. If you have black box timing constraints, you must enter them in the source code.
- 4. You can also add vendor-specific attributes in the constraint file using `define_attribute`. See [Specifying Attributes in the Constraints File, on page 93](#) for more information.
- 5. Save the file.
- 6. Add the file to the project as described in [Making Changes to a Project, on page 58](#), and run synthesis.

Generating Constraint Files for Forward Annotation (Legacy)

The tool automatically generates vendor-specific constraint files that you can use for forward-annotation. The synthesis constraints are mapped to the appropriate vendor constraints. You can control this process with some attributes as described in the following procedure.

1. For some Lattice designs, set the -from and -to false path and multi-cycle constraints on the Others tab of the SCOPE interface.

For details about these attributes, see the *Attribute Reference Manual*.

2. Select Project->Implementation Options, and check Write Vendor Constraints in the Implementation Results tab.
3. Click OK and run synthesis.

The software converts the synthesis `define_input_delay`, `define_output_delay`, `define_clock` (including the `define_clock` constraints generated by auto constraining), `define_multicycle_path`, `define_false_path`, `define_max_delay`, and global-frequency constraints into corresponding commands in the `$DESIGN_synplyf.lydc` file for Lattice.

Open the Lattice Radiant place-and-route tool, then import the \$DESIGN_synplify.ldc file before running PAR. If a user-created ldc file already exists, Radiant backs it up into ldc.bak and copies the contents of \$DESIGN_synplify.ldc into the \$DESIGN.ldc file which is then used for all computations.

The software converts the following synthesis and global frequency constraints into corresponding commands in the *.ldc file for Lattice Radiant designs.

- define_clock
- define_input_delay
- define_output_delay
- define_false_path
- define_multicycle_path
- define_path_delay

See the *Reference Manual* for details about forward annotation.

CHAPTER 4

Setting Up a Logic Synthesis Project

When you synthesize a design with the Synopsys FPGA synthesis tool, you must set up a project for your design. The following describe the procedures for setting up a project for logic synthesis:

- [Setting Up Project Files](#), on page 54
- [Managing Project File Hierarchy](#), on page 64
- [Setting Up Implementations](#), on page 70
- [Setting Logic Synthesis Implementation Options](#), on page 72
- [Specifying Attributes and Directives](#), on page 86
- [Working with Hierarchical Projects](#), on page 95
- [Searching Files](#), on page 114
- [Archiving Files and Projects](#), on page 117

Setting Up Project Files

This section describes the basics of how to set up and manage a project file for your design, including the following information:

- [Creating a Project File](#), on page 54
- [Opening an Existing Project File](#), on page 58
- [Making Changes to a Project](#), on page 58
- [Setting Project View Display Preferences](#), on page 59
- [Updating Verilog Include Paths in Older Project Files](#), on page 62

For information about working with hierarchical projects, refer to [Working with Hierarchical Projects](#), on page 95.

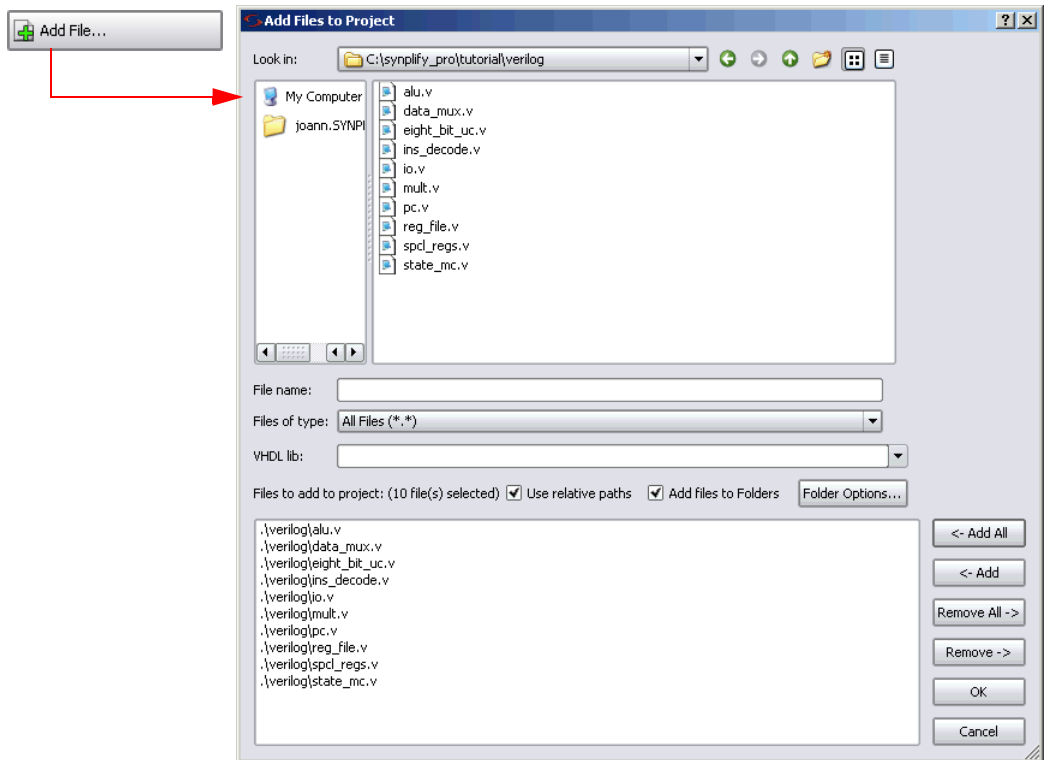
Creating a Project File

You must set up a project file for each project. A project contains the data needed for a particular design: the list of source files, the synthesis results file, and your device option settings. The following procedure shows you how to set up a project file using individual commands.

1. Start by selecting one of the following: File->Build Project, File->Open Project, or the P icon. Click New Project.

The Project window shows a new project. Click the Add File button, press F4, or select the Project->Add Source File command. The Add Files to Project dialog box opens.

2. Add the source files to the project.
 - Make sure the Look in field at the top of the form points to the right directory. The files are listed in the box. If you do not see the files, check that the Files of Type field is set to display the correct file type. If you have mixed input files, follow the procedure described in [Using Mixed Language Source Files](#), on page 42.



- To add all the files in the directory at once, click the Add All button on the right side of the form. To add files individually, click on the file in the list and then click the Add button, or double-click the file name.

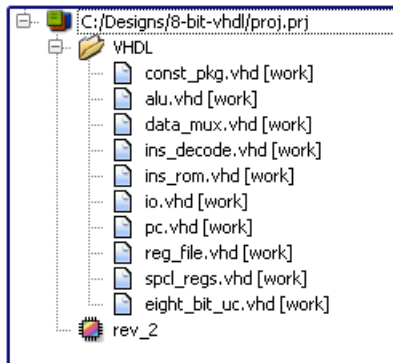
You can add all the files in the directory and then remove the ones you do not need with the Remove button.

If you are adding VHDL files, select the appropriate library from the VHDL Library popup menu. The library you select is applied to all VHDL files when you click OK in the dialog box.

Your project window displays a new project file. If you click on the plus sign next to the project and expand it, you see the following:

- A folder (two folders for mixed language designs) with the source files. If your files are not in a folder under the project directory, you can set this preference by selecting Options->Project View Options and checking the View project files in folders box. This separates one kind of file from another in the Project view by putting them in separate folders.

- The implementation, named `rev_1` by default. Implementations are revisions of your design within the context of the synthesis software, and do not replace external source code control software and processes. Multiple implementations let you modify device and synthesis options to explore design options. Each implementation has its own synthesis and device options and its own project-related files.



3. Add any libraries you need, using the method described in the previous step to add the Verilog or VHDL library file.

- For vendor-specific libraries, add the appropriate library file to the project. Note that for some families, the libraries are loaded automatically and you do not need to explicitly add them to the project file.

To add a third-party VHDL package library, add the appropriate `vhd` file to the design, as described in step 2. Right click the file in the Project view and select File Options, or select Project-> Set VHDL library. Specify a library name that is compatible with the simulators. For example, MYLIB. Make sure that this package library is before the top-level design in the list of files in the Project view.

For information about setting Verilog and VHDL file options, see [Setting Verilog and VHDL Options, on page 80](#). You can also set these file options later, before running synthesis. For additional information about using vendor macro libraries and black boxes, see [Chapter 16, Optimizing for Lattice Designs](#).

- For generic technology components, you can either add the technology-independent Verilog library supplied with the software (`install_dir/lib/generic_technology/gtech.v`) to your design, or add your

own generic component library. Do not use both together as there may be conflicts.

4. Check file order in the Project view. File order is important for all HDL files.
 - Make sure the files are ordered correctly. To re-order files from the UI, drag files to their correct locations in the order. Alternatively, use the `add_file` command in the project file to add the input files in the correct sequence. See [Ordering Input Files, on page 61](#) for details about the order sequence for Verilog and VHDL files.
 - In the Project view, check that the last file in the Project view is the top-level source file. Alternatively, you can specify the top-level file when you set the device options.

An example of correct file order for a design that consists of one top module (`top.v`) and two sub module files (`module1.v` and `module2.v`):

```
add_file -verilog -lib work module1.v
add_file -verilog -lib work module2.v
add_file -verilog -lib work top.v
```

5. Select File->Save, type a name for the project, and click Save. The Project window reflects your changes.
6. To close a project file, select the Close Project button or File->Close Project.

Opening an Existing Project File

There are two ways to open a project file: the Open Project and the generic File ->Open command.

1. If the project you want to open is one you worked on recently, you can select it directly: File->Recent Projects-> *projectName*.
2. Use one of the following methods to open any project file:

Open Project Command	File->Open Command
Select File->Open Project, click the Open Project button on the left side of the Project window, or click the P icon. To open a recent project, double-click it from the list of recent projects. Otherwise, click the Existing Project button to open the Open dialog box and select the project.	Select File->Open. Specify the correct directory in the Look In: field. Set File of Type to Project Files (*.prj). The box lists the project files. Double-click on the project you want to open.

The project opens in the Project window.

Making Changes to a Project

Typically, you add, delete, or replace files.

1. To add source or constraint files to a project, select the Add Files button or Project->Add Source File to open the Select Files to Add to Project dialog box. See [Creating a Project File, on page 54](#) for details.
2. To delete a file from a project, click the file in the Project window, and press the Delete key.
3. To replace a file in a project,
 - Select the file you want to change in the Project window.
 - Click the Change File button, or select Project->Change File.
 - In the Source File dialog box that opens, set Look In to the directory where the new file is located. The new file must be of the same type as the file you want to replace.

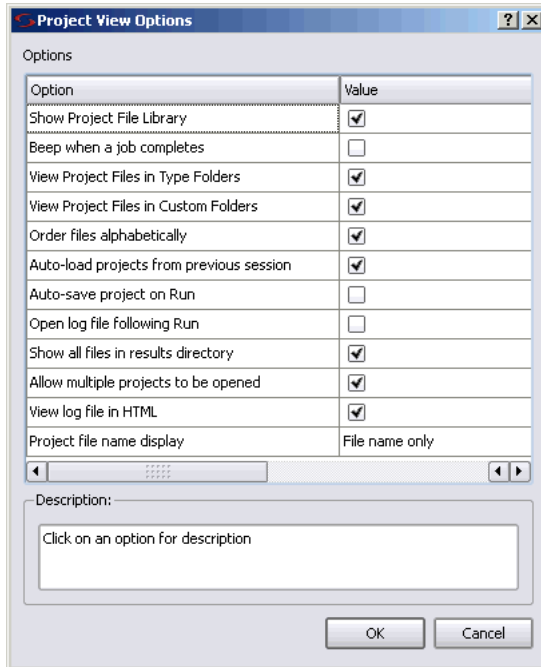
- If you do not see your file listed, select the type of file you need from the Files of Type field.
 - Double-click the file. The new file replaces the old one in the project list.
4. To specify how project files are saved in the project, right click on a file in the Project view and select File Options. Set the Save File option to either Relative to Project or Absolute Path.
 5. To check the time stamp on a file, right click on a file in the Project view and select File Options. Check the time that the file was last modified. Click OK.

Setting Project View Display Preferences

You can customize the organization and display of project files.

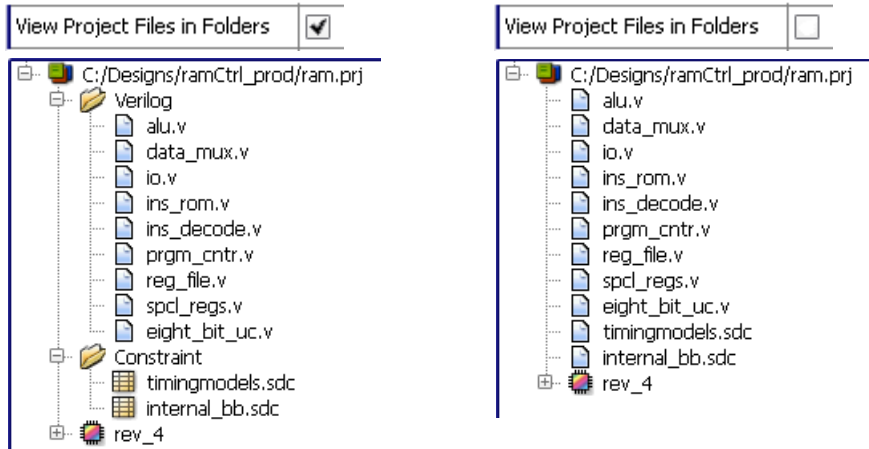
1. Select Options->Project View Options.

The Project View Options form opens.

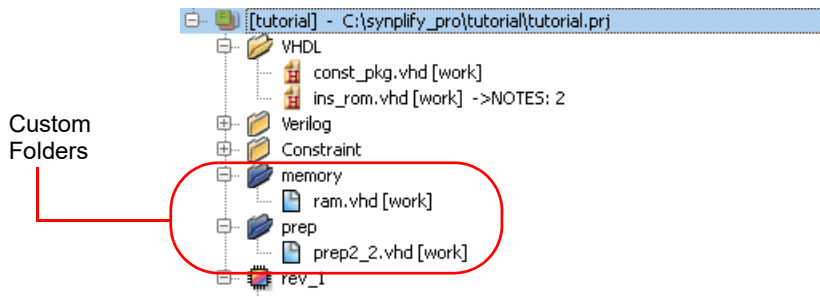


2. To organize different kinds of input files in separate folders, check View Project Files in Folders.

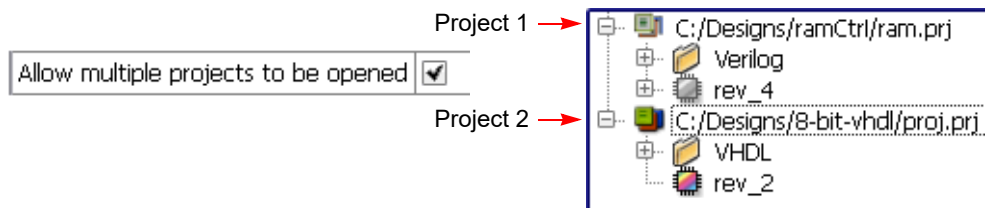
Checking this option creates separate folders in the Project view for constraint files and source files.



3. Control file display with the following:
 - Automatically display all the files, by checking Show Project Library. If this is unchecked, the Project view does not display files until you click on the plus symbol and expand the files in a folder.
 - Check one of the boxes in the Project File Name Display section of the form to determine how filenames are displayed. You can display just the filename, the relative path, or the absolute path.
4. To view project files in customized custom folders, check View Project Files in Custom Folders. For more information, see [Creating Custom Folders, on page 64](#). Type folders are only displayed if there are multiple types in a custom folder.



5. To open more than one implementation in the same Project view, check Allow Multiple Projects to be Opened.



6. Control the output file display with the following:
- Check the Show all Files in Results Directory box to display all the output files generated after synthesis.
 - Change output file organization by clicking in one of the header bars in the Implementation Results view. You can group the files by type or sort them according to the date they were last modified.
7. To view file information, select the file in the Project view, right-click, and select File Options. For example, you can check the date a file was modified.

Ordering Input Files

Correct file order is important for all HDL designs, to ensure that the packages are compiled quickly. Incorrect file order causes unwanted errors/warnings and longer run times. The general rule is to instantiate macros or packages that are referenced by other files and list them before the files that instantiate them. The top-level file must be last on the list.

Do the following to correctly arrange VHDL files:

- To automatically arrange the VHDL files, select the Run->Arrange VHDL option from the menu. Run this command only once.
- To manually arrange the VHDL files, make sure to list the package files first because they must be compiled. Then follow this file order for big designs spread out over multiple FPGAs: entity file, then the architecture file and lastly the configuration file.

Follow this file order when specifying Verilog files for a design:

- For Verilog designs, list package files first because they are compiled before use. After that, list the corresponding HDL files, and list the top-level source file last.
- For SystemVerilog designs, ensure that package, macro, and component files are listed first. Next add the files that instantiate the packages and macros. For example:
 - Package1.sv (package file)
 - Test.sv (import Package1.sv)
 - Top.sv (instantiates Test.sv)

Do the following to arrange files for mixed designs:

- Arrange Verilog/System Verilog files as described above
- Use the Run->Arrange VHDL option from the menu for VHDL files.

Updating Verilog Include Paths in Older Project Files

If you have a project file created with an older version of the software (prior to 8.1), the Verilog include paths in this file are relative to the results directory or the source file with the ``include` statements. In releases after 8.1, the project file ``include` paths are relative to the project file only. The GUI in the more recent releases does not automatically upgrade the older `PRJ` files to conform to the newer rules. To upgrade and use the old project file, do one of the following:

- Manually edit the `PRJ` file in a text editor and add the following on the line before each `set_option -include_path`:

```
set_option -project_relative_includes 1
```

- Start a new project with a newer version of the software and delete the old project. This will make the new PRJ file obey the new rule where includes are relative to the PRJ file.

Managing Project File Hierarchy

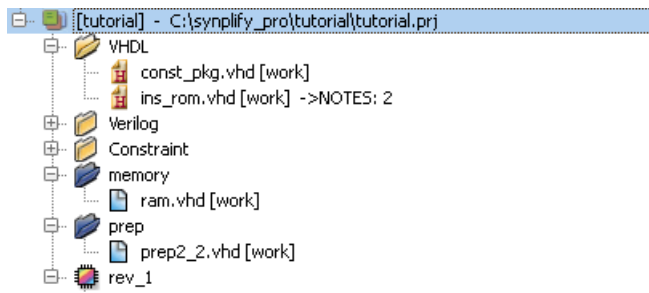
The following sections describe how you can create and manage customized folders and files in the Project view:

- [Creating Custom Folders](#)
- [Manipulating Custom Project Folders](#)
- [Manipulating Custom Files](#)

This information applies to file management; for information about managing and working with hierarchical projects, see [Working with Hierarchical Projects](#), on page 95.

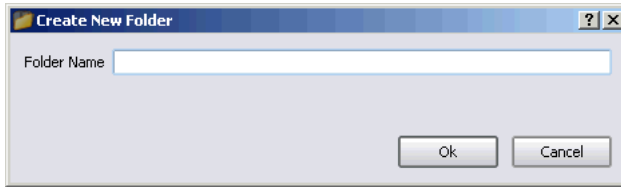
Creating Custom Folders

You can create logical folders and customize files in various hierarchy groupings within your Project view. These folders can be specified with any name or hierarchy level. For example, you can arbitrarily match your operating system file structure or HDL logic hierarchy. Custom folders are distinguished by their blue color.



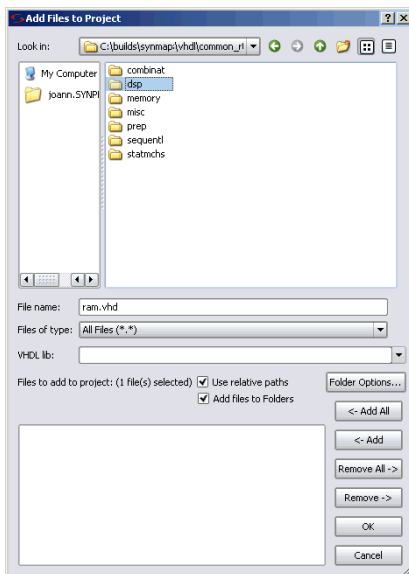
There are several ways to create custom folders and then add files to them in a project. Use one of the following methods:

1. Right-click on a project file or another custom folder and select **Add Folder** from the popup menu. Then perform any of the following file operations:
 - Right-click on a file or files and select **Place in Folder**. A sub-menu displays so that you can either select an existing folder or create a new folder.



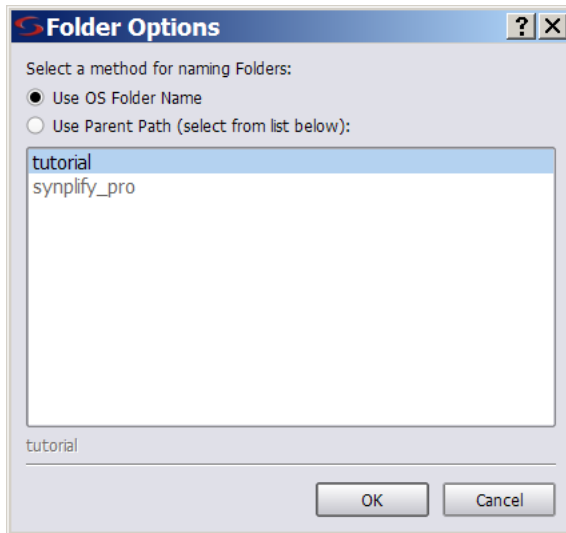
Note that you can arbitrarily name the folder, however do not use the character (/) because this is a hierarchy separator symbol.

- To rename a folder, right-click on the folder and select Rename from the popup menu. The Rename Folder dialog box appears; specify a new name.
2. Use the Add Files to Project dialog box to add the entire contents of a folder hierarchy, and optionally place files into custom folders corresponding to the OS folder hierarchies listed in the dialog box display.



- To do this, select the Add File button in the Project view.
- Select any requested folders such as dsp from the dialog box, then click the Add button. This places all the files from the dsp hierarchy into the custom folder you just created.

- To automatically place the files into custom folders corresponding to the OS folder hierarchy, check the option called Add Files to Custom Folders on the dialog box.
- By default, the custom folder name is the same name as the folder containing files or folder to be added to the project. However, you can modify how folders are named, by clicking on the Folders Option button. The following dialog box is displayed.



To use:

- Only the folder containing files for the folder name, click on Use OS Folder Name.
 - The path name to the selected folder to determine the level of hierarchy reflected for the custom folder path.
3. You can drag and drop files and folders from an OS Explorer application into the Project view. This feature is available on Windows and Linux desktops running KDE.
- When you drag and drop a file, it is immediately added to the project. If no project is open, the software creates a project.

- When you drag and drop a file over a folder, it will be placed in that folder. Initially, the Add Files to Project dialog box is displayed asking you to confirm the files to be added to the project. You can click OK to accept the files. If you want to make changes, you can click the Remove All button and specify a new filter or option.

Note: To display custom folders in the Project view, select the Options->Project View Options menu, then enable/disable the check box for View Project Files in Custom Folders on the dialog box.

Manipulating Custom Project Folders

The following procedure describes how you can remove files from folders, delete folders, and change the folder hierarchy.

1. To remove a file from a custom folder, either:
 - Drag and drop it into another folder or onto the project.
 - Highlight the file, right-click and select Remove from Folder from the popup menu.

Do not use the Delete (DEL) key, as this removes the file from the project.
2. To delete a custom folder, highlight it then right-click and select Delete from the popup menu or press the DEL key. When you delete a folder, make one of the following choices:
 - Click Yes to delete the folder and the files contained in the folder from the project.
 - Click No to just delete the folder.
3. To change the hierarchy of the custom folder:
 - Drag and drop the folder within another folder so that it is a sub-folder or over the project to move it to the top-level.
 - To remove the top-level hierarchy of a custom folder, drag and drop the desired sub-level of hierarchy over the project. Then delete the empty root directory for the folder.

For example, if the existing custom folder directory is:

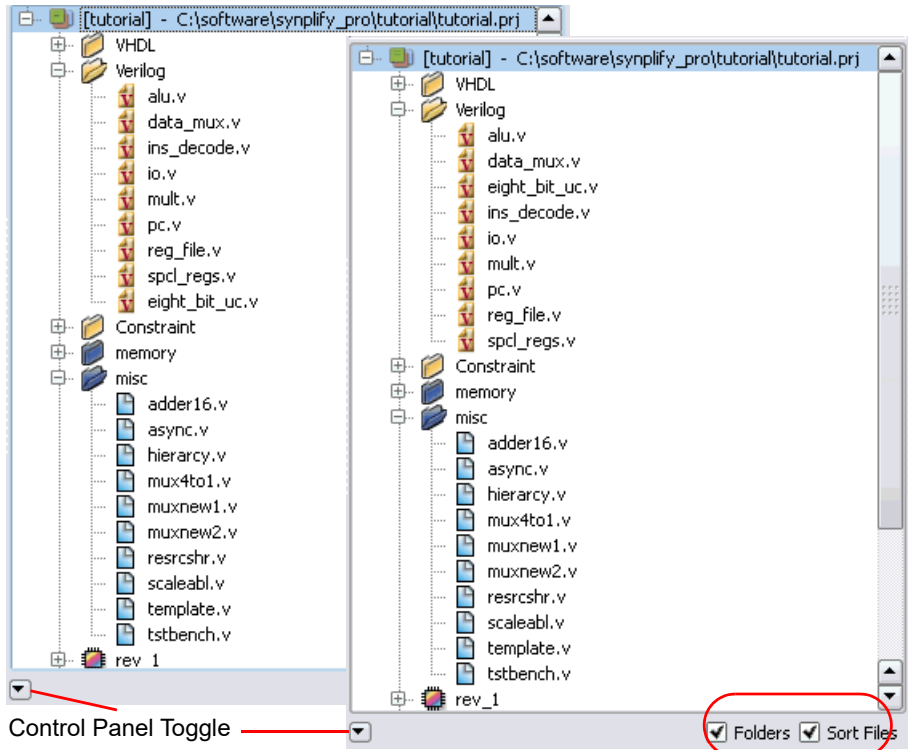
`/Examples/Verilog/RTL`

Suppose you want a single-level RTL hierarchy only, then drag and drop RTL over the project. Thereafter, you can delete the `/Examples/Verilog` directory.

Manipulating Custom Files

Additionally, you can perform the following types of custom file operations:

1. To suppress the display of files in the Type folders, right-click in the Project view and select Project View Options or select Options->Project View Options. Disable the option View Project Files in Type Folders on the dialog box.
2. To display files in alphabetical order instead of project order, check the Sort Files button in the Project view control panel. Click the down arrow key in the bottom-left corner of the panel to toggle the control panel on and off.



3. To change the order of files in the project:
 - Make sure to disable custom folders and sorting files.
 - Drag and drop a file to the desired position in the list of files.
4. To change the file type, drag and drop it to the new type folder. The software will prompt you for verification.

Setting Up Implementations

An implementation is a version of a project, implemented with a specific set of constraints and other settings. A project can contain multiple implementations, each one with its own settings.

Working with Multiple Implementations

The synthesis tools let you create multiple implementations of the same design and then compare results. This lets you experiment with different settings for the same design. Implementations are revisions of your design within the context of the synthesis software, and do not replace external source code control software and processes.

1. Click the Add Implementation button or select Project->New Implementation and set new device options (Device tab), new options (Options tab), or a new constraint file (Constraints tab).

The software creates another implementation in the project view. The new implementation has the same name as the previous one, but with a different number suffix. The following figure shows two implementations, rev1 and rev2, with the current (active) implementation highlighted.



The new implementation uses the same source code files, but different device options and constraints. It copies some files from the previous implementation: the `tlg` log file, the `vm` RTL netlist file. The software keeps a repeatable history of the synthesis runs.

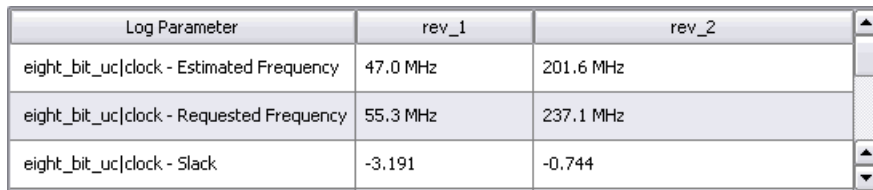
2. Run synthesis again with the new settings.
 - To run the current implementation only, click Run.
 - To run all the implementations in a project, select Run->Run All Implementations.

You can use multiple implementations to try a different part or experiment with a different frequency. See [Setting Logic Synthesis Implementation Options, on page 72](#) for information about setting options.

The Project view shows all implementations with the active implementation highlighted and the corresponding output files generated for the active implementation displayed in the Implementation Results view on the right; changing the active implementation changes the output file display. The Watch window monitors the active implementation. If you configure this window to watch all implementations, the new implementation is automatically updated in the window.

3. Compare the results.

- Use the Watch window to compare selected criteria. Make sure to set the implementations you want to compare with the Configure Watch command. See [Using the Watch Window, on page 214](#) for details.



Log Parameter	rev_1	rev_2
eight_bit_uc clock - Estimated Frequency	47.0 MHz	201.6 MHz
eight_bit_uc clock - Requested Frequency	55.3 MHz	237.1 MHz
eight_bit_uc clock - Slack	-3.191	-0.744

- To compare details, compare the log file results.

4. To rename an implementation, click the right mouse button on the implementation name in the project view, select Change Implementation Name from the popup menu, and type a new name.

Note that the current UI overwrites the implementation; releases prior to 9.0 preserve the implementation to be renamed.

5. To copy an implementation, click the right mouse button on the implementation name in the project view, select Copy Implementation from the popup menu, and type a new name for the copy.
6. To delete an implementation, click the right mouse button on the implementation name in the project view, and select Remove Implementation from the popup menu.

Setting Logic Synthesis Implementation Options

You can set global options for your synthesis implementations, some of them technology-specific. This section describes how to set global options like device, optimization, and file options with the Implementation Options command. For information about setting constraints for the implementation, see [Specifying SCOPE Constraints, on page 139](#). For information about overriding global settings with individual attributes or directives, see [Specifying Attributes and Directives, on page 86](#).

This section discusses the following topics:

- [Setting Device Options, on page 72](#)
- [Setting Optimization Options, on page 74](#)
- [Specifying Global Frequency and Constraint Files, on page 76](#)
- [Specifying Result Options, on page 78](#)
- [Specifying Timing Report Output, on page 80](#)
- [Setting Verilog and VHDL Options, on page 80](#)

Setting Device Options

Device options are part of the global options you can set for the synthesis run. They include the part selection (technology, part and speed grade) and implementation options (I/O insertion and fanouts). The options and the implementation of these options can vary from technology to technology, so check the vendor chapters of the *Reference Manual* for information about your vendor options.

1. Open the Implementation Options form by clicking the Implementation Options button or selecting Project->Implementation Options, and click the Device tab at the top if it is not already selected.
2. Select the technology, part, package, and speed. Available options vary, depending on the technology you choose.



Technology:	Part:	Speed:	Package:
Lattice ECP2M	LFE2M20E	-5	F256C

3. Set the device mapping options. The options vary, depending on the technology you choose.
 - If you are unsure of what an option means, click on the option to see a description in the box below. For full descriptions of the options, click F1 or refer to the appropriate vendor chapter in the *Reference Manual*.
 - To set an option, type in the value or check the box to enable it.

For more information about setting fanout limits, pipelining, and retiming, see [Setting Fanout Limits, on page 430](#), [Retiming, on page 418](#), and [Retiming, on page 418](#), respectively. For details about other vendor-specific options, refer to the appropriate vendor chapter and technology family in the *Reference Manual*.

Device Mapping Options	
Option	Value
Fanout Guide	1000
Disable I/O Insertion	☐
Update Compile Point Timing Data	☐
Automatic Read/Write Check Insertion for RAM	☐
Force GSR Usage	no
Write Declared Clocks Only	☒
Resolve Mixed Drivers	☐

4. Set other implementation options as needed (see [Setting Logic Synthesis Implementation Options, on page 72](#) for a list of choices). Click OK.
5. Click the Run button to synthesize the design. The software compiles and maps the design using the options you set.
6. To set device options with a script, use the `set_option` Tcl command. The following table contains an alphabetical list of the device options on the Device tab mapped to the equivalent Tcl commands. Because the options are technology- and family-based, all of the options listed in the table may not be available in the selected technology. All commands begin with `set_option`, followed by the syntax in the column as shown. Check the *Reference Manual* for the most comprehensive list of options for your vendor.

The following table shows a majority of the device options.

Option	Tcl Command (set_option ...)
Annotated Properties for Analyst This option is supported for Lattice iCE devices.	<code>-run_prop_extract {1 0}</code>
Disable I/O Insertion	<code>-disable_io_insertion {1 0}</code>
Fanout Guide	<code>-fanout_limit <i>fanout_value</i></code>
Package	<code>-package <i>pkg_name</i></code>
Part	<code>-part <i>part_name</i></code>
Resolve Mixed Drivers	<code>-resolve_multiple_driver {1 0}</code>
Speed	<code>-speed_grade <i>speed_grade</i></code>
Technology	<code>-technology <i>keyword</i></code>
Update Compile Point Timing Data	<code>-update_models_cp {0 1}</code>

Setting Optimization Options

Optimization options are part of the global options you can set for the implementation. This section tells you how to set options like frequency and global optimization options like resource sharing. You can also set some of these options with the appropriate buttons on the UI.

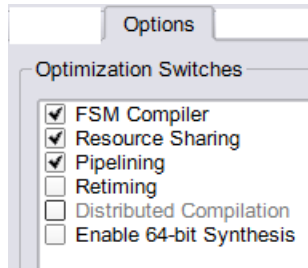
1. Open the Implementation Options form by clicking the Implementation Options button or selecting Project->Implementation Options, and click the Options tab at the top.
2. Click the optimization options you want, either on the form or in the Project view. Your choices vary, depending on the technology. If an option is not available for your technology, it is greyed out. Setting the option in one place automatically updates it in the other.

Optimization Options

Project View

FSM Compiler	☒
Resource Sharing	☒
Pipelining	☒
Retiming	☐

Implementation Options->Options



For details about using these optimizations refer to the following sections:

FSM Compiler	<i>Optimizing State Machines</i> , on page 437

Resource Sharing	<i>Sharing Resources</i> , on page 434

Pipelining	<i>Retiming</i> , on page 418

Retiming	<i>Retiming</i> , on page 418

The equivalent Tcl `set_option` command options are as follows:

Option	<code>set_option</code> Tcl Command Option
FSM Compiler	<code>-symbolic_fsm_compiler {1 0}</code>
Resource Sharing	<code>-resource_sharing {1 0}</code>
Pipelining	<code>-pipe {0 1}</code>
Retiming	<code>-retiming {1 0}</code>

3. Set other implementation options as needed (see [Setting Logic Synthesis Implementation Options, on page 72](#) for a list of choices). Click OK.
4. Click the Run button to run synthesis.

The software compiles and maps the design using the options you set.

Specifying Global Frequency and Constraint Files

This procedure tells you how to set the global frequency and specify the constraint files for the implementation.

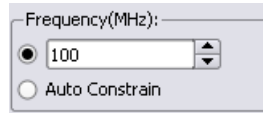
1. To set a global frequency, do one of the following:
 - Type a global frequency in the Project view.
 - Open the Implementation Options form by clicking the Implementation Options button or selecting Project->Implementation Options, and click the Constraints tab.

The equivalent Tcl `set_option` command is `-frequency frequencyValue`.

You can override the global frequency with local constraints, as described in [Specifying SCOPE Constraints, on page 139](#). In the Synplify Pro tool, you can automatically generate clock constraints for your design instead of setting a global frequency. See [Using Auto Constraints, on page 377](#) for details.

Global Frequency and Constraints

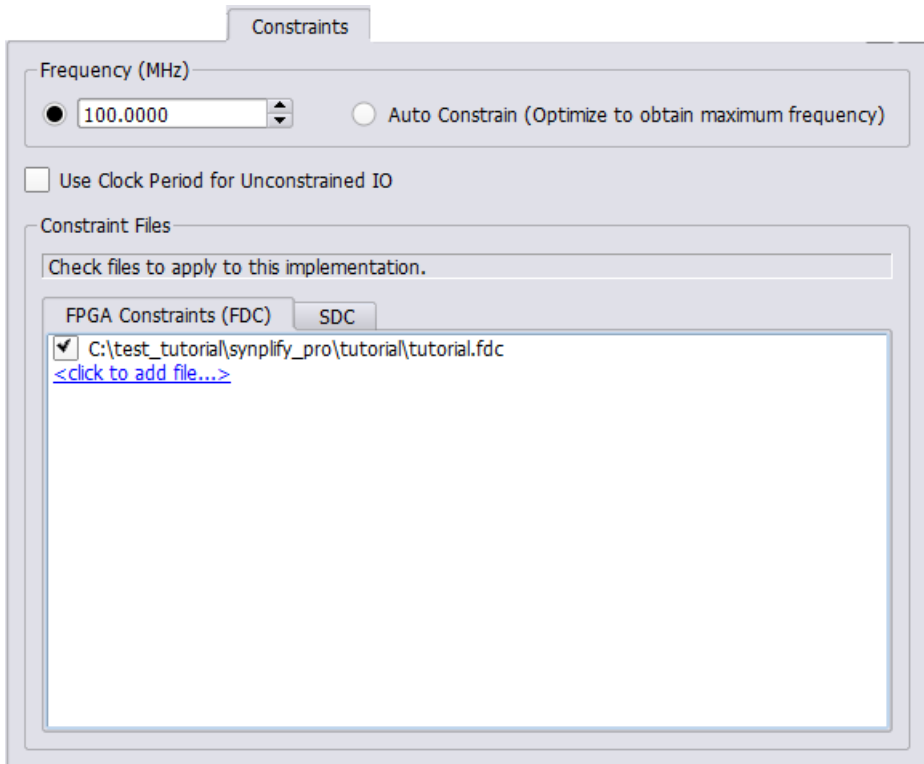
Project View



Frequency(MHz):

☒ 100 ☐ Auto Constrain

Implementation Options->Constraints



Constraints

Frequency (MHz)

☒ 100.0000 ☐ Auto Constrain (Optimize to obtain maximum frequency)

☐ Use Clock Period for Unconstrained IO

Constraint Files

Check files to apply to this implementation.

FPGA Constraints (FDC) SDC

☒ C:\test_tutorial\synplify_pro\tutorial\tutorial.fdc
[<click to add file...>](#)

2. To specify constraint files for an implementation, do one of the following:
 - Select Project->Implementation Options->Constraints. Check the constraint files you want to use in the project.
 - From the Implementation Options->Constraints panel, you can also click to add a constraint file.
 - With the implementation you want to use selected, click Add File in the Project view, and add the constraint files you need.

To create constraint files, see [Specifying SCOPE Constraints, on page 139](#).

3. To remove constraint files from an implementation, do one of the following:
 - Select Project->Implementation Options->Constraints. Click off the checkbox next to the file name.
 - In the Project view, right-click the constraint file to be removed and select Remove from Project.

This removes the constraint file from the implementation, but does not delete it.

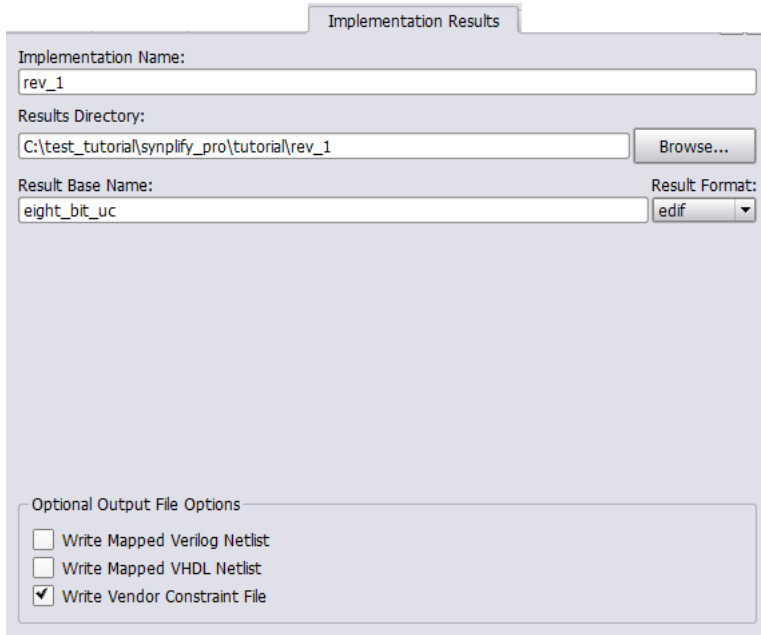
4. Set other implementation options as needed (see [Setting Logic Synthesis Implementation Options, on page 72](#) for a list of choices). Click OK.

When you synthesize the design, the software compiles and maps the design using the options you set.

Specifying Result Options

This section shows you how to specify criteria for the output of the synthesis run.

1. Open the Implementation Options form by clicking the Implementation Options button or selecting Project->Implementation Options, and click the Implementation Results tab at the top.



The image shows a dialog box titled "Implementation Results". It contains the following fields and options:

- Implementation Name:** A text field containing "rev_1".
- Results Directory:** A text field containing "C:\test_tutorial\synplify_pro\tutorial\rev_1" and a "Browse..." button.
- Result Base Name:** A text field containing "eight_bit_uc".
- Result Format:** A dropdown menu currently set to "edif".
- Optional Output File Options:** A section with three checkboxes:
 - ☐ Write Mapped Verilog Netlist
 - ☐ Write Mapped VHDL Netlist
 - ☒ Write Vendor Constraint File

2. Specify the output files you want to generate.
 - To generate mapped netlist files, click Write Mapped Verilog Netlist or Write Mapped VHDL Netlist.
 - To generate a vendor-specific constraint file for forward annotation, click Write Vendor Constraint File. See [Generating Constraint Files for Forward Annotation \(Legacy\)](#), on page 50 for more information.
3. Set the directory to which you want to write the results.
4. Set the format for the output file. The equivalent Tcl command for scripting is `project -result_format format`.

You might also want to set attributes to control name-mapping. For details, refer to the appropriate vendor chapter in the *Reference Manual*.

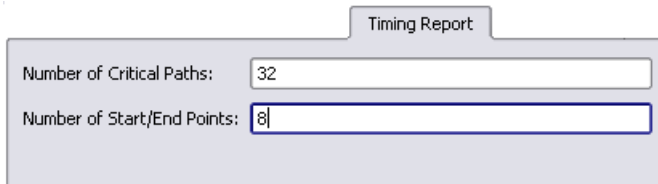
5. Set other implementation options as needed (see [Setting Logic Synthesis Implementation Options](#), on page 72 for a list of choices). Click OK.

When you synthesize the design, the software compiles and maps the design using the options you set.

Specifying Timing Report Output

You can determine how much is reported in the timing report by setting the following options.

1. Selecting Project->Implementation Options, and click the Timing Report tab.
2. Set the number of critical paths you want the software to report.



The screenshot shows a dialog box with a tab labeled "Timing Report". Inside the dialog, there are two input fields. The first field is labeled "Number of Critical Paths:" and contains the value "32". The second field is labeled "Number of Start/End Points:" and contains the value "8".

3. Specify the number of start and end points you want to see reported in the critical path sections.
4. Set other implementation options as needed (see [Setting Logic Synthesis Implementation Options, on page 72](#) for a list of choices). Click OK.

When you synthesize the design, the software compiles and maps the design using the options you set.

Setting Verilog and VHDL Options

When you set up the Verilog and VHDL source files in your project, you can also specify certain compiler options.

Setting Verilog File Options

You set Verilog file options by selecting either Project->Implementation Options->Verilog, or Options->Configure Verilog Compiler.

The screenshot shows the 'Verilog' tab of a configuration window. It includes sections for 'Top Level Module', 'Verilog Language' (with checkboxes for Verilog 2001 and System Verilog), 'Compiler Directives and Parameters' (with a table for parameter overrides and an 'Extract Parameters' button), 'Compiler Directives' (with a text field for directives like SIZE=8), 'Loop Limit' (set to 2000), 'Include Path Order', 'Library Directories', and 'Library Extensions'.

Top Level Module:

Verilog Language

- ☒ Verilog 2001
- ☒ System Verilog
- ☐ Incremental Compile
- ☒ Push Tristates
- ☐ Allow Duplicate Modules
- ☒ Multiple File Compilation Unit
- ☐ Beta Features for Verilog

Loop Limit: 2000

Compiler Directives and Parameters

Parameter Name	Override Value

Extract Parameters

Compiler Directives: e.g. SIZE=8

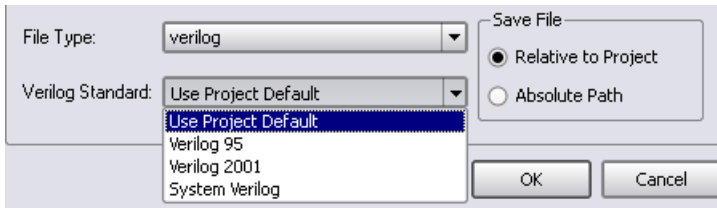
Include Path Order: (Relative to Project File)

Library Directories:

Library Extensions (space separated)

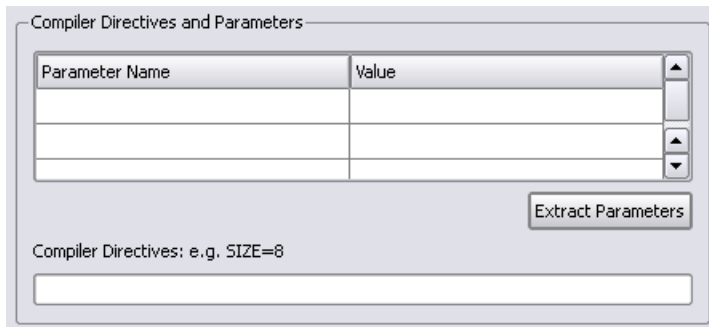
1. Specify the Verilog format to use.

- To set the compiler globally for all the files in the project, select Project->Implementation Options->Verilog. If you are using Verilog 2001 or SystemVerilog, check the *Command Reference Manual* for supported constructs.
- To specify the Verilog compiler on a per file basis, select the file in the Project view. Right-click and select File Options. Select the appropriate compiler. The default Verilog file format for new projects is SystemVerilog.



2. Specify the top-level module if you did not already do this in the Project view.
3. To extract parameters from the source code, do the following:
 - Click Extract Parameters.
 - To override the default, enter a new value for a parameter.

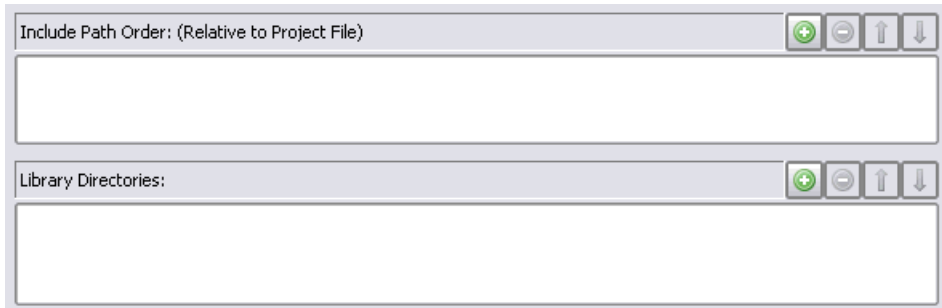
The software uses the new value for the current implementation only. Note that parameter extraction is not supported for mixed designs.



4. Type in the directive in Compiler Directives, using spaces to separate the statements.

You can type in directives you would normally enter with 'ifdef and 'define statements in the code. For example, `ABC=30` results in the software writing the following statements to the project file:

```
set_option -hdl_define -set "ABC=30"
```

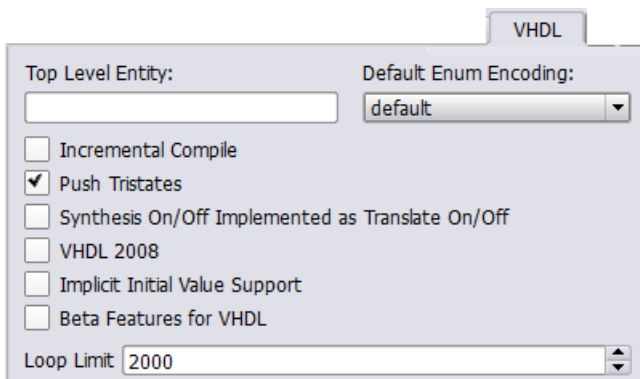


5. In the Include Path Order, specify the search paths for the include commands for the Verilog files that are in your project. Use the buttons in the upper right corner of the box to add, delete, or reorder the paths.
6. In the Library Directories, specify the path to the directory which contains the library files for your project. Use the buttons in the upper right corner of the box to add, delete, or reorder the paths.
7. Set other implementation options as needed (see [Setting Logic Synthesis Implementation Options, on page 72](#) for a list of choices). Click OK.

When you synthesize the design, the software compiles and maps the design using the options you set.

Setting VHDL File Options

You set VHDL file options by selecting either Project->Implementation Options->VHDL, or Options->Configure VHDL Compiler.



For VHDL source, you can specify the options described below.

1. Specify the top-level module if you did not already do this in the Project view. If the top-level module is not located in the default work library, you must specify the library where the compiler can find the module. For information on how to do this, see [VHDL Panel, on page 374](#).

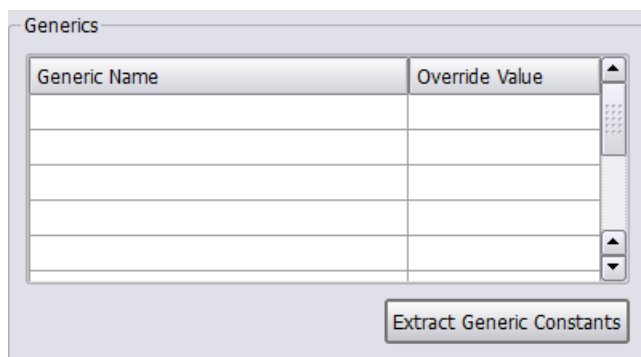
You can also use this option for mixed language designs or when you want to specify a module that is not the actual top-level entity for HDL Analyst displaying and debugging in the schematic views.

2. For user-defined state machine encoding, do the following:
 - Specify the kind of encoding you want to use.
 - Disable the FSM compiler.

When you synthesize the design, the software uses the compiler directives you set here to encode the state machines and does not run the FSM compiler, which would override the compiler directives. Alternatively, you can define state machines with the `syn_encoding` attribute, as described in [Defining State Machines in VHDL, on page 394](#).

3. To extract generics from the source code, do this:
 - Click Extract Generic Constants.
 - To override the default, enter a new value for a generic.

The software uses the new value for the current implementation only. Note that you cannot extract generics if you have a mixed language design.



4. To push tristates across process/block boundaries, check that Push Tristates is enabled. For details, see [Push Tristates Option, on page 381](#) in the *Reference Manual*.
5. Determine the interpretation of the `synthesis_on` and `synthesis_off` directives:
 - To make the compiler interpret `synthesis_on` and `synthesis_off` directives like `translate_on/translate_off`, enable the Synthesis On/Off Implemented as Translate On/Off option.
 - To ignore the `synthesis_on` and `synthesis_off` directives, make sure that this option is not checked. See [translate_off/translate_on](#), on page 315 in the *Reference Manual* for more information.
6. Set other implementation options as needed (see [Setting Logic Synthesis Implementation Options, on page 72](#) for a list of choices). Click OK.

When you synthesize the design, the software compiles and maps the design using the options you set.

Specifying Attributes and Directives

Attributes and directives are specifications that you assign to design objects to control the way your design is analyzed, optimized, and mapped.

Attributes control mapping optimizations and directives control compiler optimizations. Because of this difference, you must specify directives in the source code or the compiler directives file. This table describes the methods that are available to create attribute and directives specifications:

	Attributes	Directives
VHDL	Yes	Yes
Verilog	Yes	Yes
SCOPE Editor	Yes	No
Constraints File	Yes	No

It is better to specify attributes in the SCOPE editor or the constraints file, because you do not have to recompile the design first. For directives, you must compile the design for them to take effect.

If SCOPE/constraints file and the HDL source code are specified for a design, the constraints has the highest priority when there are conflicts. Then, the `cdc` compiler directives take precedence over the HDL source code.

For further details, refer to the following:

- [Specifying Attributes and Directives in VHDL](#), on page 87
- [Specifying Attributes and Directives in Verilog](#), on page 88
- [Specifying Attributes Using the SCOPE Editor](#), on page 89
- [Specifying Attributes in the Constraints File](#), on page 93

Specifying Attributes and Directives in VHDL

You can use other methods to add attributes to objects, as listed in [Specifying Attributes and Directives, on page 86](#). However, you can specify directives only in the source code. There are two ways of defining attributes and directives in VHDL:

- Using the predefined attributes package
- Declaring the attribute each time it is used

For details of VHDL attribute syntax, see [VHDL Attribute and Directive Syntax, on page 420](#) in the *HDL Reference Manual*.

Using the Predefined VHDL Attributes Package

The advantage to using the predefined package is that you avoid redefining the attributes and directives each time you include them in source code. The disadvantage is that your source code is less portable. The attributes package is located in `installDirectory/lib/vhd/synattr.vhd`.

1. To use the predefined attributes package included in the software library, add these lines to the syntax:

```
library synplify;  
use synplify.attributes.all;
```

2. Add the attribute or directive you want after the design unit declaration.

declarations;

attribute *attribute_name* of *objectName* : *objectType* is *value*;

For example:

```
entity simplifiedff is  
  port (q: out bit_vector(7 downto 0);  
        d : in bit_vector(7 downto 0);  
        clk : in bit);  
  attribute syn_noclockbuf of clk : signal is true;
```

For details of the syntax conventions, see [VHDL Attribute and Directive Syntax, on page 420](#) in the *Reference Manual*.

3. Add the source file to the project.

Declaring VHDL Attributes and Directives

If you do not use the attributes package, you must redefine the attributes each time you include them in source code.

1. Every time you use an attribute or directive, define it immediately after the design unit declarations using the following syntax:

```
design_unit_declaration;  
attribute attributeName : dataType;  
attribute attributeName of objectName : objectType is value;
```

For example:

```
entity simplifiedff is  
  port (q: out bit_vector(7 downto 0);  
        d : in bit_vector(7 downto 0);  
        clk : in bit);  
  attribute syn_noclockbuf : boolean;  
  attribute syn_noclockbuf of clk :signal is true;
```

2. Add the source file to the project.

Specifying Attributes and Directives in Verilog

You can use other methods to add attributes to objects, as described in [Specifying Attributes and Directives, on page 86](#). However, you can specify directives only in the source code.

Verilog does not have predefined synthesis attributes and directives, so you must add them as comments. The attribute or directive name is preceded by the keyword `synthesis`. Verilog files are case sensitive, so attributes and directives must be specified exactly as presented in their syntax descriptions. For syntax details, see [Verilog Attribute and Directive Syntax, on page 132](#) in the *Reference Manual*.

1. To add an attribute or directive in Verilog, use Verilog line or block comment (C-style) syntax directly following the design object. Block comments must precede the semicolon, if there is one.

Verilog Block Comment Syntax

```
/* synthesis attributeName = value */  
/* synthesis directoryName = value */
```

Verilog Line Comment Syntax

```
// synthesis attributeName = value  
// synthesis directoryName = value
```


For details of the syntax rules, see [Verilog Attribute and Directive Syntax, on page 132](#) in the *Language Support Reference Manual*. The following are examples:

```
module fifo(out, in) /* synthesis syn_hier = "hard" */;

module b_box(out, in); // synthesis syn_black_box
```

2. To attach multiple attributes or directives to the same object, separate the attributes with white spaces, but do not repeat the synthesis keyword. Do not use commas. For example:

```
case state /* synthesis full_case parallel_case */;
```

3. If multiple registers are defined using a single Verilog reg statement and an attribute is applied to them, then the synthesis software only applies the last declared register in the reg statement. For example:

```
reg [5:0] q, q_a, q_b, q_c, q_d /* synthesis syn_preserve=1 */;
```

The syn_preserve attribute is only applied to q_d. This is the expected behavior for the synthesis tools. To apply this attribute to all registers, you must use a separate Verilog reg statement for each register and apply the attribute.

Specifying Attributes Using the SCOPE Editor

The SCOPE window provides an easy-to-use interface to add any attribute. You cannot use it for adding directives, because they must be added to the source files. (See [Specifying Attributes and Directives in VHDL, on page 87](#) or [Specifying Attributes and Directives in Verilog, on page 88](#)). The following procedure shows how to add an attribute directly in the SCOPE window.

1. Start with a compiled design and open the SCOPE window. To add the attributes to an existing constraint file, open the SCOPE window by clicking on the existing file in the Project view. To add the attributes to a new file, click the SCOPE icon and click Initialize to open the SCOPE window.
2. Click the Attributes tab at the bottom of the SCOPE window.

You can either select the object first (step 3) or the attribute first (step 4).

	Enable	Object Type	Object	Attribute	Value	Value Type	Description	Comment
1	☒	<any>	<Global>	syn_noclockbuf	1	boolean	Use normal input buffer	
2								
3								

Attributes ☐

- To specify the object, do one of the following in the Object column. If you already specified the attribute, the Object column lists only valid object choices for that attribute.
 - Select the type of object in the Object Filter column, and then select an object from the list of choices in the Object column. This is the best way to ensure that you are specifying an object that is appropriate, with the correct syntax.
 - Drag the object to which you want to attach the attribute from the RTL or Technology views to the Object column in the SCOPE window. For some attributes, dragging and dropping may not select the right object. For example, if you want to set `syn_hier` on a module or entity like an and gate, you must set it on the view for that module. The object would have this syntax: `v:moduleName` in Verilog, or `v:library.moduleName` in VHDL, where you can have multiple libraries.
 - Type the name of the object in the Object column. If you do not know the name, use the Find command or the Object Filter column. Make sure to type the appropriate prefix for the object where it is needed. For example, to set an attribute on a view, you must add the `v:` prefix to the module or entity name. For VHDL, you might have to specify the library as well as the module name.
- If you specified the object first, you can now specify the attribute. The list shows only the valid attributes for the type of object you selected. Specify the attribute by holding down the mouse button in the Attribute column and selecting an attribute from the list.

	Enable	Object Type	Object	Attribute	Value	Value Type	Description	Comment
1	☒	<any>	<Global>					
2				syn_allow_retiming				
3				syn_dspstyle				
4				syn_encoding				
5				syn_global_buffers				
6				syn_hier				
				syn_insert_buffer				
				syn_netlist_hierarchy				
				syn_noclockbuf				
				syn_noclockpad				
				syn_ramstyle				

Attributes

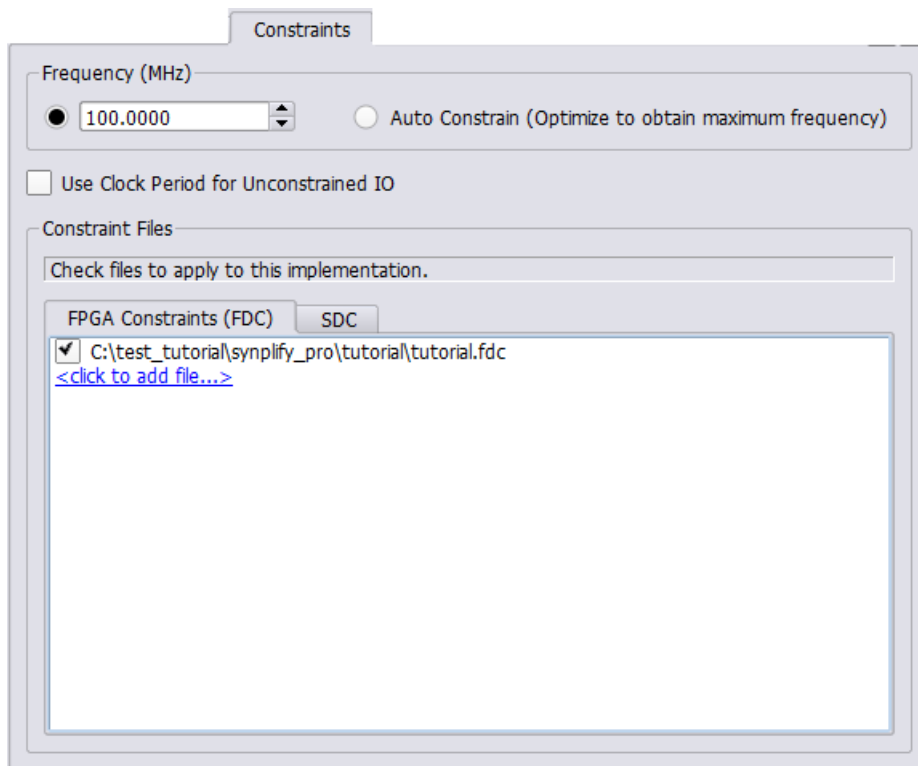
If you selected the object first, the choices available are determined by the selected object and the technology you are using. If you selected the attribute first, the available choices are determined by the technology.

When you select an attribute, the SCOPE window tells you the kind of value you must enter for that attribute and provides a brief description of the attribute. If you selected the attribute first, make sure to go back and specify the object.

5. Fill out the value. Hold down the mouse button in the Value column, and select from the list. You can also type in a value.
6. Save the file.

The software creates a Tcl constraint file composed of `define_attribute` statements for the attributes you specified. See [How Attributes and Directives are Specified, on page 8](#) of the *Reference Manual* for the syntax description.

7. Add it to the project, if it is not already in the project.
 - Choose Project -> Implementation Options.
 - Go to the Constraints panel and check that the file is selected. If you have more than one constraint file, select all those that apply to the implementation.



The software saves the SCOPE information in a Tcl constraint file, using `define_attribute` statements. When you synthesize the design, the software reads the constraint file and applies the attributes.

Specifying Attributes in the Constraints File

When you use the SCOPE window ([Specifying Attributes Using the SCOPE Editor, on page 89](#)), the attributes are automatically written to a constraint file using the Tcl `define_attribute` syntax. This is the preferred method for defining constraints as the syntax is determined for you.

However, the following procedure explains how you can specify attributes directly in the constraint file.

1. Open a file in a text editor.
2. Enter the desired attributes. For example,

```
define_attribute {objectName} attributeName value
```

For commands and syntax, see [Summary of Attributes and Directives, on page 10](#) in the *Attribute Reference Manual*.

3. Save the constraints in a file using the FDC file extension.

The following code excerpt provides an example of attributes defined in the constraint file:

```
# Use a regular buffer instead of a clock buffer for clock "clk_slow".
define_attribute {clk_slow} syn_noclockbuf 1

# Relax timing by not buffering "clk_slow", because it is the slow clock
# Set the maximum fanout to 10000.
define_attribute {clk_slow} syn_maxfan 10000
```

For information about editing constraints, see [Using a Text Editor for Constraint Files \(Legacy\), on page 49](#).

Handling Properties with Attributes or Directives

Any property added to an object (i.e. net or instance) that is preserved or kept during the flow will be annotated in the netlist. Only properties for the `syn_*` attributes/directives are processed by the tool; while all other properties are simply annotated in the netlist when the object is available in the flow.

Examples

Suppose `top_property_handling.v` contains `MyProp=Value`, which is associated with `out` and is annotated in the netlist. If you apply the property on the net `intermediate_net`, then the property is not annotated in the netlist since this net is not preserved/kept in the flow.

Using the `syn_keep` Attribute

The `syn_keep` attribute helps preserve the specified net. If `top_syn_keep.v` applies `syn_keep` on the net `intermediate_net`, then this same net is kept.

Using Properties with the `syn_keep` Attribute

For this example, `top.v` applies the `syn_keep` to the net `intermediate_net` where the `MyProp=Value` is associated with `intermediate_net`. In this case, the property is annotated in the netlist since `syn_keep` instructs the tool to preserve/keep the net in the flow.

Working with Hierarchical Projects

You can use the hierarchical project management (HPM) top-down or bottom-up flows to develop hierarchical designs. You can also use top-down or bottom-up synthesis or a combination of the two to synthesize these designs. For more information about these flows, see [, on page 25](#).

This section describes how to set up and work with hierarchical projects in the flows mentioned above. The following topics provide more detail:

- [Block-Based Subprojects](#), on page 95
- [Creating Hierarchical Subprojects by Exporting Blocks](#), on page 95
- [Creating Nested Subprojects](#), on page 98
- [Working with Multiple Implementations in Hierarchical Projects](#), on page 104
- [Working with Multiple Instances and Parameterized Modules](#), on page 105
- [Configuring Synthesis Runs for Hierarchical Projects](#), on page 108
- [Analyzing Synthesis Results for Hierarchical Projects](#), on page 112

Block-Based Subprojects

Any changes made to a block-based subproject affect all instances of the block. You cannot assign timing budgets or resources to block-based subprojects.

Creating Hierarchical Subprojects by Exporting Blocks

When you use a top-first approach to team design, you must partition the design and specify subprojects. This procedure shows you how to designate a block as a subproject; see [Creating Hierarchical Subprojects by Exporting Blocks, on page 95](#) for information about creating subprojects based on instances.

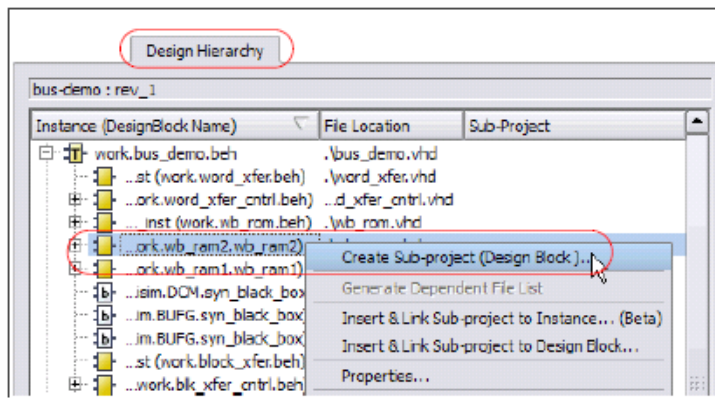
Unlike an instance-based subproject, any changes made to a block-based subproject affect all instances of the block.

1. Open the compiled top-level design.

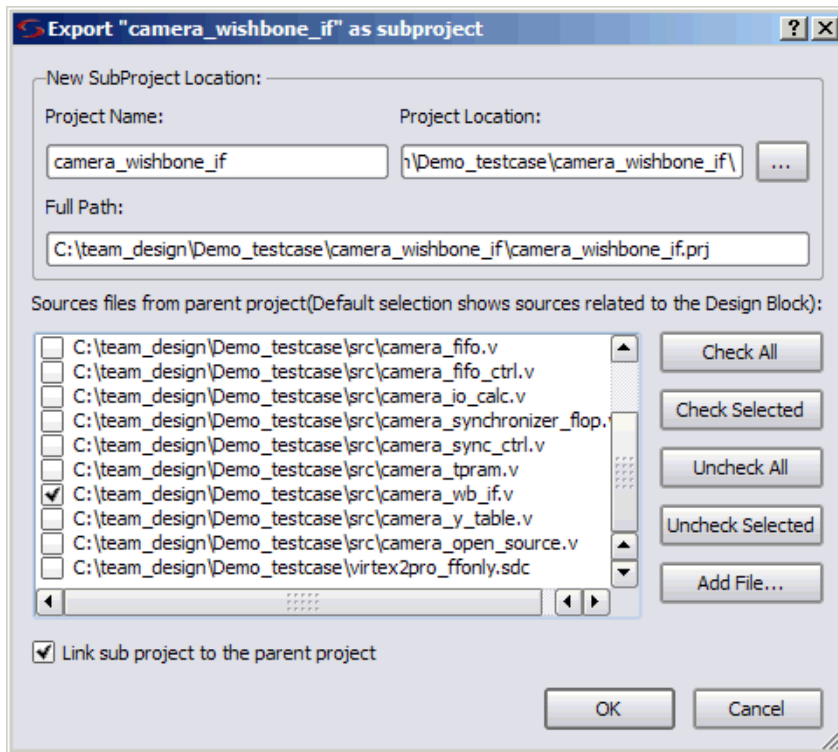
You must compile the design because you cannot export subprojects from an uncompiled design. It does not matter if the top level has black boxes.

2. Export a subproject.

- Go to the Design Hierarchy view, and right-click the block you want to make into a subproject; then select Create Subproject (Design Block) from the popup menu.



The Export "*blockname*" as subproject dialog box opens.



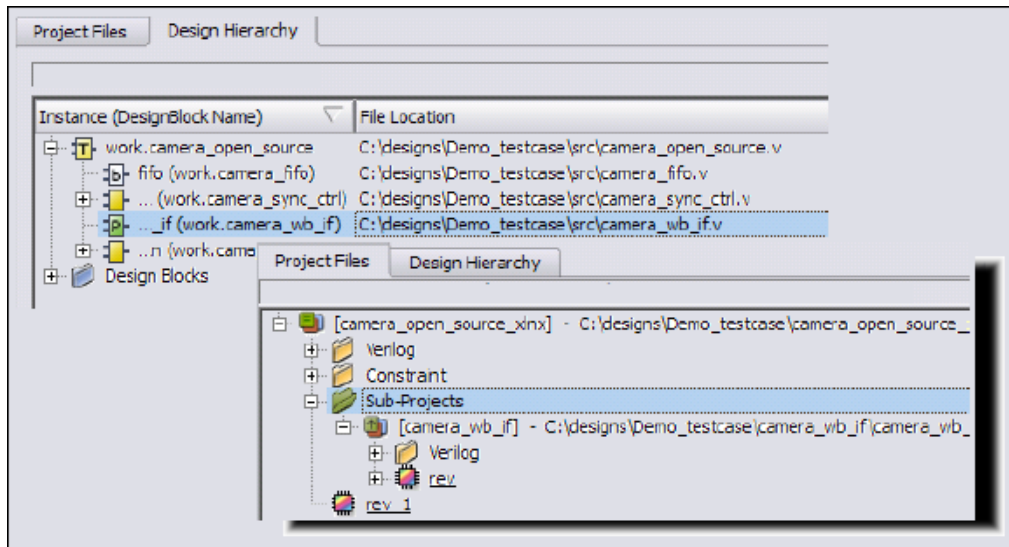
- Specify a name for the project and a location.
- Specify source files for the subproject. Check the selected files in the list. The tool automatically enables needed source files from the top-level design, but the list might not be complete. If a source file is not listed, click the Add File button and add the missing file to the list. The file is automatically selected when it is added with Add File. When the tool creates the subproject, it adds the selected files to it.

You must have at least one file in order to create a subproject. Some files may overlap with the parent project or a sibling project.

- To link updates to the projects, enable the Link sub projects to the parent project option.
- Click OK.

The tool creates a new subproject for the specified block and automatically links it to the top-level project. The block icon in the Design Hierarchy view changes to a green rectangle with a P in it, to indicate that it is now

defined as a subproject. In the Project Files view, you see a Subprojects directory that contains the new block project. The implementation options for the block project match the options of the active implementation of the parent project.



See [Working with Multiple Implementations in Hierarchical Projects](#), on page 104 and [Working with Multiple Instances and Parameterized Modules](#), on page 105 for information about creating additional implementations for the block.

3. Save the design. Until you do so, the hierarchy you created is not saved.
4. Synthesize the design.

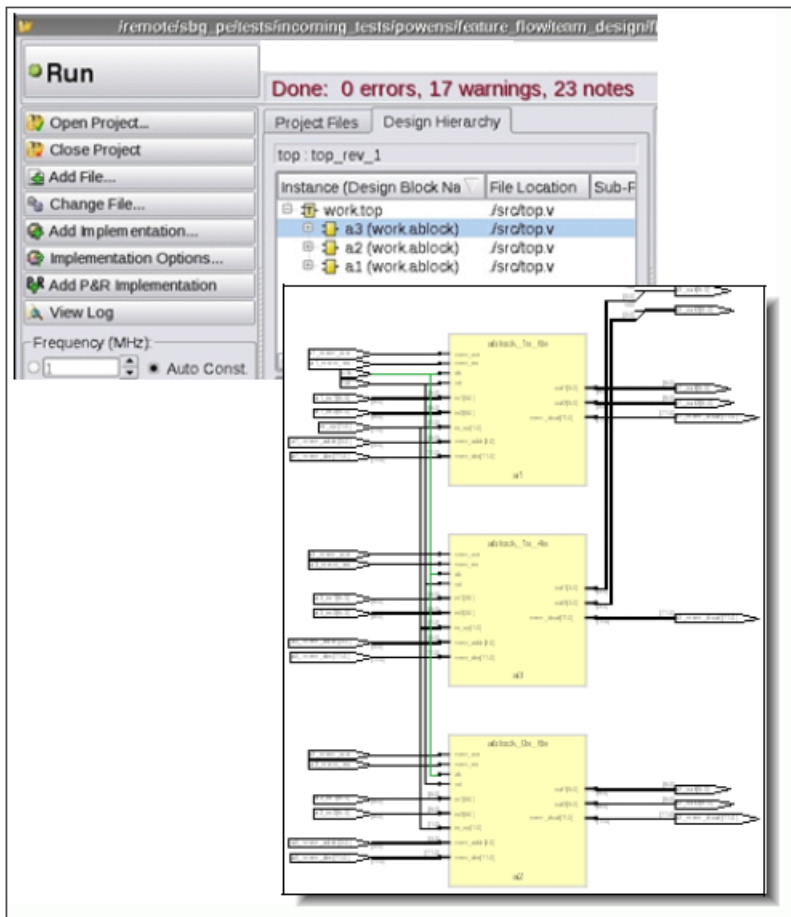
Creating Nested Subprojects

With hierarchical design, you can have subprojects that contain other subprojects. You can use the block and instance exporting techniques to recursively export parts of the design as subprojects.

The following procedure shows how to export nested subprojects, and is based on a design containing three instances of ablock. The ablock module instantiates two bblock instances (b1, b2), bblock instantiates two instances of cblock (c1, c2), and cblock instantiates two instances of dblock (d1, d2).

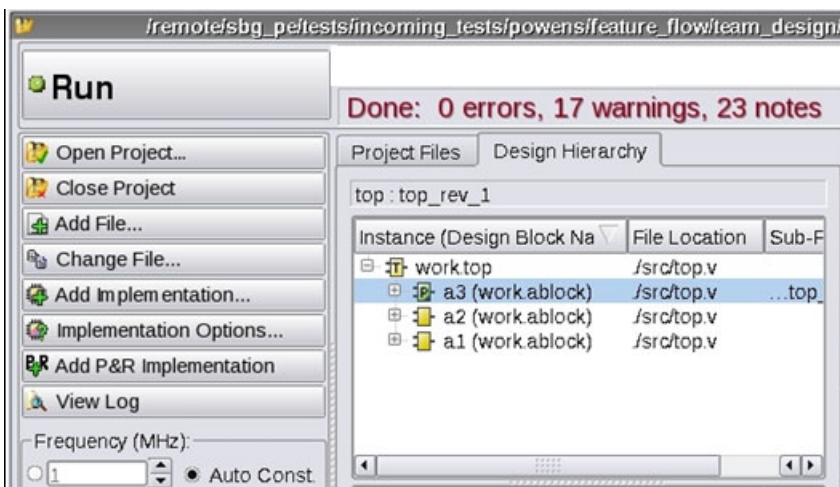
1. Load the top-level project and compile it with Run->Compile Only.

You must compile the design before you can export an instance as a subproject. The Design Hierarchy tab and the RTL view show the a1, a2, and a3 instances at ablock at the top level.

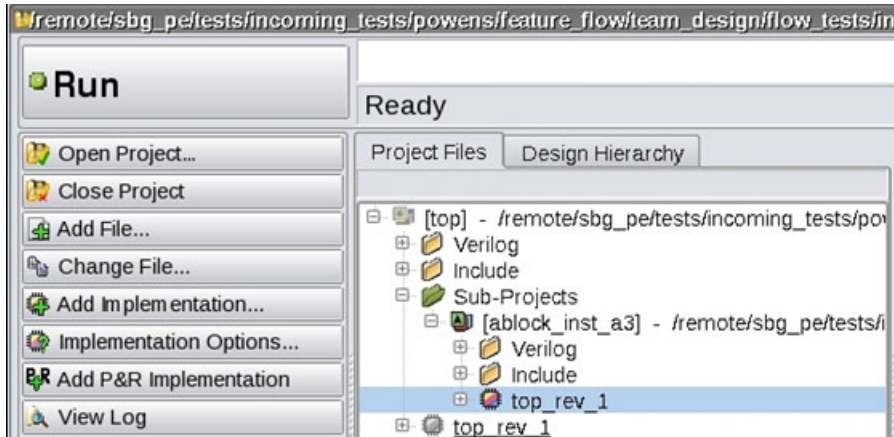


2. Create a subproject.

- In the Design Hierarchy view, right-click an instance (a3) and select the Create Subproject command. For details about using the command to export subprojects, see [Creating Hierarchical Subprojects by Exporting Blocks, on page 95](#).
- A dialog box shows you the files needed for the selected instance. Edit the list as needed to reflect the file list for the instance.
- Click OK. The view now shows a green P icon next to the instance to indicate it is an exported subproject.



The Project Files view shows the newly-created subproject. It has its own implementation, separate from the top-level implementation.



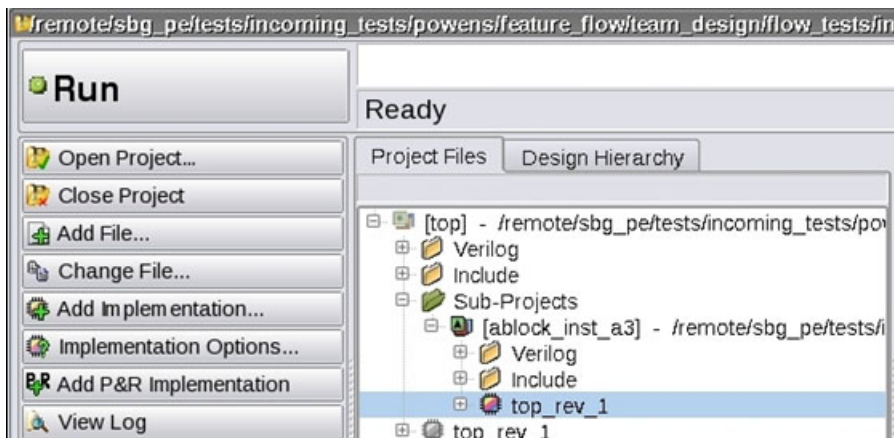
3. Save the design.

Until you save the design, the subproject you just created is not written to disk.

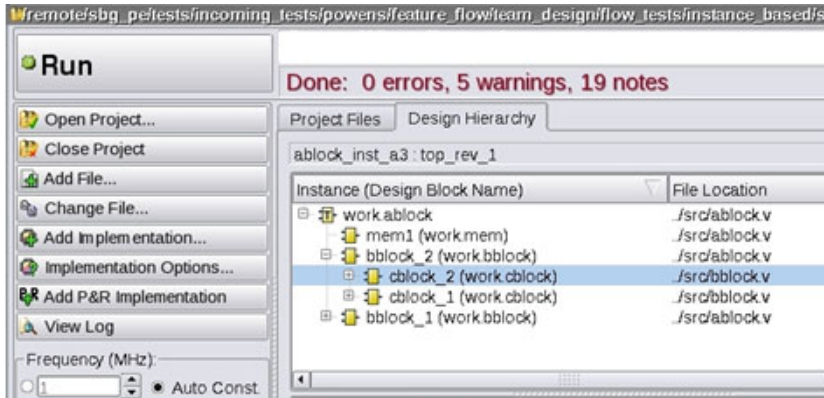
4. Create a subproject within a subproject.

- Compile the subproject under which you want to create another subproject (a3), by selecting the subproject implementation and selecting Run->Compile Only.

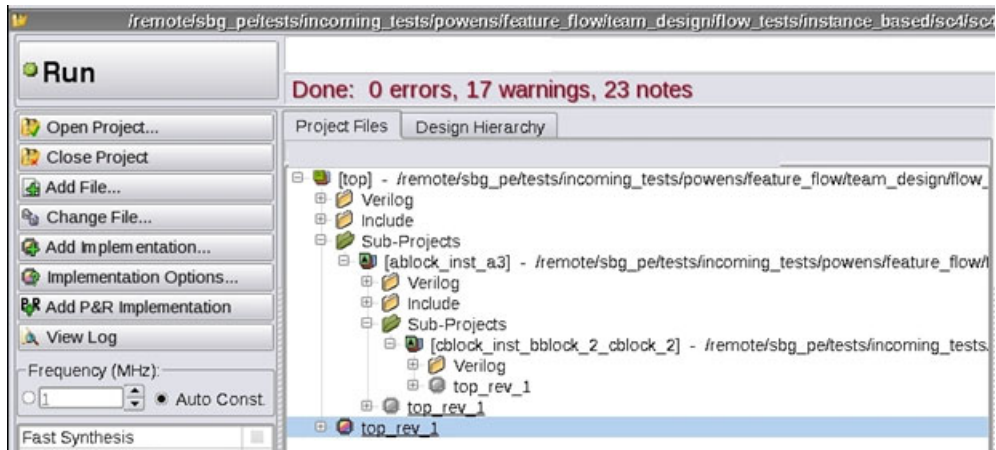
You must compile again at this level, because you cannot export a subproject from an uncompiled design, and at this point the a3 subproject is not compiled.



- In the Design Hierarchy view, right-click an instance and select the Create Subproject command to export it. You can select any instance in the hierarchy. The following figure shows the current subproject expanded, the next level skipped (bblock), and a cblock instance selected from two levels below:



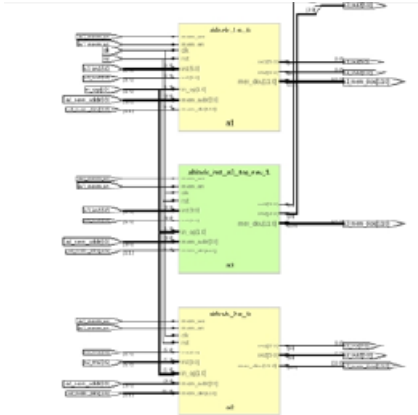
- Check that the list of dependent files for the instance is accurate in the window that opens, and click OK when you are done. The Design Hierarchy and Project Files views reflect the creation of a new cblock subproject under the parent ablock subproject.



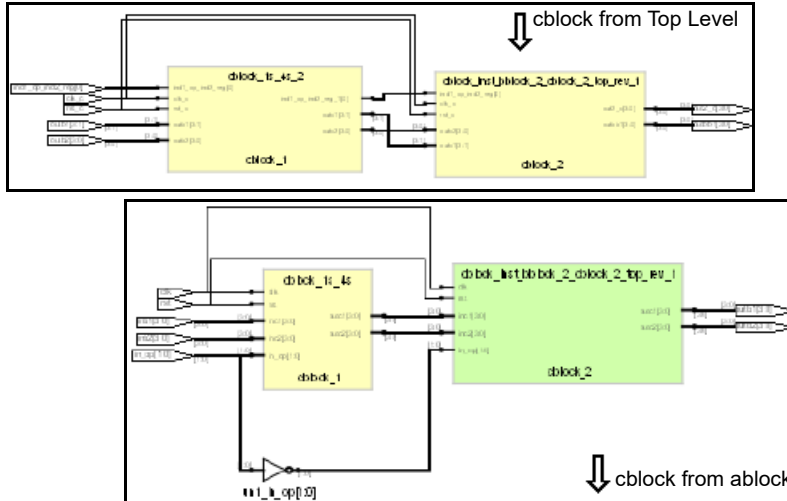
5. Save the design.

6. Compile and map the entire design by selecting the top-level implementation and clicking Run.

The top-level RTL shows exported subprojects in green, to indicate that they were exported. The following figure shows the a3 instance that was exported from the top level.



The RTL view indicates exported subprojects relative to the current level. If you push into the a3 instance from the top level, and then descend into the bblock instance, the exported cblock instance is not in green, because the reference point is the top level. However, if you select the ablock subproject and then generate an RTL view, the cblock instance is in green when you push down to it. This is because the cblock instance was exported relative to the ablock subproject.



Working with Multiple Implementations in Hierarchical Projects

The following procedure show you how to work with multiple implementations in hierarchical projects. For example, you might want to create multiple implementations with different run configurations if you are experimenting with different implementation options.

1. Open the top-level hierarchical project and compile it (F7).
This updates the hierarchical project views.
2. To add an implementation to the top-level project or a block-level project, do the following in the Project Files view:
 - Right-click the top level or the block and select Add Implementation.
 - Set the new implementation options you want.
3. To add subproject implementations to the top-level project, do one of the following:
 - In the Project Files view, right-click a top-level implementation and select Add Subproject Implementations.
 - Select a top-level implementation in the Project Files view. Select the Project-> Add Subproject Implementation command.

The tool automatically adds a new implementation to each defined subproject, and updates the top-level implementation to use the new block implementations. The new block implementations are based on the selected top-level implementation.

4. To specify which implementation to run, do the following:

- Select Project->Edit Run Configuration.
- In the dialog box, set Implementation to the block implementation you want to synthesize. To prevent synthesis from being run set this option to <off>. If there are red cells in the table, synchronize options across implementations as needed with the Sync buttons. Click OK.
- To run all implementations, select the Run->Run all Implementations instead of clicking the Run button to synthesize.

By default the tool runs the active implementation.

Working with Multiple Instances and Parameterized Modules

A module can have various parameters, such as SIZE or WIDTH. When multiple instances of a module have different parameter values, they are called parameterized modules. The following procedure shows you how to work with multiple instantiations of parameterized modules.

1. Create your design and instantiate the parameterized modules.

In the design example used to illustrate this procedure, cblock is instantiated four times. The instantiations have different values defined for DWIDTH and USE_AND:

```
`include "meminfo.h"
parameter USE_AND = 1;
//parameter USE_AND = 0;
//parameter DWIDTH = 2;
parameter DWIDTH = 6;
```

The following table shows the parameter values for the four cblock instances in the design. It also shows that there are three unique sets of parameters between the instances; cblock_1 and cblock_3 share the same parameter set.

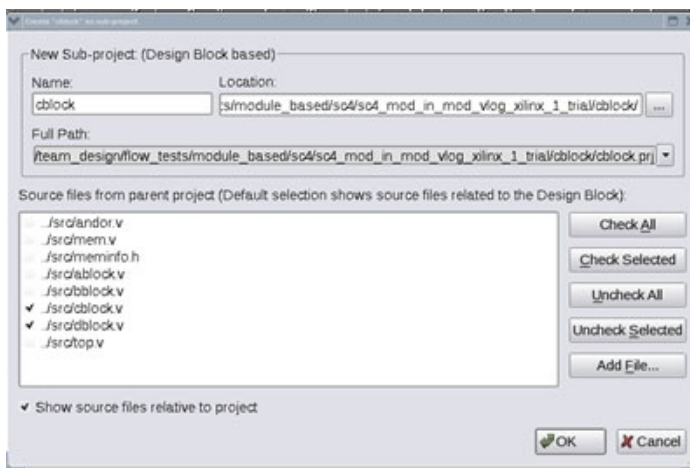
cblock Instances	USE_AND	DWIDTH	Parameter Set
cblock_1	0	6	1
cblock_2	1	6	2
cblock_3	0	6	1
dblock_4	1	2	3

2. Export one instance of the parameterized module.

- Compile the design.
- Open the Design Hierarchy tab, and select one instance of the parameterized module. You can select any instance of the module in the design to export.
- Export the subproject by right-clicking the module and selecting the Create Subproject command.

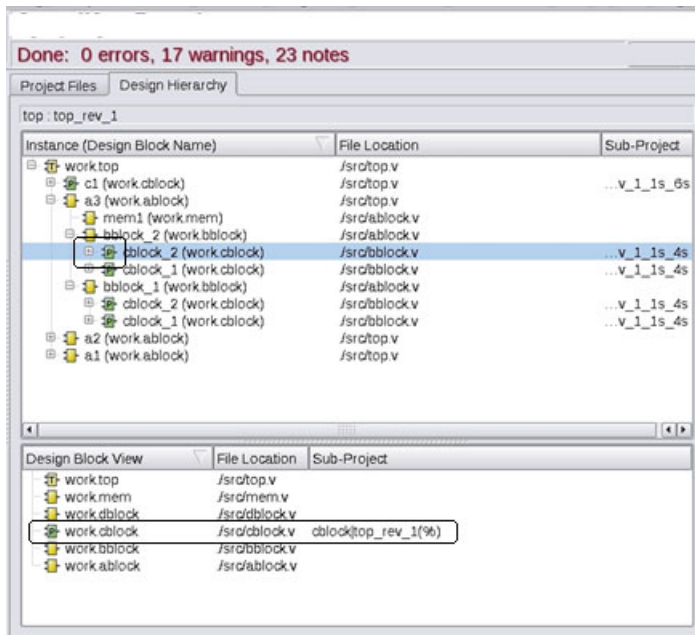
A block-based export creates a single subproject that affects all instances of the module.

- If required, add any additional files needed for the subproject to the automatically generated list that displays, and click OK.

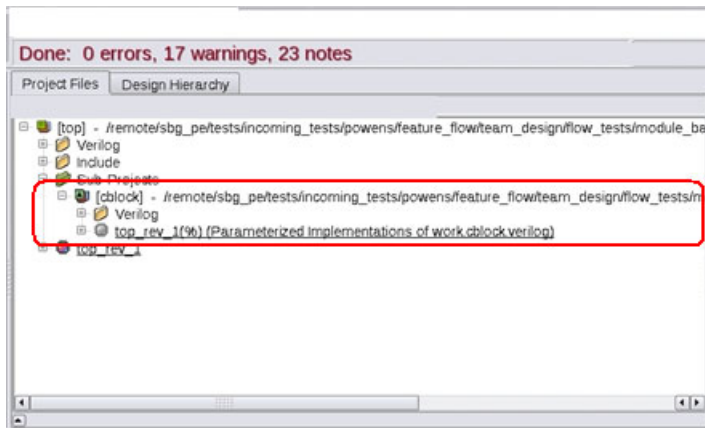


After you set the files for the subproject, the module is marked with a green icon in the Design Hierarchy view. Module-based subprojects are

denoted by `rev_#(%)`, as shown in the following figure, where `cblock` was exported as a module.



- Check the Project Files view. It shows that a subproject was created for `cblock`, and that it is a parameterized implementation:



3. Click the Save All icon () in the top bar to save all the exported subproject and project file information.
4. Set options and constraints for synthesis.
5. Run the project.

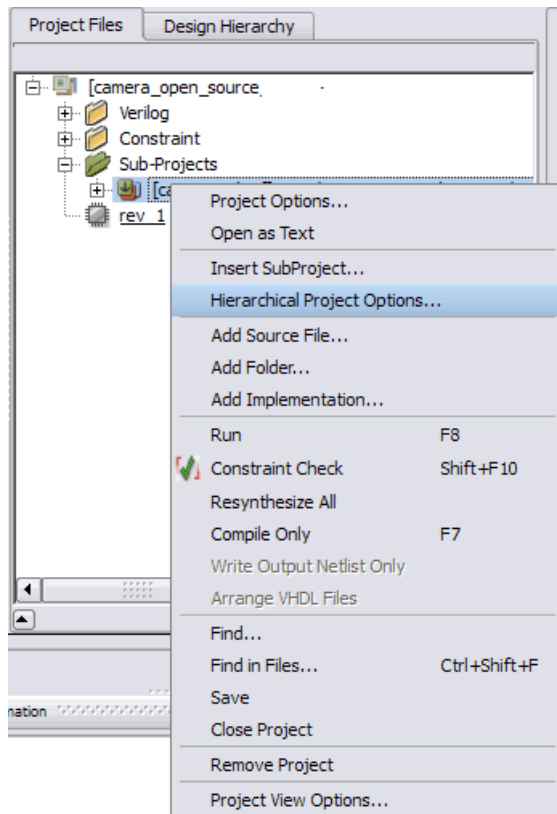
All exported subprojects are synthesized from the bottom up by default. If you want to allow the subproject to be optimized, you can change the default, as described in [Configuring Synthesis Runs for Hierarchical Projects, on page 108](#).

6. If you make changes to parameters in a design, do the following:
 - Right-click the top-level project implementation and select Subproject Parameter Sync.
 - Rerun synthesis.

Configuring Synthesis Runs for Hierarchical Projects

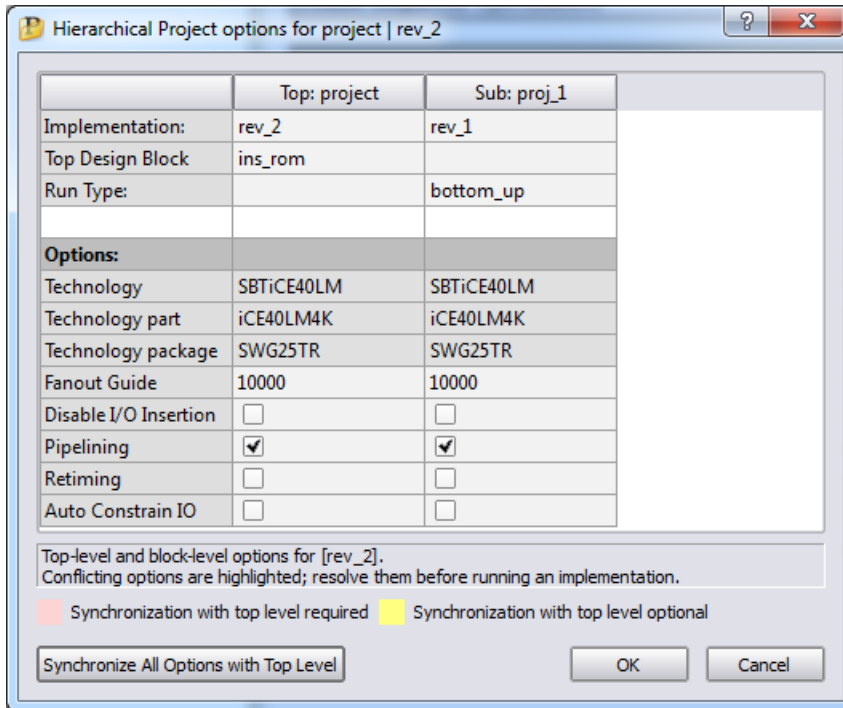
Before you synthesize the hierarchical project, set options for the synthesis run. This procedure shows you how to use the Hierarchical Project Options command and set synthesis options for the top level design or for the blocks. You can set different run configurations for different implementations.

1. Select the Hierarchical Project Options command.
 - Select the top level or the block in the Project Files view.
 - Either right-click and select Hierarchical Project Options from the popup menu or select Project->Hierarchical Project Options.



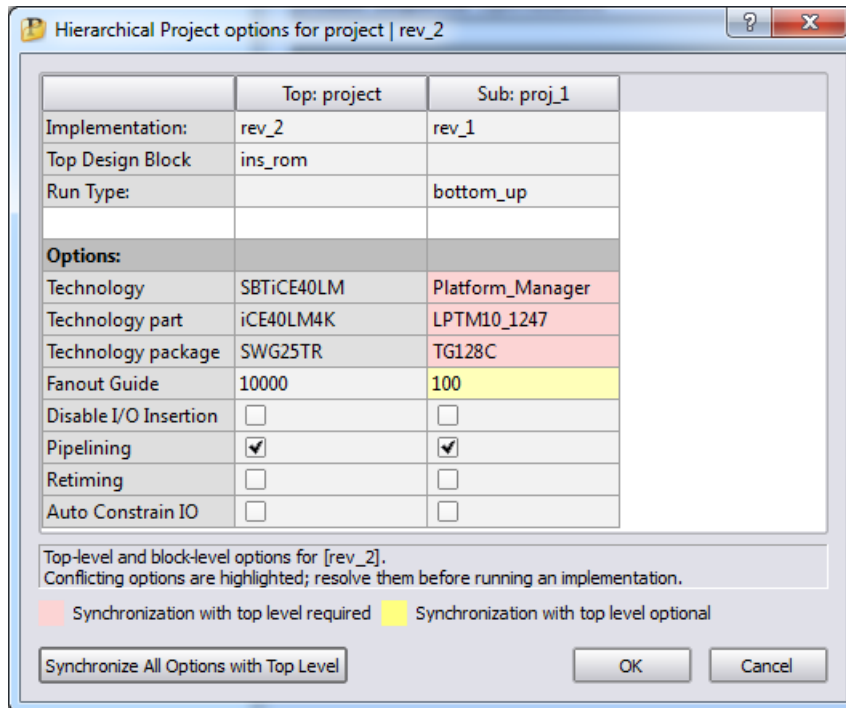
2. Set options in the dialog box that opens:

- In the block column, set Implementations to the implementation you want to run. If you do not want to synthesize a block, set it to <off>. When you set a block to <off>, that subproject is treated as a black box.



- Set Run Type for the block to reflect how you want it synthesized: top_down or bottom_up.
 - Set any other implementation options you want for the synthesis run.
3. If necessary, synchronize the top-level and block settings. If you see a red cell in the table, it means that option must be synchronized before synthesis.

The following figure shows that pink highlighting in the device option field. This indicates that you must synchronize the subproject with the top level setting. The Disable I/O Insertion option is yellow and synchronization with the top level is optional.



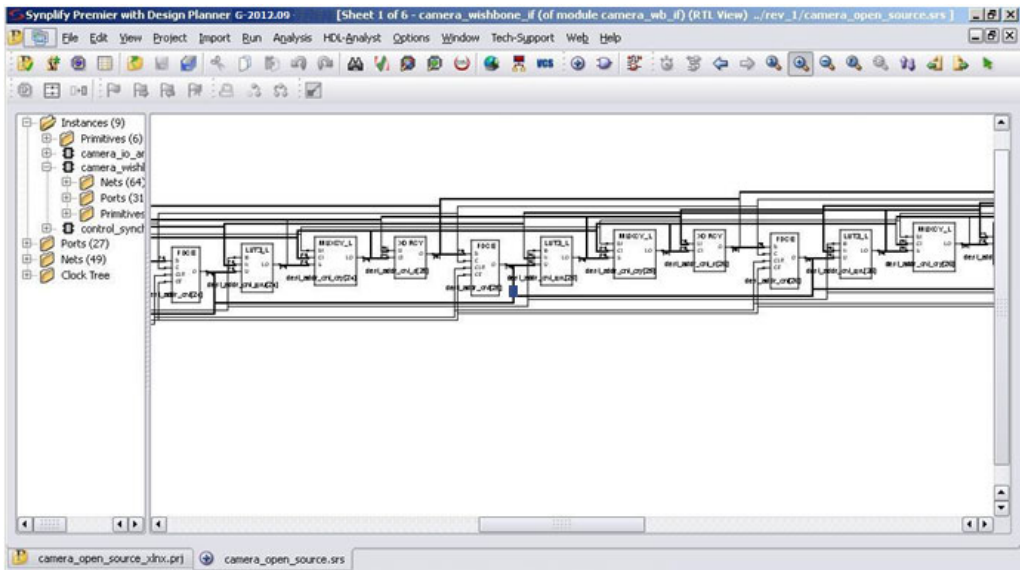
- Click the Synchronize All Options with Top Level button at the bottom of the box to match the required block device settings to the settings at the top level. If the required device options match, this button is grayed out and unavailable.
4. Click OK.
 5. Synthesize the design.

Analyzing Synthesis Results for Hierarchical Projects

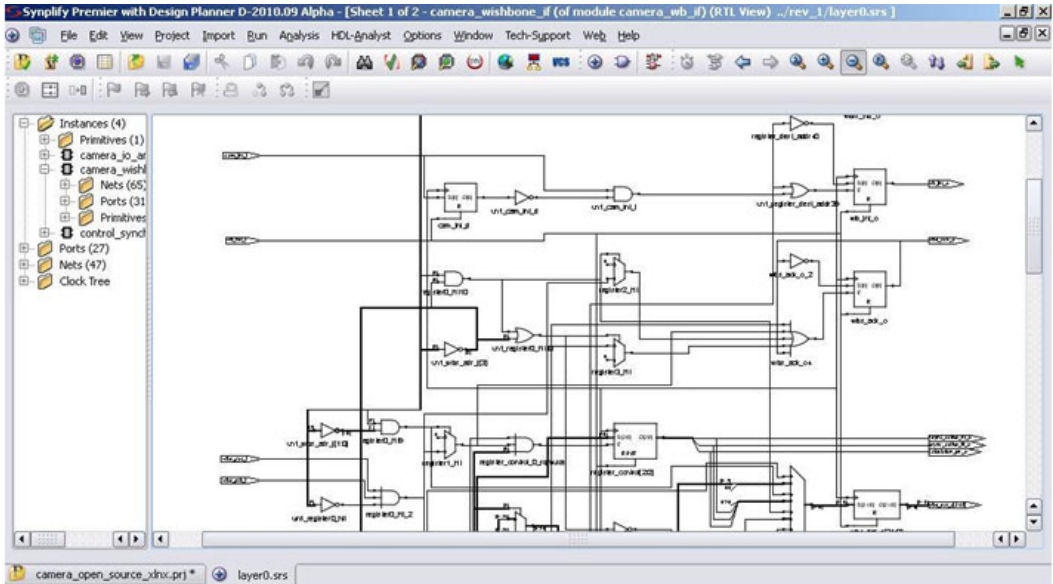
After synthesizing a hierarchical project, you can analyze the results as described below.

1. To view the synthesis results, check the RTL view.

The following figure of the top-level RTL view shows the linked netlists from subprojects.



This is an example of an RTL view of the subprojects:



2. Check the log file for hierarchical reports for each subproject.
3. For resource usage, check the corresponding sections in the top-level and subproject log files.

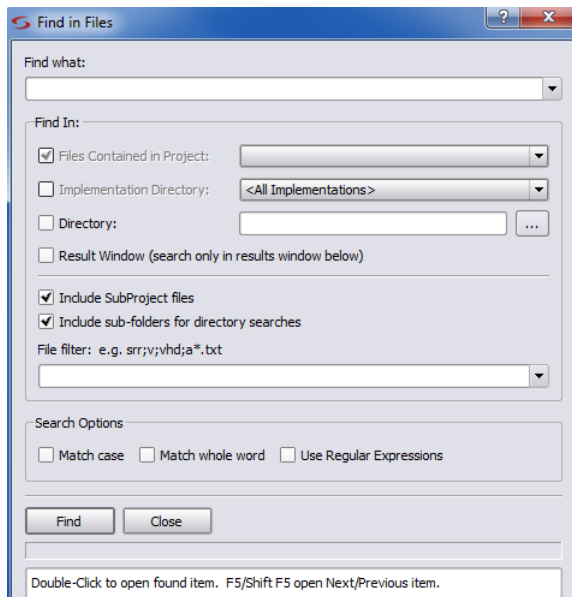
Searching Files

A find-in-files feature is available to perform string searches within a specified set of files. Advantages to using this feature include:

- Ability to restrict the set of files to be searched to a project or implementation.
- Ability to cross probe the search results.

The find-in-files feature uses a dialog box to specify the search pattern, the criteria for selecting the files to be searched, and any search options such as match case or whole word. The files that meet the criteria are searched for the pattern, and a list of the files containing the search pattern are displayed at the bottom of the dialog box.

To use the find-in-files feature, open the Find in Files dialog box by selecting Edit->Find in Files and enter the search pattern in the Find what field at the top of the dialog box.



Identifying the Files to Search

The Find In section at the top of the dialog box identifies the files to be searched:

- **Project Files** – Searches the files included in the selected project (use the drop-down menu to select the project). By default, the files in the active project are searched. The files can reside anywhere on the disk; any project 'include files are also searched.
- **Implementation Directory** – Searches all files in the specified implementation directory (use the drop-down menu to select the implementation). By default, the files in the active implementation are searched. You can search all implementations by selecting <All Implementations> from the drop-down menu. If Include sub-folders for directory searches is also selected, all files in the implementation directory hierarchy are searched.
- **Subproject** – Searches all files of a subproject included in the top-level project file, when Include SubProject files is selected.
- **Directory** – Searches all files in the specified directory (use the browser button to select the directory). If Include sub-folders for directory searches is also selected, all files in the directory hierarchy are searched.

All of the above selection methods can be applied concurrently when searching for a specified pattern.

The Result Window selection is used after any of the above selection methods to search the resulting list of files for a subsequent sub-pattern.

Filtering the Files to Search

A file filter allows the file set to be searched to be further restricted based on the matching of patterns entered into the File filter field.

- A pattern without a wildcard or a "." (period) is interpreted as a filename extension. For example, `fdc` restricts the search to only constraint files.
- Multiple patterns can be specified using a semicolon delimiter. For example, `v;vhd` restricts the files searched to only Verilog and VHDL files.
- Wildcard characters can be used in the pattern to match file names. For example, `a*.vhd` restricts the files searched to VHDL files that begin with an "a" character.

- Leaving the File filter field empty searches all files that meet the Find In criteria.
- The Match Case, Whole Word, and Regular Expressions search options can be used to further restrict searches.

Initiating the Search

After entering the search criteria, click the Find button to initiate the search. All matches found are listed in the results area at the bottom of the dialog box; the status line just below the Find button reports the number of matches found in the indicated number of files and the total number of files searched.

While the find operation is running, the status line is continually updated with how many matches are found in how many files and how many files are being searched.

Search Results

The search results are displayed in the results window at the bottom of the dialog box. For each match found, the entire line of the file is displayed in the following format:

fullpath_to_file(lineNumber): matching_line_text

For example, the entry

```
C:\Designs\leon\dcache.vhd(487): wdata := r.wb.data1;
```

indicates that the search pattern (data1) was found on line 487 of the dcache.vhd file.

To open the target file at the specified line, double-click the line in the results window.

Archiving Files and Projects

Use the archive utility to archive, extract (unarchive), or copy design projects. Archived files are in a proprietary format and saved to a file name using the `sar` extension. The archive utility is available through the Project menu in the GUI or using the project command in the Tcl window.

This document provides a description of how to use the utility.

- [Archive a Project](#)
- [Un-Archive a Project](#)
- [Copy a Project](#)
- [Support for Hierarchical Include Paths](#)

Archive a Project

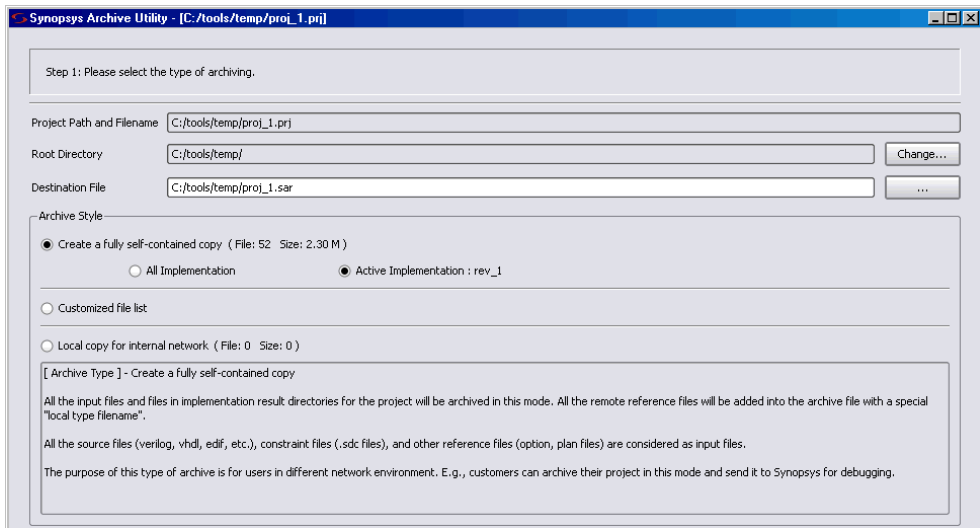
Use the archive utility to store the files for a design project into a single archive file in a proprietary format (`sar`). You can archive an entire project or selected files from a project. If you want to create a copy of a project without archiving the files, see [Copy a Project, on page 124](#).

Here are the steps to create an archive:

1. In the Project view, select Project->Archive Project to bring up the wizard.

The Tcl command equivalent is `project -archive`. For a complete description of the project Tcl command options for archiving, see [project, on page 83](#) of the *Reference Manual*.

The archive utility automatically runs a syntax check on the active project (Run->Syntax Check command) to ensure that a complete list of project files is generated. If you have Verilog 'include files in your project, the utility includes the complete list of Verilog files. It also checks the syntax automatically for each implementation in the project to ensure that the file list is complete for each implementation as well. The wizard displays the name of the project to archive, the top-level directory where the project file is located (root directory), and other information.

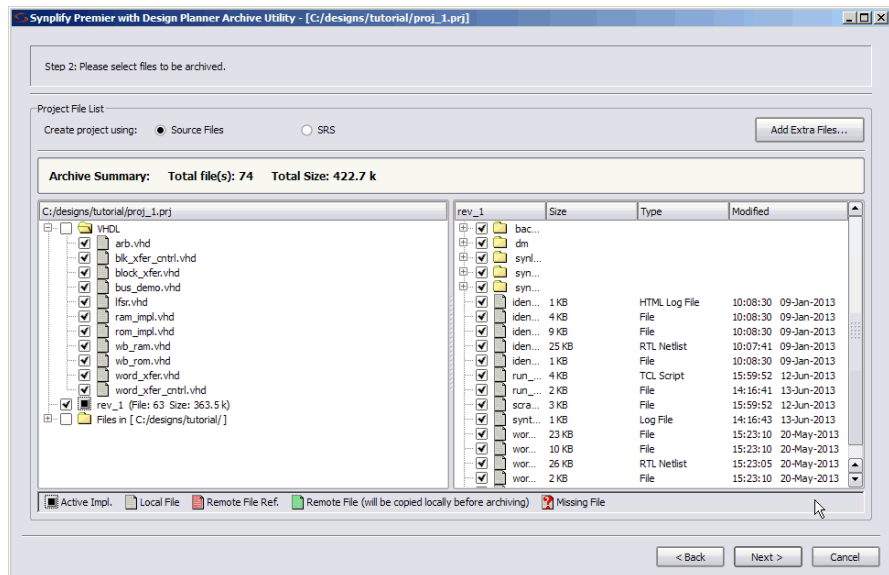
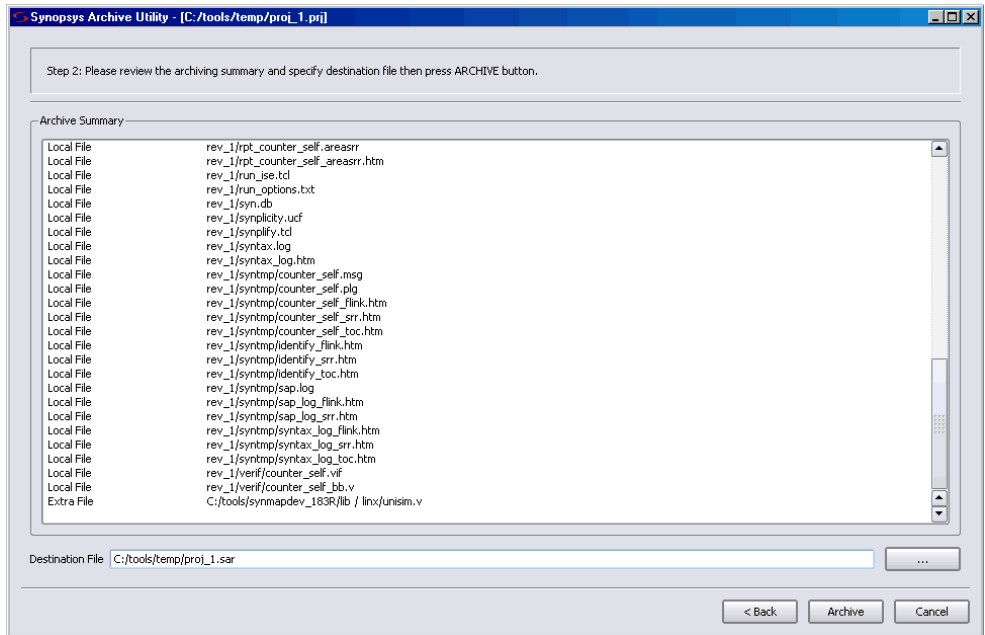


2. Do the following on the first page of the wizard:

- Fill in Destination File with a location for the archive file.
- Set Archive Style. You can archive all the project files with all the implementations or selectively archive files and implementations
- To archive only the active implementation, enable Active Implementation.
- To selectively archive files, enable Customized file list, click Next, and use the check boxes to include files in or exclude files from the archive. Use the Add Extra Files button on the this page to include additional files in the project.

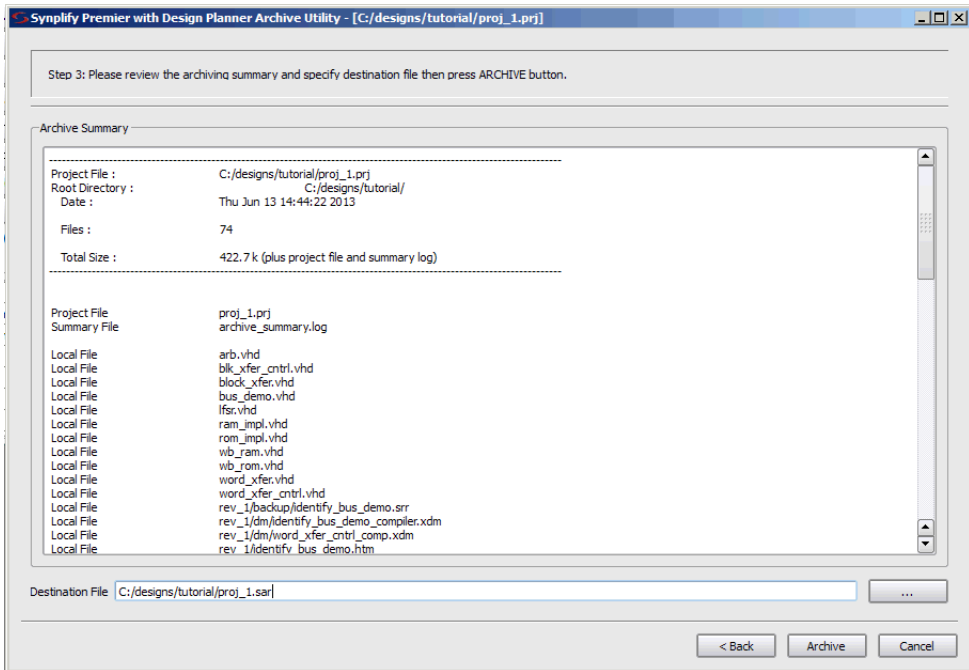
3. Click Next.

If you did not select Customized file list, the tool summary displays all the files in the archive and shows the full uncompressed file size as shown in step 5 (the actual size is smaller after the archiving operation as there is no duplication of files). When you select Customized file list, the following interim menu is displayed to allow you to exclude specific file from the archive.



4. Click next to advance to the next screen (step 3).

5. Verify that the current archive contains the files that you want, then click **Archive** which creates the project archive `sar` file. If the list of files is incorrect, click **Back** and include/exclude any desired files.



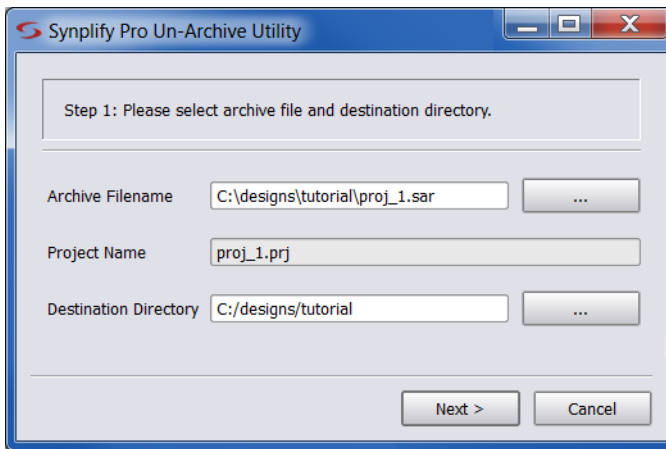
6. Click **Archive** if you are finished. The synthesis tool reports the archive success and the path location of the archive file.

Un-Archive a Project

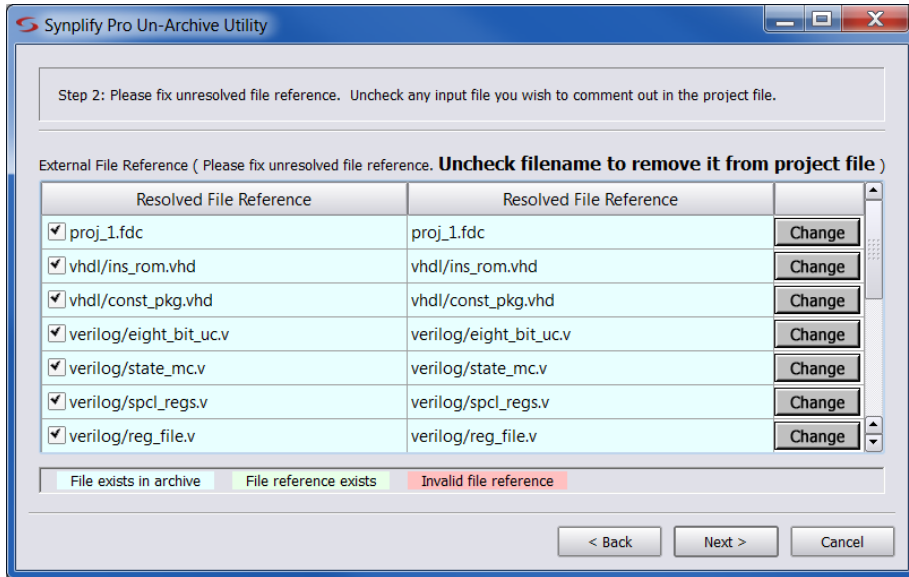
Uses this procedure to extract design project files from an archive file (sar).

1. In the Project view, select Project->Un-Archive Project to display the wizard

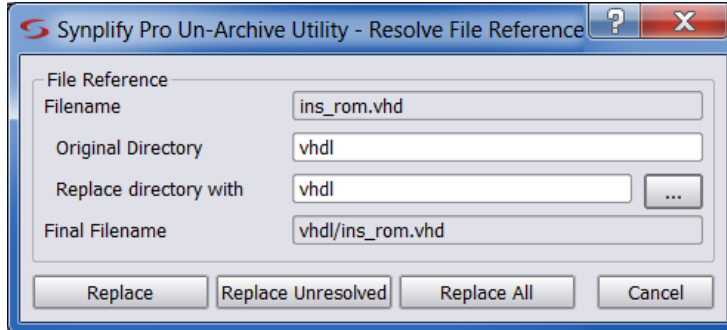
The Tcl command equivalent is project [-unarchive](#). For a complete description of the project Tcl command options for archiving, see [project](#), on [page 83](#) of the *Reference Manual*.



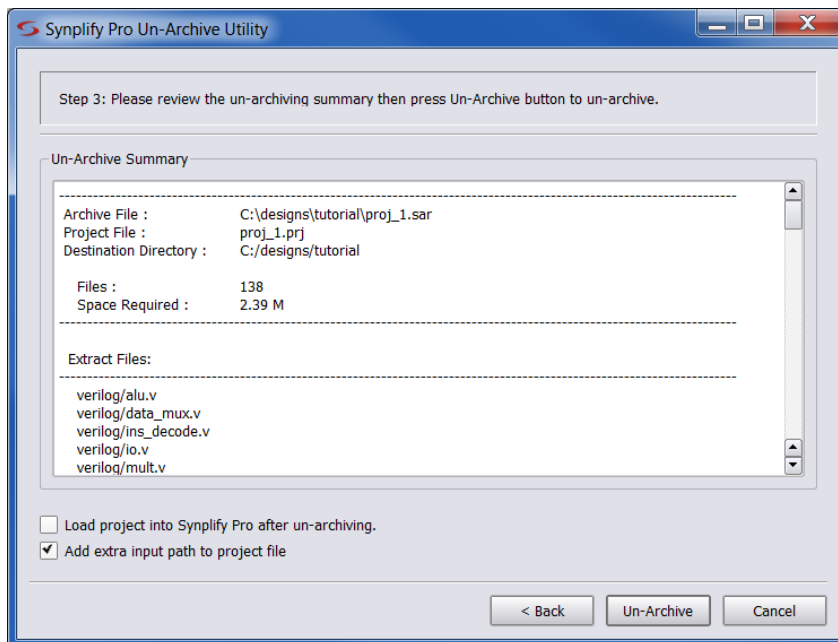
2. In the wizard, enter the following:
 - Name of the sar file containing the project files.
 - Name of project to extract (un-archive). This field is automatically extracted from the sar file and cannot be changed.
 - Pathname of directory in which to write the project files (destination).
 - Click Next.



3. Make sure all the files that you want to extract are checked and references to these files are resolved.
 - If there are files in the list that you do not want to include when the project is un-archived, uncheck the box next to the file. The unchecked files will be commented out in the project file (`proj`) when project files are extracted.
 - If you need to resolve a file in the project before un-archiving, click the Resolve button and fill out the dialog box.
 - If you want to replace a file in the project, click the Change button and fill out the dialog box. Put the replacement files in the directory you specify in Replace directory. You can replace a single file, any unresolved files, or all the files. You can also undo the replace operation.



4. Click Next and verify that the project files you want are displayed in the Un-Archive Summary.



5. If you want to load this project in the UI after files have been extracted, enable the Load project into Synplify Pro after un-archiving option.
6. When the Add extra input path to project file option is enabled, the archive utility finds all include files and copies them into a directory called extra_input. This directory is added to the unarchived project file.

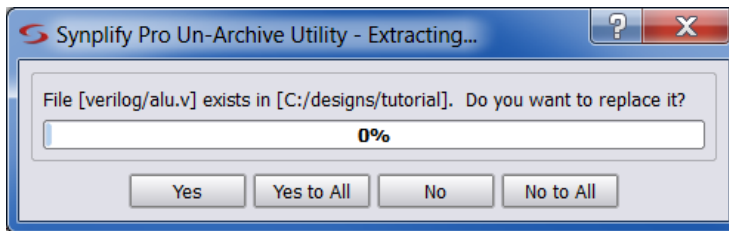
The Tcl command equivalent is `set_option -include_path "/extra_input"`.

If the archive files contain relative or absolute include paths, the `_SEARCHFILENAMEONLY_` directive can have the compiler remove the relative/absolute paths from the 'include and search only for the file names. To use the `_SEARCHFILENAMEONLY_` directive, all include files must have unique names. For details, see [_SEARCHFILENAMEONLY_, on page 388](#).

7. Click Un-Archive.

A message dialog box is displayed while the files are being extracted.

8. If the destination directory already contains project files with the same name as the files you are extracting, you are prompted so that the existing files can be overwritten by the extracted files.



Copy a Project

Use this utility to create an unarchived copy of a design project. You can copy an entire project or just selected files from the project. However, if you want to create an archive of the project, where the entire project is stored as a single file, see [Archive a Project, on page 117](#).

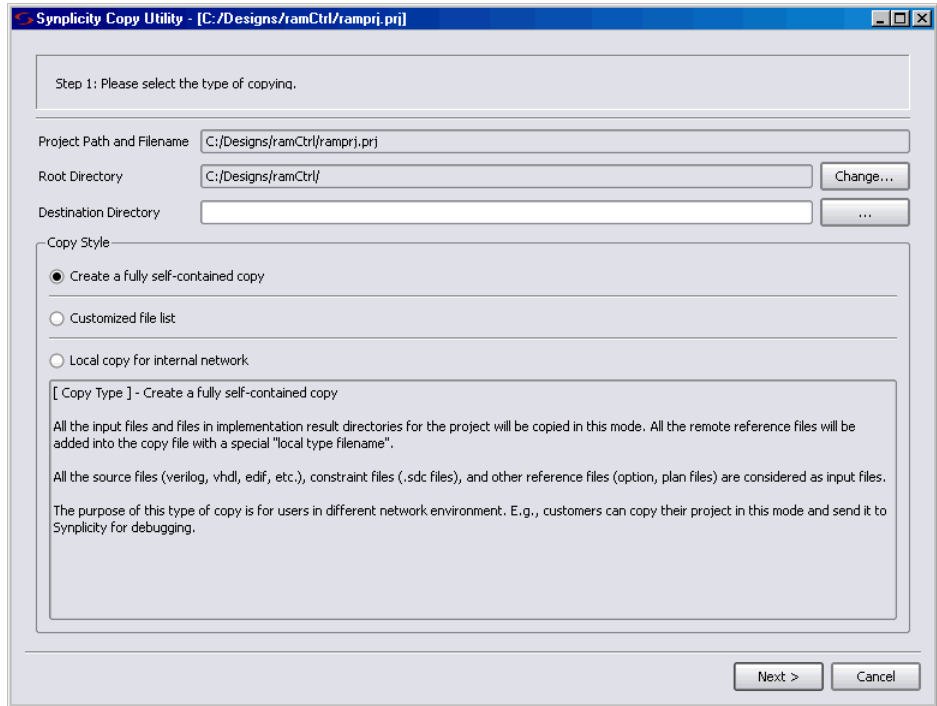
Here are the steps to create a copy of a design project:

1. From the Project view, select Project->Copy Project.

The Tcl command equivalent is `project -copy`. For a complete description of the project Tcl command options for archiving, see [project, on page 83](#) of the *Reference Manual*.

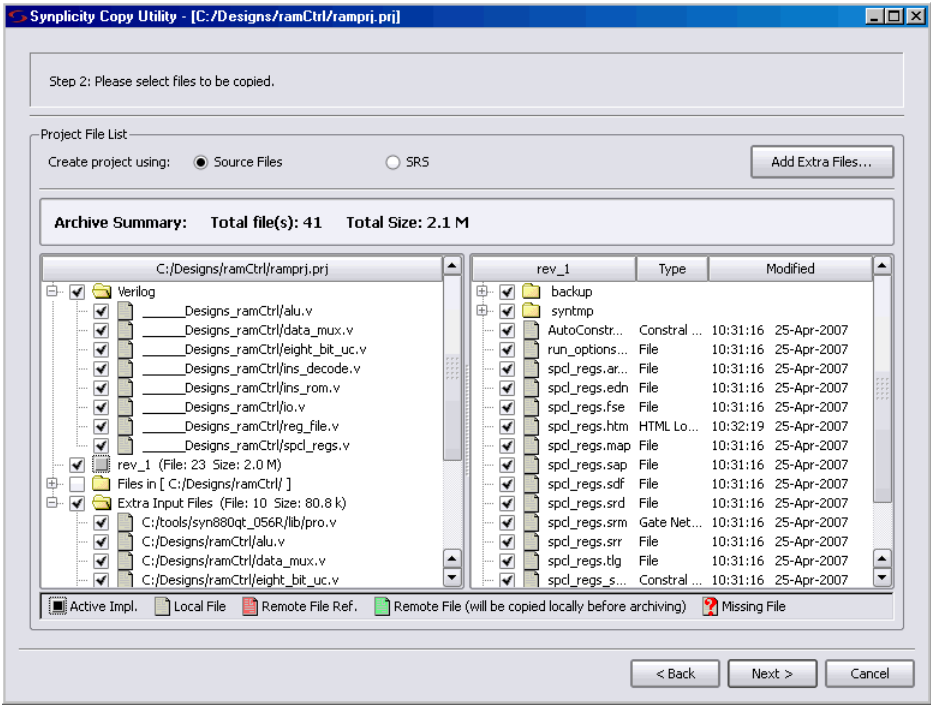
This command automatically runs a syntax check on the active project (Run->Syntax Check command) to ensure that a complete list of project files is generated. If you have Verilog include files in your project, they are included. The utility runs this check for each implementation in the

project to ensure that the file list is complete for each implementation and then displays the wizard, which contains the name of the project and other information.

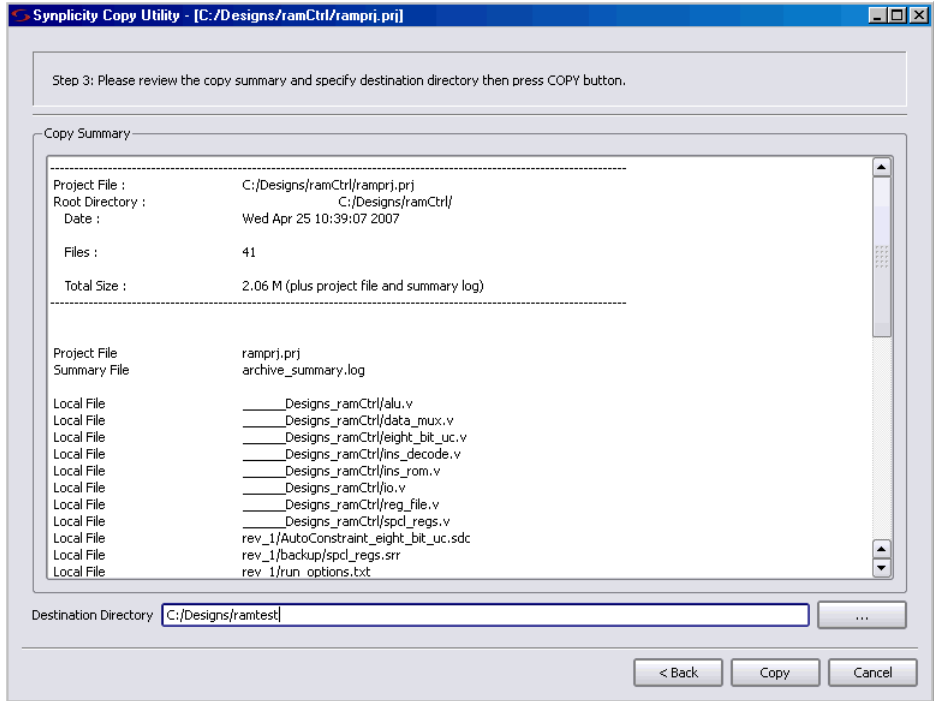


2. Do the following in the wizard:

- Specify the destination directory where you want to copy the files.
- Select the files to copy. You can choose to copy all the project files; one or more individual files, input files only, or customize the list to be copied.
- To specify a custom list of files, enable Customized file list. Use the check boxes to include or exclude files from the copy. Enable SRS if you want to copy all srs files (RTL schematics). You cannot enable the Source Files option if you select this. Use the Add Extra Files button to include additional files in the project.



– Click Next.



3. Do the following:

- Verify the copy information.
- Enter a destination directory. If the directory does not exist it will be created.
- Click Copy.

This creates the project copy.

Support for Hierarchical Include Paths

The archive utility can support various forms of include path hierarchies to locate files for a project. For example:

- The include path can be relative to the location of the source file.

```
block_a/
block_a.v -> `include "block_a.h"
```

- The include path can be a relative path outside of the project.

```
block_b.v -> `include "../..../block_b.h"
```

The archive utility can determine the absolute path for the file from the relative path as shown below:

```
remote/sbg_pe/tests/feature_flow/include/block_b.h
```

After unarchiving the project, you can see the directory structure for the equivalent absolute path relative to the project.

```
"../remote/sbg_pe/tests/feature_flow/include/block_b.h"
```

- The file location can be specified by `include_path` in the project file.

```
block_c/  
block_c.v -> `include "block_c.h"
```

Where this file is located in the directory `/include1/`.

- The include path can be an absolute path outside of the project.

```
block_d/  
block_d.v -> `include "/slowfs/sbg/tests/include2/block_d.h"
```

When you archive the project, the absolute path becomes a relative path. After unarchiving the project, you can see the directory structure for the relative path to the project.

```
"../slowfs/sbg/tests/include2/block_d.h"
```

- The file location can be specified by `include_path` in the project file.

```
top_block/  
top_block.v -> `include "top_block.h"
```

Where the `top_block.v` file is located in the directory `/include2/`.

- Any additional search paths specified in the project file are copied and included as relative paths to the project.

After you archive and unarchive the project, the relative paths in the original project become absolute paths in the new unarchived project. In the project file, the `set_option -include_path` preserves the original search order for the files.

Using the `__SEARCHFILENAMEONLY__` Compiler Directive

Whenever you have a `SAR` file that contains relative or absolute include paths for the files in the project, you can also use the `__SEARCHFILENAMEONLY__` directive to have the compiler remove the relative/absolute paths from the 'include and search only for the file names. Otherwise, you may have problems using the archive utility. For details, see [__SEARCHFILENAMEONLY__, on page 388](#).

CHAPTER 5

Specifying Constraints

This chapter describes how to specify constraints for your design. It covers the following:

- [Using the SCOPE Editor](#), on page 132
- [Specifying SCOPE Constraints](#), on page 139
- [Specifying Timing Exceptions](#), on page 151
- [Finding Objects with Tcl find and expand](#), on page 157
- [Using Collections](#), on page 167
- [Converting SDC to FDC](#), on page 177
- [Using the SCOPE Editor \(Legacy\)](#), on page 178
- [Generating Constraint Files for Technologies Supported in Radiant Software](#), on page 198

The following chapters discuss related information:

- [Chapter 4, Constraints](#) (*Reference Manual*) for an overview of constraints
- [Chapter 5, SCOPE Constraints Editor](#), (*Reference Manual*) for a description of the SCOPE editor

Using the SCOPE Editor

The SCOPE (Synthesis Constraints OPTimization Environment[®]) presents a spreadsheet-like editor with a number of panels for entering and managing timing constraints and synthesis attributes. The SCOPE GUI is good for editing most constraints, but there are some constraints (like black box constraints) which can only be entered as directives in the source files. The SCOPE GUI also includes an advanced text editor that can help you edit constraints easily.

These constraints are saved to the FPGA Design Constraint (FDC) file. The FDC file contains *Synopsys SDC Standard* timing constraints (for example, `create_clock`, `set_input_delay`, and `set_false_path`), along with the non-timing constraints (design constraints) (for example, `define_attribute`, `define_scope_collection`, and `define_io_standard`). When working with these constraints, use the following processes:

- For existing designs, run the `sdc2fdc` script to translate legacy SDC constraints and create a constraint file that contains Synopsys SDC standard timing constraints and design constraints. For details about this script, see [Converting SDC to FDC, on page 177](#).
- For new designs, use the SCOPE editor. See [Creating Constraints in the SCOPE Editor, on page 132](#) for more information.
- For new designs, use the `create_fdc_template` Tcl command. See [Creating Constraints With the FDC Template Command, on page 137](#) for details.

Creating Constraints in the SCOPE Editor

The following procedure shows you how to use the SCOPE editor to create constraints for the FDC constraint file.

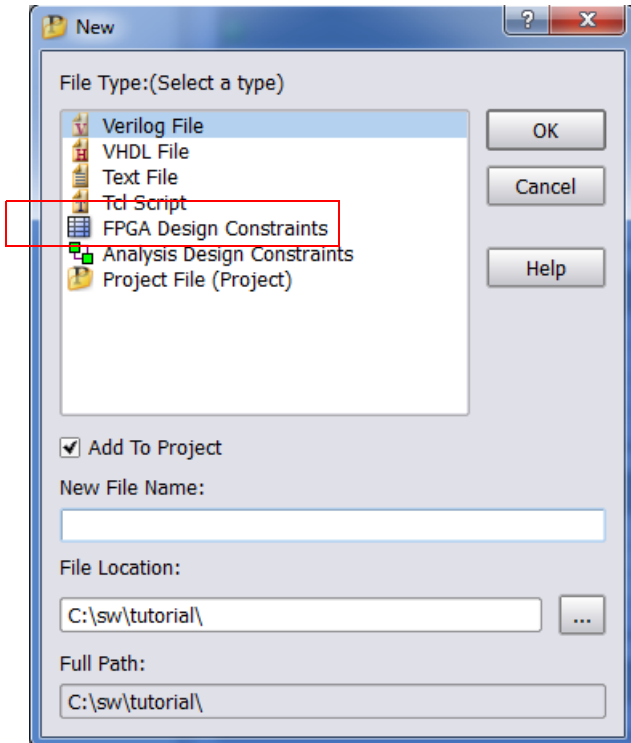
1. To create a new constraint file, follow these steps:
 - Compile the design (F7).
 - Open the SCOPE window by:

Clicking the SCOPE icon in the toolbar ().

This brings up the New Constraint File dialog box.

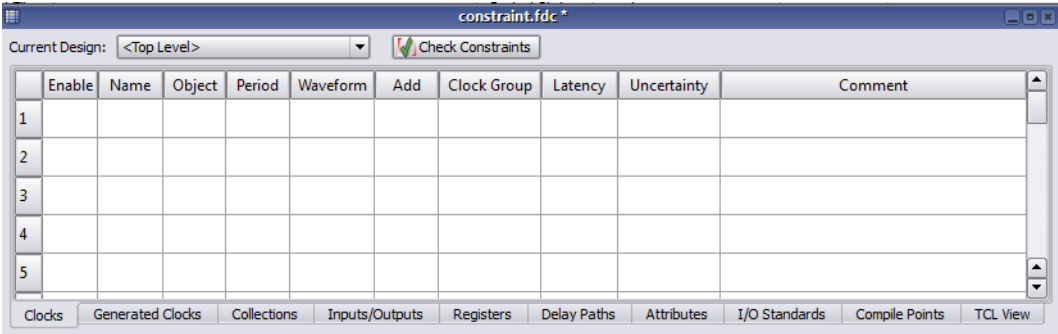
OR

Pressing Ctrl-n or selecting File -> New. This brings up the New dialog box.

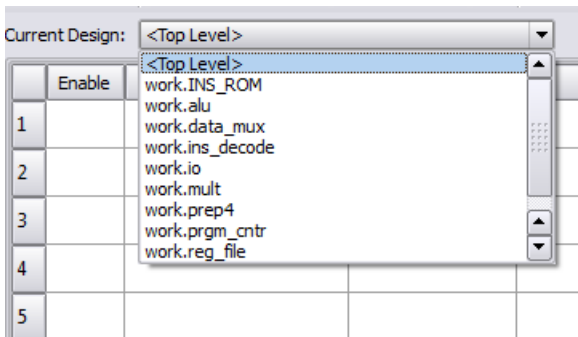


2. To open an existing file, do one of the following:
 - Double-click the file from the Project view.
 - Press Ctrl-o or select File->Open. In the dialog box, set the kind of file you want to open to Constraint Files (SCOPE) (fdc), and double-click to select the file from the list.

An empty SCOPE spreadsheet window opens. The tabs along the bottom of the SCOPE window list the different kinds of constraints you can add. For each kind of constraint, the columns contain specific data.



3. Select if you want to apply the constraint to the top-level or for modules from the Current Design option drop-down menu located at the top of the SCOPE editor.



4. You can enter or edit the following types of constraints:
- Timing constraints – on the Clocks, Generated Clocks, Inputs/Outputs, Registers, or Delay Paths tab.
 - Design constraints – on the Collections, Attributes, I/O Standards, or Compile Points tab.

For details about these constraints, see [Specifying SCOPE Constraints, on page 139](#).

For information about ways to enter constraints within the SCOPE editor, see [Guidelines for Entering and Editing Constraints, on page 148](#).

5. The free form constraint editor is located in the TCL View tab, which is the last tab in SCOPE. The text editor has a help window on the right-

hand side. For more information about this text editor, see [Using the TCL View of SCOPE GUI, on page 146](#).

6. Click on the Check Constraints button to run the constraint checker. The output provides information on how the constraints are interpreted by the tool.

All constraint information is saved in the same FPGA Design Constraint file (FDC) with clearly marked beginning and ending for each section. Do not manually modify these pre-defined SCOPE sections.

The following example shows the contents of an FDC file.

```
#####
# FDC constraints translated from Synplify Legacy Timing & Design Constraints
#####

set_rtl_ff_names {#}
###==== BEGIN Header

# Synopsys, Inc. constraint file
# D:\bugs\timing_88\clk_prior\scratch\top.fdc
# Written on Wed Jun 20 10:50:15 2012
# by Synplify Premier with Design Planner, G-2012.09 FDC Constraint Editor

# Custom constraint commands may be added outside of the SCOPE tab sections bounded with BEGIN/END.
# These sections are generated from SCOPE spreadsheet tabs.

###==== END Header

###==== BEGIN Clocks - (Populated from tab in SCOPE, do not edit)
create_clock -name {clk_a} {p:clk_a} -period 10 -waveform {0 5.0}
create_clock -name {clk_b} {p:clk_b} -period 6.667 -waveform {0 3.3335}
set_clock_groups -derive -name default_clkgroup_0 -asynchronous -group {get_clocks {clk_a}}
set_clock_groups -derive -name default_clkgroup_1 -asynchronous -group {get_clocks {clk_b}}
###==== END Clocks

###==== BEGIN "Generated Clocks" - (Populated from tab in SCOPE, do not edit)
###==== END "Generated Clocks"

###==== BEGIN Collections - (Populated from tab in SCOPE, do not edit)
define_scope_collection all_inputs_fdc {find -port * -filter @direction==input}
define_scope_collection all_outputs_fdc {find -port * -filter @direction==output}
define_scope_collection all_clocks_fdc {find -hier -clock *}
define_scope_collection all_registers_fdc {find -hier -seq *}
define_scope_collection all_grp {define_collection [find -inst {i:FirstStbcPhase}] [find -inst {i:Norm}]
define_scope_collection fdc_cmd_0 {find -seq {*y*.q[*]}}
define_scope_collection fdc_cmd_1 {find {n:foo}}
define_scope_collection fdc_cmd_2 {expand -hier -seq -from $fdc_cmd_1}
###==== END Collections

###==== BEGIN Inputs/Outputs - (Populated from tab in SCOPE, do not edit)
set_input_delay -clock {c:clk_a} -clock_fall -add_delay 0.000 $all_inputs_fdc
set_output_delay -clock {c:clk_a} -add_delay 0.000 $all_outputs_fdc
set_input_delay -clock {c:clk_a} -add_delay 2.00 {p:a[7:0]}
set_input_delay -clock {c:clk_a} -add_delay 0 {p:rst}
###==== END Inputs/Outputs

###==== BEGIN "Delay Paths" - (Populated from tab in SCOPE, do not edit)
set_multicycle_path 3 -end -from $fdc_cmd_0
set_false_path -comment {false foo[0] free{idh}} -from {b:ena}
set_false_path -from $fdc_cmd_2 -to {i:abc.def.g_reg} -through {n:bar}
set_false_path -from {#boing} -to {c:dcmlclk0fx_derived_clock[2]} -through {n:fudge}
set_false_path -from {c:dcmlclk0fx_derived_clock} -to {i:abc.def.g_reg[0] i:abc} -through {n:fudge}
###==== END "Delay Paths"
```


Creating Constraints With the FDC Template Command

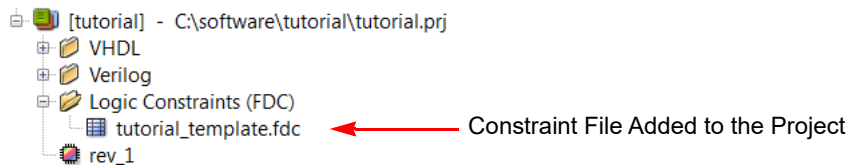
Use the Tcl command `create_fdc_template` to create an initial constraint file (fdc) for your specific design. This command lets you specify port clocks, I/O delays, and initial `set_clock_groups` for the clocks for which text headers are generated that can help guide you when creating this constraint.

The following procedure shows you how to create constraints in the FDC constraints file with the `create_fdc_template` command:

1. Create a project for your design.
2. Compile the design.
3. At the command line, for example, you can specify the following:

```
create_fdc_template -period 10 -out_delay 1.5
```

The command automatically updates your project to reflect the new constraint file(s). Do **Ctrl+s** to save the new settings.



4. If you open the SCOPE editor, you can check that the clock period and output delay values were added to the constraint file as shown in the following figure.

	Enable	Name	Object	Period	Waveform	Add	Clock Group	Latency	Uncertainty	Comment		
1	☒	clock	[get_ports {p:clock}]	10	0 5.0	☐	<default>					
2												
3		Enable	Delay Type	Port	Rise	Fall	Max	Min	Clock	Clock Fall	Add Delay	Value
4	1	☒	input	p:porta[7:0]	☐	☐	☐	☐	c:clock	☐	☒	0
5	2	☒	input	p:portb[7:0]	☐	☐	☐	☐	c:clock	☐	☒	0
6	3	☒	input	p:portc[7:0]	☐	☐	☐	☐	c:clock	☐	☒	0
7	4	☒	input	p:resetsn	☐	☐	☐	☐	c:clock	☐	☒	0
Clocks	5	☒	output	p:porta[7:0]	☐	☐	☐	☐	c:clock	☐	☒	1.5
	6	☒	output	p:portb[7:0]	☐	☐	☐	☐	c:clock	☐	☒	1.5
	7	☒	output	p:portc[7:0]	☐	☐	☐	☐	c:clock	☐	☒	1.5
Inputs/Outputs												

5. Each port clock includes a `set_clock_groups` header with details displayed in the TCL View, which can help you determine whether clocks have been optimized away or if there are any derived clocks.

However, if there is only one clock port and no derived clocks, no explicit clock groups are created since they are not needed, as shown below.

```
#####
### Individual "set_clock_groups" commands for all "clock" derived clocks
### appear at the end of this file. Enabling a given command will make the
### given clock asynchronous to all other clocks. If a given clock (below) does not
### appear in the final Performance Summary (in the *.srr file after synthesis),
### the clock may have been optimized away due to Gated/Generated Clock Conversion.
### See the "CLOCK OPTIMIZATION REPORT" in the *.srr file.
### Below is a list of any clocks derived from "clock":
###   clock DERIVED CLOCKS:
#####
```

For details about the command syntax, see [create_fdc_template](#), on [page 41](#).

6. You can continue using the SCOPE editor to create other constraints.
7. Save the constraint file.

Specifying SCOPE Constraints

Timing constraints define the performance goals for a design. The FPGA synthesis tool supports a subset of the Synopsys SDC Standard timing constraints (for example, `create_clock`, `set_input_delay`, and `set_false_path`). For additional support, see [Synopsys Standard Timing Constraints, on page 140](#).

Design constraints let you add attributes, define collections and specify constraints for them, and select specific I/O standard pad types for your design.

You can define both timing and design constraints in the SCOPE editor. For the different types of constraints, see the following topics:

- [Entering and Editing SCOPE Constraints](#)
- [Setting Clock and Path Constraints](#)
- [Defining Input and Output Constraints](#)
- [Specifying Standard I/O Pad Types](#)

To set constraints for timing exceptions like false paths and multicycle paths, see [Specifying Timing Exceptions, on page 151](#).

For information about collections, see [Using Collections, on page 167](#).

Entering and Editing SCOPE Constraints

This section contains a description of the timing and design constraints you can enter in the SCOPE GUI that are saved to an FDC file. The SCOPE timing constraint panels include:

SCOPE Panel	See...	Tcl Commands
Clocks	Clocks	create_clock set_clock_groups set_clock_latency set_clock_uncertainty
Generated Clocks	Generated Clocks	create_generated_clock
Collections	Collections	define_scope_collection

SCOPE Panel	See...	Tcl Commands
Inputs/Outputs	Inputs/Outputs	set_input_delay set_output_delay
Registers	Registers	set_reg_input_delay set_reg_input_delay
Delay Paths	Delay Paths	set_false_path set_max_delay set_multicycle_path
Attributes	Attributes	define_attribute define_global_attribute
I/O Standards	I/O Standards	define_io_standard
Compile Points	Compile Points	define_compile_point define_current_design
TCL View	TCL View	--

Synopsys Standard Timing Constraints

The FPGA synthesis tools support Synopsys standard timing constraints for a subset of the clock definition (Clocks and Generated Clocks), I/O delay (Inputs/Outputs), and timing exception constraints (Delay Paths).

Setting Clock and Path Constraints

The following table summarizes how to set different clock and path constraints from the SCOPE window.

To define...	Pane	Do this to set the constraint...
Clocks	Clock	Select the clock object (Clock). Specify a clock name (Clock Alias), if required. Type a period (Period). Change the rise and fall edge times for the clock waveforms of the clock in nanoseconds, if needed. Change the default clock group, if needed. Check the Enabled box. See Defining Clocks , on page 185 for information about clock attributes.
Generated Clocks	Generated Clocks	Select the generated clock object. Specify the master clock source (a clock source pin in the design). Specify whether to use invert for the generated clock signal. Specify whether to use: edges, divide_by, or multiply_by. Check the Enabled box.
Input/output delays	Inputs/Outputs	See Defining Input and Output Constraints (Legacy) , on page 195 for information about setting I/O constraints.
Maximum path delay	Delay Paths	Select the Delay Type path of Max Delay. Select the start/from point for either a port or register (From/Through). See Defining From/To/Through Points for Timing Exceptions , on page 151 for more information. Select the end/to point for either an output port or register. Specify a through point for a net or hierarchical port/pin (To/Through). Set the delay value (Max Delay). Check the Enabled box.
Multicycle paths	Delay Paths	See Defining Multicycle Paths , on page 155.

To define...	Pane	Do this to set the constraint...
False paths	Delay Paths	See Defining False Paths , on page 156 for details.
Global attributes	Attributes	Set Object Type to <global>. Select the object (Object). Set the attribute (Attribute) and its value (Value). Check the Enabled box.
Attributes	Attributes	Do either of the following: • Select the type of object (Object Type). Select the object (Object). Set the attribute (Attribute) and its value (Value). Check the Enabled box. • Set the attribute (Attribute) and its value (Value). Select the object (Object). Check the Enabled box.

Defining Input and Output Constraints

In addition to setting I/O delays in the SCOPE window as described in [Setting Clock and Path Constraints, on page 182](#), you can also set the Use clock period for unconstrained IO option.

- Open the SCOPE window, click Inputs/Outputs, and select the port (Port). You can set the constraint for
 - All inputs and outputs (globally in the top-level netlist)
 - For a whole bus
 - For single bits

You can specify multiple constraints for the same port. The software applies all the constraints; the tightest constraint determines the worst slack. If there are multiple constraints from different levels, the most specific overrides the more global. For example, if there are two bit constraints and two port constraints, the two bit constraints override the two port constraints for that bit. The other bits get the two port constraints.

- Specify the constraint value in the SCOPE window:
 - Select the type of delay: input or output (Type).
 - Type a delay value (Value).
 - Check the Enabled box, and save the constraint file in the project.

Make sure to specify explicit constraints for each I/O path you want to constrain.

- To determine how the I/O constraints are used during synthesis, do the following:
 - Select Project->Implementation Options, and click Constraints.
 - To use only the explicitly defined constraints disable Use clock period for unconstrained IO.
 - To synthesize with all the constraints, using the clock period for all I/O paths that do not have an explicit constraint enable Use clock period for unconstrained IO.
 - Synthesize the design. When you forward-annotate the constraints, the constraints used for synthesis are forward-annotated for place-and-route.

- Input or output ports with explicitly defined constraints, but without a reference clock (-ref option) are included in the System clock domain and are considered to belong to every defined or inferred clock group.
- If you do not meet timing goals after place-and-route and you need to adjust the input constraints; do the following:
 - Open the SCOPE window with the input constraint.
 - Use the `set_clock_route_delay` command to translate the -route option for the constraint, so that you can specify the actual route delay in nanoseconds, as obtained from the place-and-route results. Adding this constraint is equivalent to putting a register delay on the input register. Use it as a fudge factor to force the synthesis engine to accommodate a routing delay that is larger than expected.
 - Resynthesize your design.

Specifying Standard I/O Pad Types

You can specify a standard I/O pad type to use in the design. The equivalent Tcl command is `define_io_standard`.

1. Open the SCOPE window and go to the I/O Standard tab.
2. In the Port column, select the port. This determines the port type in the Type column.
3. Enter an appropriate I/O pad type in the I/O Standard column. The Description column shows a description of the I/O standard you selected.

For details of supported I/O standards, see [Industry I/O Standards, on page 181](#).

4. Where applicable, set other parameters like drive strength, slew rate, and termination.

You cannot set these parameter values for industry I/O standards whose parameters are defined by the standard.

Specifying Limits for FDC Wildcard Reporting

For FDC constraints specified with wildcard expressions, you can set a maximum limit to the number of such objects when they are reported in the constraint checker.

1. To reduce the number of objects, in the FDC file, specify the `set ::wildcard_report_limit` command with the value set to the upper limit you want to see.

The default is 1000, but the following command restricts the maximum to 150 objects:

```
set ::wildcard_report_limit 150
```

You can set it to 0, or any positive value that will fit into a 32-bit signed integer.

2. If you see want to see more objects than are reported (message MF892), you can increase the limit. For example:

```
set ::wildcard_report_limit 1000000000
```

Using the TCL View of SCOPE GUI

The TCL View of the SCOPE GUI is an advanced text file editor used for FPGA timing and design constraints. This text editor provides the following capabilities:

- Uses dynamic keyword expansion and tool tips for commands that
 - Automatically completes the command from a popup list
 - Displays complete command syntax as a tool tip
 - Displays parameter options for the command from a popup list
 - Includes a keyword command syntax help
- Checks command syntax and uses color indicators that
 - Validates commands and command syntax
 - Distinguishes between FPGA design constraints and SCOPE legacy constraints
- Allows for standard editor commands, such as copy, paste, comment/un-comment a group of lines, and highlighting of keywords

To use the TCL View of the SCOPE GUI:

1. Click on the TCL View of the SCOPE GUI.
2. You can specify FPGA design constraints as follows:
 - Type the command; after you type three characters a popup menu displays the design constraint command list. Select a command.
 - When you type a dash (-), the options popup menu list is displayed. Select an option.
 - When you hover over a command, a tool tip is displayed for the selected commands.

Command List Popup Menu

```

1 create_clock -period 10
2 set_false_path -from i:dmux.alua[7:0]
3
4 set
5
6 set_clock_groups
7 set_clock_latency
8 set_clock_uncertainty
9 set_false_path
10 set_hierarchy_separator
11 set_input_delay
12 set_max_delay
13 set_multicycle_path
14 set_output_delay
15 set_rtl_ff_names

```

Options List Popup Menu

```

1 create_clock -period 10
2 set_false_path -from i:dmux.alua[7:0]
3
4 set_clock_groups -
5
6
7
8
9 define_clock -fr

```

```

1 create_clock -period 10
2 set_false_path -from i:dmux.alua[7:0]
3
4 set_clock_groups
5
6
7
8
9 define_clock

```

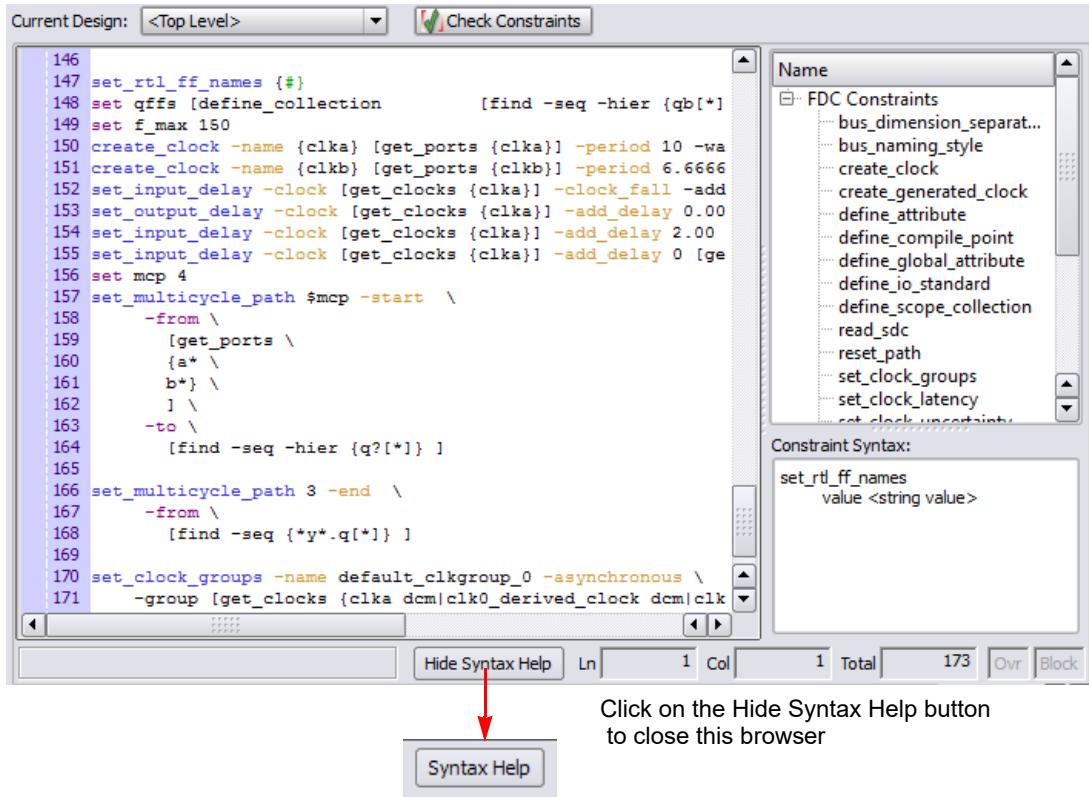
```

set_clock_groups
[-asynchronous]
[-exclusive]
[-logically_exclusive]
[-physically_exclusive]
[-name <string value>]
-group

```

Command Tool Tip

3. You can also specify a command by using the constraints browser that displays a constraints command list and associated syntax.
 - Double-click on the specified constraint to add the command to the editor window.
 - Then, use the constraint syntax window to help you specify the options for this command.
 - Click on the Hide Syntax Help button at the bottom of the editor window to close the syntax help browser.



- When you save this file, the constraint file is added to your project in the Constraint directory if the Add to Project option is checked on the New dialog box. Thereafter, you can double-click on the FDC constraint file to open it in the text editor.

Guidelines for Entering and Editing Constraints

- Enter or edit constraints as follows:
 - For attribute cells in the spreadsheet, click in the cell and select from the pull-down list of available choices.
 - For object cells in the spreadsheet, click in the cell and select from the pull-down list. When you select from the list, the objects automatically have the proper prefixes in the SCOPE window.

Alternatively, you can drag and drop an object from an HDL Analyst view into the cell, or type in a name. If you drag a bus, the software enters the whole bus (busA). To enter busA[3:0], select the appropriate bus bits before you drag and drop them. If you drag and drop or type a name, make sure that the object has the proper prefix identifiers:

Prefix Identifiers	Description for...
<i>v:design_name</i>	hierarchies or “views” (modules)
<i>c:clock_name</i>	clocks
<i>i:instance_name</i>	instances (blocks)
<i>p:port_name</i>	ports (off-chip)
<i>t:pin_name</i>	hierarchical ports, and pins of instantiated cells
<i>b:name</i>	bits of a bus (port)
<i>n:net_name</i>	internal nets

- For cells with values, type in the value or select from the pull-down list.
- Click the check box in the Enabled column to enable the constraint or attribute.
- Make sure you have entered all the essential information for that constraint. Scroll horizontally to check. For example, to set a clock constraint in the Clocks tab, you must fill out Enabled, Clock, Period, and Clock Group. The other columns are optional. For details about setting different kinds of constraints, go to the appropriate section listed in [Specifying SCOPE Constraints, on page 139](#).

2. For common editing operations, refer to this table:

To...	Do...
Cut, copy, paste, undo, or redo	Select the command from the popup (hold down the right mouse button to get the popup) or from the Edit menu.

To...	Do...
Copy the same value down a column	Select Fill Down (Ctrl-d) from the Edit or popup menus.
Insert or delete rows	Select Insert Row or Delete Rows from the Edit or popup menus.
Find text	Select Find from the Edit or popup menus. Type the text you want to find, and click OK.

3. Edit your constraint file if needed. If your naming conventions do not match these defaults, add the appropriate command specifying your naming convention to the beginning of the file, as shown in these examples:

	Default	You use	Add this to your file
Hierarchy separator	A.B	Slash: A/B	set_hierarchy_separator {/}
Naming bit 5 of bus ABC	ABC[5]	Underscore	bus_naming_style {%s_%d}
Naming row 2 bit 3 of array ABC [2x16]	ABC [2] [3]	Underscore ABC [2_3]	bus_dimension_separator_style {_}

Specifying Timing Exceptions

You can specify the following timing exception constraints, either from the SCOPE interface or by manually entering the Tcl commands in a file:

- Multicycle Paths – Paths with multiple clock cycles.
- False Paths – Clock paths that you want the synthesis tool to ignore during timing analysis and assign low (or no) priority during optimization.
- Max Delay Paths – Point-to-point delay constraints for paths.

The following shows you how to specify timing exceptions in the SCOPE GUI. For the equivalent Tcl syntax, see [Chapter 2, *Tcl Synthesis Commands*](#) in the *Reference Manual*.

- [Defining From/To/Through Points for Timing Exceptions](#), on page 151
- [Defining Multicycle Paths](#), on page 155
- [Defining False Paths](#), on page 156

For information about resolving timing exception conflicts, see [Conflict Resolution for Timing Exceptions, on page 197](#) in the *Reference Manual*.

Defining From/To/Through Points for Timing Exceptions

For multi-cycle path, false path, and maximum path delay constraints, you must define paths with a combination of From/To/Through points. Whenever the tool encounters a conflict in the way timing-exception constraints are written, see [Conflict Resolution for Timing Exceptions, on page 197](#) to determine how resolution occurs based on the priorities defined.

The following guidelines provide details for defining these constraints. You must specify at least one From, To, or Through point.

- In the From field, identify the starting point for the path. The starting point can be a clock, input or bidirectional port, or register. Only black box output pins are valid. To specify multiple starting points:
 - Such as the bits of a bus, enclose them in square brackets: A[15:0] or A[*].
 - Select the first start point from the HDL Analyst view, then drag and drop this instance into the From cell in SCOPE. For each subsequent

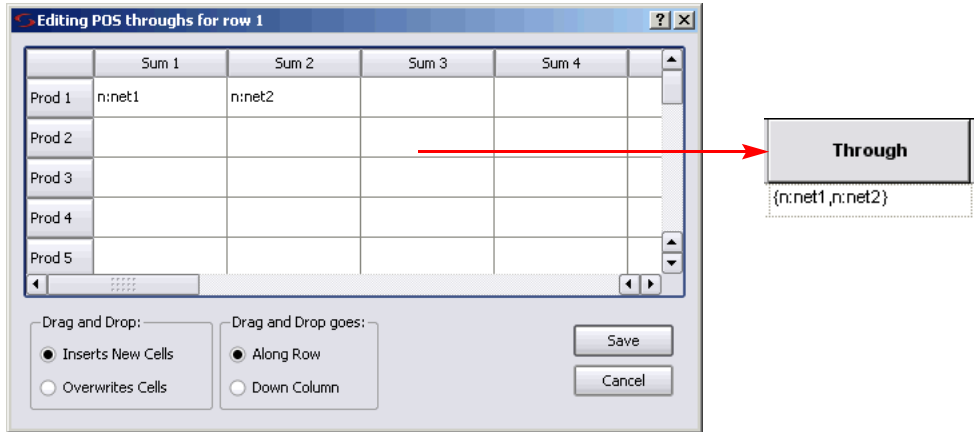
instance, press the Shift key as you drag and drop the instance into the From cell in SCOPE. For example, valid Tcl command format include:

```
set_multicycle_path -from {i:aq i:bq} 2
set_multicycle_path -from [i:aq i:bq] -through {n:xor_all} 2
```

- In the To field, identify the ending point for the path. The ending point can be a clock, output or bidirectional port, or register. Only black box input pins are valid. To specify multiple ending points, such as the bits of a bus, enclose them in square brackets: B[15:0].
- A single through point can be a combinational net, hierarchical port or instantiated cell pin. To specify a net:
 - Click in the Through field and click the arrow. This opens the Product of Sums (POS) interface.
 - Either type the net name with the n: prefix in the first cell or drag the net from an HDL Analyst view into the cell.
 - Click Save.

For example, if you specify n:net1, the constraint applies to any path passing through net1.

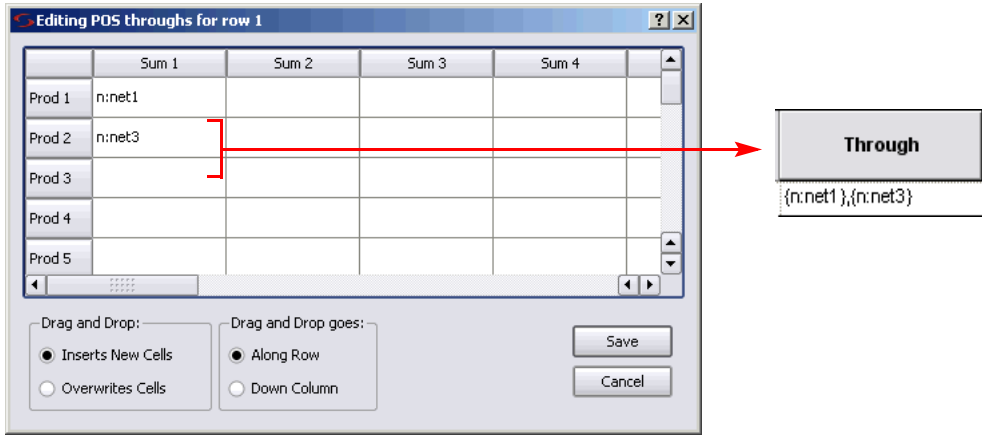
- To specify an OR when constraining a list of through points, you can type the net names in the Through field or you can use the POS UI. To do this:
 - Click in the Through field and click the arrow. This opens the Product of Sums interface.
 - Either type the first net name in a cell in a Prod row or drag the net from an HDL Analyst view into the cell. Repeat this step along the same row, adding other nets in the Sum columns. The nets in each row form an OR list.



- Alternatively, select Along Row in the SCOPE POS interface. In an HDL Analyst view, select all the nets you want in the list of through points. Drag the selected nets and drop them into the POS interface. The tool fills in the net names along the row. The nets in each row form an OR list.
- Click Save.

The constraint works as an OR function and applies to any path passing through any of the specified nets. In the example shown in the previous figure, the constraint applies to any path that passes through net1 *or* net2.

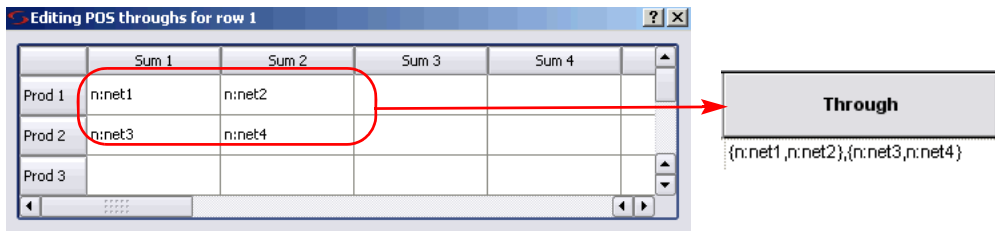
- To specify an AND when constraining a list of through points, type the names in the Through field or do the following:
 - Open the Product of Sums interface as described previously.
 - Either type the first net name in the first cell in a Sum column or drag the net from an HDL Analyst view into the cell. Repeat this step down the same Sum column.



- Alternatively, select Down Column in the SCOPE POS interface. In an HDL Analyst view, select all the nets you want in the list of through points. Drag the selected nets and drop them into the POS interface. The tool fills in the net names down the column.

The constraint works as an AND function and applies to any path passing through all the specified nets. In the previous figure, the constraint applies to any path that passes through net1 *and* net3.

- To specify an AND/OR constraint for a list of through points, type the names in the Through field (see the following figure) or do the following:
 - Create multiple lists as described previously.
 - Click Save.



In this example, the synthesis tool applies the constraint to the paths through all points in the lists as follows:

```
net1 AND net3  
OR net1 AND net4  
OR net2 AND net3  
OR net2 AND net4
```

Defining Multicycle Paths

To define a multicycle path constraint, use the Tcl `set_multicycle_path` command, or select the SCOPE Delay Paths tab and do the following;

1. From the Delay Type pull-down menu, select Multicycle.
2. Select a port or register in the From or To columns, or a net in the Through column. You must set at least one From, To, or Through point. You can use a combination of these points. See [Defining From/To/Through Points for Timing Exceptions, on page 151](#) for more information.
3. Select another port or register if needed (From/To/Through).
4. Type the number of clock cycles or nets (Cycles).
5. Specify the clock period to use for the constraint by going to the Start/End column and selecting either Start or End.

If you do not explicitly specify a clock period, the software uses the end clock period. The constraint is now calculated as follows:

$$\text{multicycle\_distance} = \text{clock\_distance} + (\text{cycles} - 1) * \text{reference\_clock\_period}$$

In the equation, `clock_distance` is the shortest distance between the triggering edges of the start and end clocks, `cycles` is the number of clock cycles specified, and `reference_clock_period` is either the specified start clock period or the default end clock period.

6. Check the Enabled box.

Defining False Paths

You define false paths by setting constraints explicitly on the Delay Paths tab or implicitly on the Clock tab. See [Defining From/To/Through Points for Timing Exceptions, on page 151](#) for object naming and specifying through points.

- To define a false path between ports or registers, select the SCOPE Delay Paths tab, and do the following:
 - From the Delay Type pull-down menu, select False.
 - Use the pull-down to select the port or register from the appropriate column (From/To/Through).
 - Check the Enabled box.

The software treats this as an explicit false constraint and assigns it the highest priority. Any other constraints on this path are ignored.

- To define a false path between two clocks, select the SCOPE Clocks tab, and assign the clocks to different clock groups:

The software implicitly assumes a false path between clocks in different clock groups. This false path constraint can be overridden by a maximum path delay constraint, or with an explicit constraint.

- To set an implicit false path on a path to/from an I/O port, do the following:
 - Select Project->Implementation Options->Constraints.
 - Disable Use clock period for unconstrained IO.

Finding Objects with Tcl find and expand

The Tcl find and expand commands are powerful search tools that you can use to quickly identify the objects you want. The following sections describe how to use these commands effectively:

- [Specifying Search Patterns for Tcl find](#), on page 157
- [Refining Tcl Find Results with -filter](#), on page 159
- [Using the Tcl Find Command to Define Collections](#), on page 161
- [Using the Tcl expand Command to Define Collections](#), on page 162
- [Checking Tcl find and expand Results](#), on page 163
- [Using Tcl find and expand in Batch Mode](#), on page 165

Once you have located objects with the find or expand commands, you can group them into collections, as described in [Using Collections, on page 167](#), and apply constraints to all the objects in the collection at the same time.

Specifying Search Patterns for Tcl find

The usage tips in the following table apply for Tcl find search patterns, regardless of whether you specify the find command in the SCOPE window or as a Tcl command. For full details of the command syntax, refer to [Tcl Find Syntax \(Standard\)](#), on page 152 of the *Reference Manual*.

Case rules

Use the case rules for the language from which the object was generated:

- VHDL: case-insensitive
- Verilog: case-sensitive. Make sure that the object name you type in the SCOPE window matches the Verilog name.

For mixed language designs, use the case rules for the parent module. The top level for this example is VHDL, so the following command finds any object in the current view that starts with either a or A:

```
find {a*} -nocase
```

Pattern matching

You have two pattern-matching choices:

- Specify the `-regex` argument, and then use regular expressions for pattern matching.
- Do not specify `-regex`, and use only the `*` and `?` wildcards for pattern matching.

For hierarchical instance names that use dots as separators, the dots must be escaped with a backward slash (`\`). For example: `abc\d`.

Scope of the search

The scope of the search varies, depending on where you enter the command. If you enter it in the SCOPE environment, the scope of the search is the entire database, but if it is entered in the Tcl window, the default scope of the search is the current HDL Analyst view. See [Comparison of Methods for Defining Collections](#), on page 167 for a list of the differences.

To set the scope to include the hierarchical levels below the current view in HDL Analyst, use the `-hier` argument. This example finds all objects below the current view that begin with a:

```
find {a*} -hier
```

Restricting search by
type of object

Use the `-object_type` argument. The following command finds all nets that contain syn:

```
find -net {*syn*}
```

Restricting search by
object property

Use the `-filter` option, as described in [Refining Tcl Find Results with -filter](#), on page 159.

Extending search
through the hierarchy

Use the `-flat` option. With this option, the `*` wildcard matches hierarchy separators as well as regular characters. In the following example, the command finds all instances that include `fft_stages` in their name, whether it just matches an instance name (`inst1_fft_stages_2`) or matches a hierarchical name that includes the hierarchy separator (`a1.fft_stages_xy`):

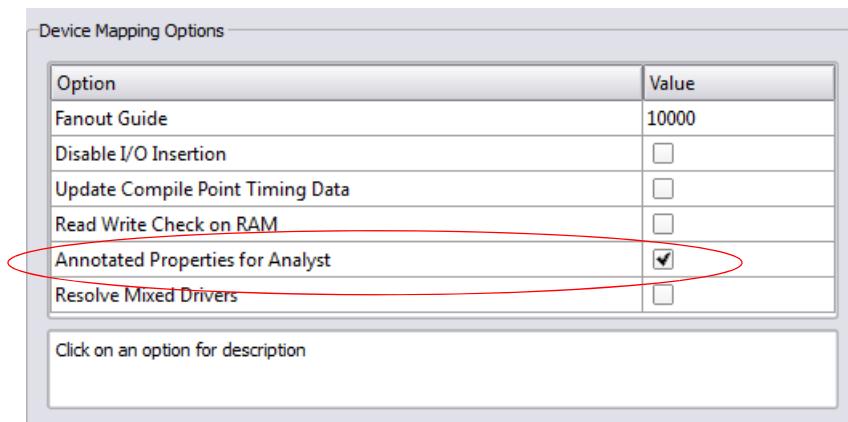
```
find -seq -flat *fft_stages* -print
```

Refining Tcl Find Results with -filter

The `-filter` option of the `find` command lets you further refine the objects located by the `find` command, according to their properties. When used with other commands, it can be a powerful tool for generating statistics and for evaluation. To filter your find results, follow these steps:

1. Enable property annotation.

- Select Project->Implementation Options. On the Device tab, enable Annotated Properties for Analyst. Alternatively, use the equivalent Tcl command: `set_option -run_prop_extract 1`.



- Compile or synthesize the design. After compilation, the tool annotates the design with properties that you can specify with the `-filter` option, like clock pins.
2. Specify the command using the find pattern as usual, and then specify the `-filter` option as the last argument:

```
find searchPattern -filter expression  
find searchPattern -filter !expression
```

With this command, the tool first finds objects that match the find *searchPattern*, and then further filters the found objects the according to the property criteria specified in `-filter expression`. Use the `!` character before *expression* if you want to select objects that do not match the properties specified in the filter *expression*.

expression can be a property name, specified as `@propertyName`, or a property name and value pair, specified as `@propertyName operator value`.

The following example finds registers in the current view that are clocked by myclk:

```
find -seq {*} -filter {@clock==myclk}
```

For further information about the command, see the following:

For...	See
Tips on using find search patterns	Specifying Search Patterns for Tcl find , on page 157
find syntax details	find , on page 151 in the Reference Manual
find -filter syntax details	find -filter , on page 165 in the Reference Manual

Examples of Useful Find -filter Commands

To find...	Use a command like this example...
Instances by slack value	<code>set slack [find -hier -inst {*} -filter @slack <= {-1.000}]</code>
Instances with negative slack	<code>set negFF [find -hier -inst {*} -filter @slack <= {0.0}]</code>
Instances within a slack range	<code>set slackRange [find -hier -inst {*} -filter @slack <= {-1.000} && @slack >= {+1.000}]</code>
Instances within a particular hierarchy	<code>find *fft_stages* -hier -filter @is_sequential -print</code>
Pins by fanout value	<code>set pinResult [find -pin *.CE -hier -filter {@fanout > 15 && @slack < 0.0} -print]</code>
Sequential elements within a clock domain	<code>set clk1FF [find -hier -seq * -filter {@clock==clk1}]</code>
Sequential components by primitive type	<code>set fdrse [find -hier -seq {*} -filter @view=={FDRSE}]</code>
Logic path between two collection of sequential hierarchical instances	<code>set srs_view [open_file ./rev_test/top.srs]</code> <code>set c1 [find -inst -seq {m2.d*}]</code> <code>set c2 [find -inst -seq {m1.do*}]</code> <code>select [expand -hier -from \$c1 -to \$c2]</code>

Using the Tcl Find Command to Define Collections

It is recommended that you use the SCOPE window rather than the Tcl window described here to specify the find command, for the reasons described in [Comparison of Methods for Defining Collections, on page 167](#).

The Tcl find command returns a collection of objects. If you want to create a collection of connectivity-based objects, use the Tcl expand command instead of find ([Specifying Search Patterns for Tcl find, on page 157](#)). This section lists some tips for using the Tcl find command.

1. Create a collection by typing the set command and assigning the results to a variable. The following example finds all instances with a primitive type DFF and assigns the collection to the variable \$result:

```
set result [find -hier -inst {*} -filter @ view == DFF]
```

The result is a random number like s:49078472, which is the collection of objects found. The following table lists some usage tips for specifying the

- find command. For full details of the syntax, refer to [Tcl Find Syntax \(Standard\)](#), on page 152 of the *Reference Manual*.
2. Check your find constraints. See [Checking Tcl find and expand Results](#), on page 163.
 3. Once you have defined the collection, you can view the objects in the collection, using one of the following methods, which are described in more detail in [Viewing and Manipulating Collections with Tcl Commands](#), on page 173:
 - Print the collection using the -print option to the find command.
 - Print the collection without carriage returns or properties, using c_list.
 - Print the collection in columns, with optional properties, using c_print.
 4. To manipulate the objects in the collection, use the commands described in [Viewing and Manipulating Collections with Tcl Commands](#), on page 173.
 5. Combine the Tcl find command with other commands:

To...	Combine with...
Create or copy objects; create collections	set define_collection
Generate reports for evaluation	c_list c_print
Generate statistics	c_info

Using the Tcl expand Command to Define Collections

The Tcl expand command returns a list of objects that are logically connected between the specified expansion points. This section contains tips on using the Tcl expand command to generate a collection of objects that are related by their connectivity. For the syntax details, refer to [expand](#), on page 173 in the *Command Reference Manual*.

1. Specify at least one from, to, or thru point as the starting point for the command. You can use any combination of these points.

The following example expands the cone of logic between reg1 and reg2.

```
expand -from {i:reg1} -to {i:reg2}
```

If you only specify a thru point, the expansion stops at sequential elements. The following example finds all elements in the transitive fanout and transitive fanin of a clock-enable net:

```
expand -thru {n:cen}
```

2. To specify the hierarchical scope of the expansion, use the `-hier` argument.

If you do not specify this argument, the command only works on the current view. The following example expands the cone of logic to `reg1`, including instances below the current level:

```
expand -hier -to {i:reg1}
```

If you only specify a thru point, you can use the `-level` argument to specify the number of levels of expansion. The following example finds all elements in the transitive fanout and transitive fanin of a clock-enable net across one level of hierarchy:

```
expand -thru {n:cen} -level 1
```

3. To restrict the search by type of object, use the `-object_type` argument.

The following command finds all pins driven by the specified pin.

```
expand -pin -from {t:i_and3.z}
```

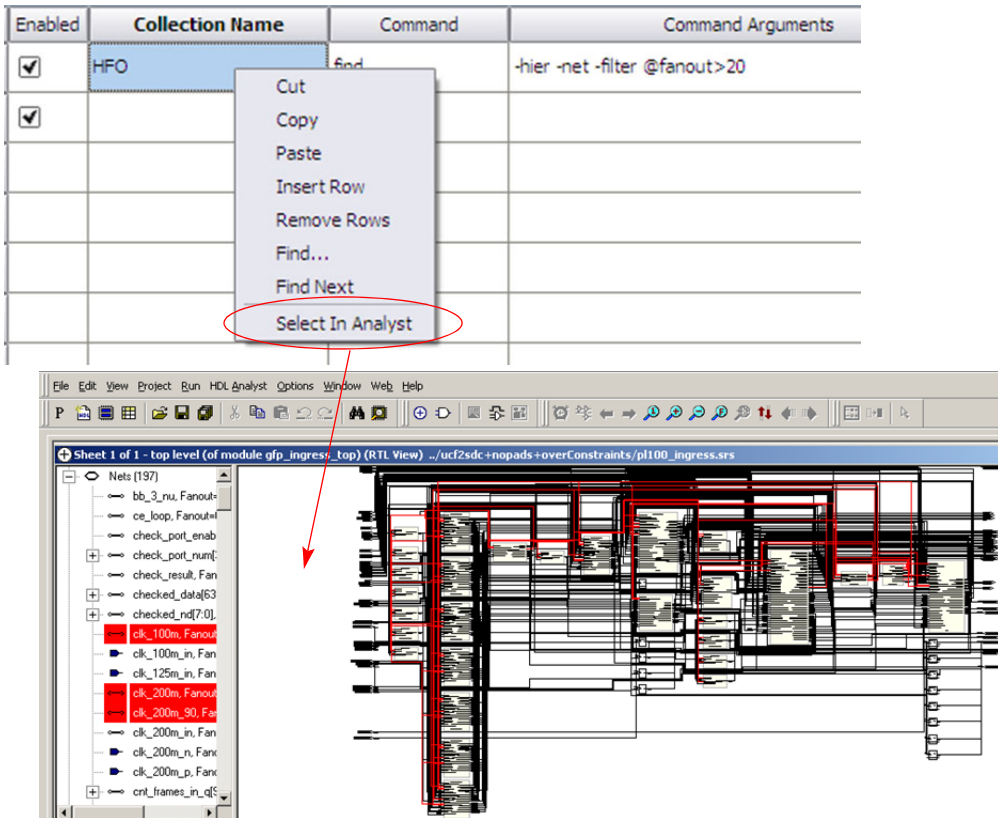
4. To print a list of the objects found, either use the `-print` argument to the `expand` command, or use the `c_print` or `c_list` commands (see [Creating Collections using Tcl Commands](#), on page 170).

Checking Tcl find and expand Results

You must check the validity of the find constraints you set. Use the methods described below.

1. Run the Constraints Checker, either from the UI or at the command line:
 - From the UI, select Run->Constraint Check.
 - At the command line specify the `-run constraint_check` option to the synthesis tool command. For example: `synplify_pro -batch design.prj -run constraint_check`.

- If there are issues, the tool reports them in the *design_cck.rpt* report file. Check the Summary and Inapplicable Constraints sections in this file.
2. To list objects selected by the find or expand commands, use one of these methods:
 - List the results by specifying the -print option to the command.
 - List the results with the `c_list` command.
 - Print out the results one item per line, using the `c_print` command.
 3. To visually validate the objects selected by the find or expand commands, do the following:
 - Run the command and save the results as a collection.
 - On the SCOPE Collections tab, select the collection.
 - Right-click and choose **Select in Analyst**. The objects in the collection are highlighted in the RTL view. The example below shows high fanout nets that drive more than 20 destinations.



Using Tcl find and expand in Batch Mode

When you use the Tcl find command in batch mode, you must specify the `open_design` command before the find or expand commands.

1. Create the Tcl file to be run in batch mode, making sure that the `open_design` command precedes the find/expand commands you want.

You cannot include the Tcl find command in Timing Analyzer scripts. Instead, run Tcl Find to TXT command and use the results.

2. Run the script at the command line. For example, if the file created in step 1 was called `analysis.tcl`, specify it at the command line, as shown below:

```
synplify_pro -batch analysis.tcl
```

The tool generates two text files as specified, with the results of the two searches.

Using Collections

A collection is a defined group of objects. The advantage offered by collections is that you can operate on all the objects in the collection at the same time. A collection can consist of a single object, multiple objects, or even other collections. You can either define collections in the SCOPE window or type the commands in the Tcl script window.

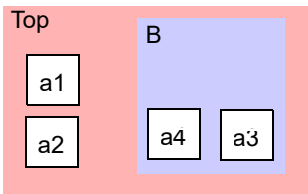
- [Creating and Using SCOPE Collections](#), on page 168
- [Creating Collections using Tcl Commands](#), on page 170
- [Viewing and Manipulating Collections with Tcl Commands](#), on page 173

Comparison of Methods for Defining Collections

You can enter the find and expand Tcl commands that are used to define collections in either the Tcl script window or in the SCOPE window. It is recommended that you use the SCOPE interface for the reasons outlined below:

	SCOPE Window	Tcl Window
Database used	Top level; includes all objects. See the example below.	Current Analyst view, which might be a lower-level view. If the current view is the Technology view after mapping, objects might be renamed, replicated, or removed.
Persistence	Collection saved in project file.	Collection only valid for the current session; you must redefine it the next time you open the project.
Constraints	Can apply to collection.	Cannot apply to collection.

In the design shown below, if you push down into B, and then type `find -hier a*` in the Tcl window, the command finds a3 and a4. However if you cut and paste the same command into the SCOPE Collections tab, your results would include a1, a2, a3, and a4, because the SCOPE interface uses the top-level database and searches the entire hierarchy.



Creating and Using SCOPE Collections

The following procedure shows you how to define collections in the SCOPE window. The SCOPE method is preferred over typing the commands in the Tcl window ([Creating Collections using Tcl Commands, on page 170](#)) for the reasons described in [Comparison of Methods for Defining Collections, on page 167](#).

1. Define a collection by doing the following:
 - Open the SCOPE window and click the Collections tab.
 - In the Name column, type a name for the collection.

	Enable	Name	Command
1	☒	find_all	find -hier -inst {*usbSlaveControl.u_endpMux.*}
2	☒	find_reg	find -hier -seq {*usbSlaveControl.u_endpMux.*}
3	☒	find_comb	find -hier -inst {*usbSlaveControl.u_endpMux.*} -filter@is_combination

Collections

- In the Command column, enter the command. See the *Command Reference* for complete syntax details. Additional information about specifying search patterns is described in [Specifying Search Patterns for Tcl find, on page 157](#).

You can also paste in a command. If you cut and paste a Tcl Find command from the Tcl window into the SCOPE Collections tab, remember that the SCOPE interface works on the top-level database, while the find command in the Tcl window works on the current level displayed in the Analyst view.

Objects in a collection do not have to be of the same type. The collections shown in the preceding figure do the following:

Collection	Finds...
find_all	All components in the module endpMux
find_reg	All registers in the module endpMux
find_comb	All combinational components under endpMux

The collections you define appear in the SCOPE pull-down object lists, so you can use them to define constraints.

You can crossprobe the objects selected by the find and expand commands by right-clicking and choosing **Select in Analyst column**. The schematic views highlight the objects located by these commands. For other viewing operations, see [Viewing and Manipulating Collections with Tcl Commands](#), on page 173.

Note: Using collections with Tcl control constructs (such as if, for, foreach, and while) can produce unexpected synthesis results. Avoid defining constraints for collections with control constructs, especially since the constraint checker does not recognize these built-in Tcl commands.

2. To create a collection that is made up of other collections, do this:
 - Define the collections as described in the previous step. These collections must be defined before you can concatenate them or add them together in a new collection.
 - To concatenate collections or add to collections, type a name for the new collection in the Name column. Type the appropriate operator command like `c_union` or `c_diff` in the Command column. See [Creating Collections using Tcl Commands](#), on page 170 for a list of available commands and the *Command Reference* for their syntax.

The software saves the collection information in the constraint file for the project.

3. To apply constraints to a collection do the following:
 - Define a collection as described in the previous steps.

- Go to the appropriate SCOPE tab and specify the collection name where you would normally specify the object name. Collections defined in the SCOPE interface are available from the pull-down object lists. The following figure shows the collections defined in step 1 available for setting a false path constraint.

	Enabled	Delay Type	From	To	Through	Start/End	Cycles	Max Delay(ns)	Comment
1	☒	False	special_regs.status[7:0]						
2	☐		i:decode.opcode_goto						
3	☐		i:decode.opcode_call						
4	☐		i:decode.opcode_retlw						
5	☐		i:decode.skip						
6	☐		i:decode.decodes[13:0]						
			i:prgmctr...r_0 [10:0]						
			i:prgmctr...r_1 [10:0]						
			i:prgmctr...r_2 [10:0]						
			i:prgmctr...r_3 [10:0]						
			i:prgmctr...r_4 [10:0]						

Delay Paths

- Specify the rest of the constraint as usual. The software applies the constraint to all the objects in the collection.

Creating Collections using Tcl Commands

This section describes how to use the Tcl collection commands at the command line or in a script instead of entering them in the SCOPE window (*Creating and Using SCOPE Collections*, on page 168). There are differences in operation depending on where the collection commands are entered, and it is recommended that you use the SCOPE window, for the reasons described in *Comparison of Methods for Defining Collections*, on page 167.

For details of the syntax for the commands described here, refer to *Collections*, on page 163 in the *Reference Manual*.

1. To create a collection using a Tcl command line command, name it with the `set` command and assign it to a variable.

A collection can consist of individual objects, Tcl lists (which can consist of a single element), or other collections. You can embed the Tcl `find` and `expand` commands in the `set` command to locate objects for the collection (see *Using the Tcl Find Command to Define Collections*, on page 161 and *Specifying Search Patterns for Tcl find*, on page 157). The following example creates a collection called `my_collection` which consists of all the modules (views) found by the embedded `find` command:

```
set my_collection [find -view {*}]
```

2. To create collections derived from other collections, do the following:
 - Define a new variable for the collection.
 - Create the collection with one of the operator commands from this table:

To...	Use this command...
Add objects to a collection	<code>c_union</code> . See Examples: <code>c_union</code> Command , on page 172
Concatenate collections	<code>c_union</code> . See Examples: <code>c_union</code> Command , on page 172.
Isolate differences between collections	<code>c_diff</code> . See Examples: <code>c_diff</code> Command , on page 172.
Find common objects between collections	<code>c_intersect</code> . See Examples: <code>c_intersect</code> Command , on page 172.
Find objects that belong to just one collection	<code>c_symdiff</code> . See Examples: <code>c_symdiff</code> Command , on page 173.

3. If your Tcl collection includes instances that use special characters, make sure to use extra curly braces or use a backslash to escape the special character.

Curly Braces { }	<code>define_scope_collection GRP_EVENT_PIPE2 {find -seq {EventMux[2\].event_inst?_sync[*]} -hier}</code> <code>define_scope_collection mytn {find -inst {i:count1.co[*]}}</code>
Backslash Escape Character (\\)	<code>define_scope_collection mytn {find -inst i:count1.co\[*\]}</code>

Once you have created a collection, you can do various operations on the objects in the collection (see [Viewing and Manipulating Collections with Tcl Commands](#), on page 173), but you cannot apply constraints to the collection.

Examples: **c_union** Command

This example adds the `reg3` instance to `collection1`, which contains `reg1` and `reg2` and names the new collection `sumCollection`.

```
set sumCollection [c_union $collection1 {i:reg3}]
c_list $sumCollection
    {"i:reg1" "i:reg2" "i:reg3"}
```

If you added `reg2` and `reg3` with the `c_union` command, the command removes the redundant instances (`reg2`) so that the new collection would still consist of `reg1`, `reg2`, and `reg3`.

This example concatenates `collection1` and `collection2` and names the new collection `combined_collection`:

```
set combined_collection [c_union $collection1 $collection2]
```

Examples: **c_diff** Command

This example compares a list to a collection (`collection1`) and creates a new collection called `subCollection` from the list of differences:

```
set collection1 {i:reg1 i:reg2}
set subCollection [c_diff $collection1 {i:reg1}]
c_print $subCollection
    "i:reg2"
```

You can also use the command to compare two collections:

```
set reducedCollection [c_diff $collection1 $collection2]
```

Examples: **c_intersect** Command

This example compares a list to a collection (`collection1`) and creates a new collection called `interCollection` from the objects that are common:

```
set collection1 {i:reg1 i:reg2}
set interCollection [c_intersect $collection1 {i:reg1 i:reg3}]
c_print $interCollection
    "i:reg1"
```

You can also use the command to compare two collections:

```
set common_collection [c_intersect $collection1 $collection2]
```

Examples: c_syndiff Command

This example compares a list to a collection (collection1) and creates a new collection called diffCollection from the objects that are different. In this case, reg1 is excluded from the new collection because it is in the list and collection1.

```
set collection1 {i:reg1 i:reg2}
set diffCollection [c_syndiff $collection1 {i:reg1 i:reg3}]
c_list $diffCollection
    {"i:reg2" "i:reg3"}
```

You can also use the command to compare two collections:

```
set syndiff_collection [c_syndiff $collection1 $collection2]
```

Examples: Names with Special Characters

Your instance names might include special characters, as for example when your HDL code uses a generate statement. If your instance names have special characters, do the following:

Make sure that you include extra curly braces {}, as shown below:

```
define_scope_collection GRP_EVENT_PIPE2 {find -seq
    {EventMux\[2\].event_inst?_sync[*]} -hier}
define_scope_collection mytn {find -inst {i:count1.co[*]}}
```

Alternatively, use a backslash to escape the special character:

```
define_scope_collection mytn {find -inst i:count1.co\[*\]}
```

Viewing and Manipulating Collections with Tcl Commands

The following section describes various operations you can do on the collections you defined. For full details of the syntax, see [Collections, on page 163](#) in the *Reference Manual*.

1. To view the objects in a collection, use one of the methods described in subsequent steps:
 - Select the collection in an HDL Analyst view (step 2).
 - Print the collection without carriage returns or properties (step 3).
 - Print the collection in columns (step 4).
 - Print the collection in columns with properties (step 5).

2. To select the collection in an HDL Analyst view, type `select <collection>`.

For example, `select $result` highlights all the objects in the `$result` collection.

3. To print a simple list of the objects in the collection, uses the `c_list` command, which prints a list like the following:

```
{i:EP0RxFifo.u_fifo.dataOut[0]} {i:EP0RxFifo.u_fifo.dataOut[1]}
{i:EP0RxFifo.u_fifo.dataOut[2]} ...
```

The `c_list` command prints the collection without carriage returns or properties. Use this command when you want to perform subsequent Tcl commands on the list. See [Example: `c\_list` Command, on page 176](#).

4. To print a list of the collection objects in column format, use the `c_print` command. For example, `c_print $result` prints the objects like this:

```
{i:EP0RxFifo.u_fifo.dataOut[0]}
{i:EP0RxFifo.u_fifo.dataOut[1]}
{i:EP0RxFifo.u_fifo.dataOut[2]}
{i:EP0RxFifo.u_fifo.dataOut[3]}
{i:EP0RxFifo.u_fifo.dataOut[4]}
{i:EP0RxFifo.u_fifo.dataOut[5]}
```

5. To print a list of the collection objects and their properties in column format, use the `c_print` command as follows:

- Annotate the design with a full list of properties by selecting Project->Implementation Options, going to the Device tab, and enabling Annotated Properties for Analyst. Synthesize the design. If you do not enable the annotation option, properties like clock pins will not be annotated as properties.
- Check the properties available by right-clicking on the object in the HDL Analyst view and selecting Properties from the popup menu. You see a window with a list of the properties that can be reported.
- In the Tcl window, type the `c_print` command with the `-prop` option. For example, typing `c_print -prop slack -prop view -prop clock $result` lists the objects in the `$result` collection, and their slack, view and clock properties.

```
Object Name          slack  view  clock
{i:EP0RxFifo.u_fifo.dataOut[0]} 0.3223 "FDE" clk
{i:EP0RxFifo.u_fifo.dataOut[1]} 0.3223 "FDE" clk
{i:EP0RxFifo.u_fifo.dataOut[2]} 0.3223 "FDE" clk
{i:EP0RxFifo.u_fifo.dataOut[3]} 0.3223 "FDE" clk
```

```
{i:EP0RxFifo.u_fifo.dataOut[4]} 0.3223 "FDE" clk
{i:EP0RxFifo.u_fifo.dataOut[5]} 0.3223 "FDE" clk
{i:EP0RxFifo.u_fifo.dataOut[6]} 0.3223 "FDE" clk
{i:EP0RxFifo.u_fifo.dataOut[7]} 0.3223 "FDE" clk
{i:EP0TxFifo.u_fifo.dataOut[0]} 0.1114 "FDE" clk
{i:EP0TxFifo.u_fifo.dataOut[1]} 0.1114 "FDE" clk
```

- To print out the results to a file, use the `c_print` command with the `-file` option. For example, `c_print -prop slack -prop view -prop clock $result -file results.txt` writes out the objects and properties listed above to a file called `results.txt`. When you open this file, you see the information in a spreadsheet format.

6. You can do a number of operations on a collection, as listed in the following table. For details of the syntax, see [Collections, on page 163](#) in the *Reference Manual*.

To...	Do this...
Copy a collection	Create a new variable for the copy and copy the original collection to it with the <code>set</code> command. When you make changes to the original, it does not affect the copy, and vice versa. <pre>set my_collection_copy \$my_collection</pre>
List the objects in a collection	Use the <code>c_print</code> command to view the objects in a collection, and optionally their properties, in column format: <pre>"v:top" "v:block_a" "v:block_b"</pre> Alternatively, you can use the <code>-print</code> option to an operation command to list the objects.
Generate a Tcl list of the objects in a collection	Use the <code>c_list</code> command to view a collection or to convert a collection into a Tcl list. You can manipulate a Tcl list with standard Tcl commands. In addition, the Tcl collection commands work on Tcl lists. This is an example of <code>c_list</code> results: <pre>{"v:top" "v:block_a" "v:block_b"}</pre> Alternatively, you can use the <code>-print</code> option to an operation command to list the objects.

Example: `c_list` Command

The following provides a practical example of how to use the `c_list` command. This example first finds all the CE pins with a negative slack that is less than 0.5 ns and groups them in a collection:

```
set get_components_list [c_list [find -hier -pin {*.CE} -filter
@slack < {0.5}]]
```

The `c_list` command returns a list:

```
{t:EP0RxFifo.u_fifo.dataOut[0].CE}
{t:EP0RxFifo.u_fifo.dataOut[1].CE}
{t:EP0RxFifo.u_fifo.dataOut[2].CE} ...
```

You can use the list to find the terminal (pin) owner:

```
proc terminal_to_owner_instance {terminal_name terminal_type} {
    regsub -all $terminal_type$ $terminal_name {} suffix
    regsub -all {^t:} $suffix {i:} prefix
    return $prefix
}

foreach get_component $get_components_list {
    append owner [terminal_to_owner_instance $get_component {.CE}]
    " "
}

puts "terminal owner is $owner"
```

This returns the following, which shows that the terminal (pin) has been converted to the owning instance:

```
terminal owner is i:EP0RxFifo.u_fifo.dataOut[0]
i:EP0RxFifo.u_fifo.dataOut[1] i:EP0RxFifo.u_fifo.dataOut[2]
```


Converting SDC to FDC

The `sdc2fdc` Tcl shell command translates legacy FPGA timing constraints to Synopsys FPGA timing constraints. From the Tcl command line in the synthesis tool, the `sdc2fdc` command scans the input SDC files and attempts to convert constraints for the implementation.

To run the `sdc2fdc` Tcl shell command:

1. Load your Project file.
2. From the Tcl command line, type:

```
sdc2fdc
```
3. Check the constraint results directory for details about this translation.
4. The new constraints file is automatically updated for your project. Save the new settings.

The constraint results directory is created at

projectDir/FDC_constraints/implName

This directory includes the following results files:

- *topLevel/translated.fdc* – Contains the Synopsys FPGA design constraints (FPGA design constraints and the Synopsys standard timing constraints)
 - *topLevel|compilePoint/translate.log* – Contains details about the translation. Translation error messages explain issues and how to fix them. Any translation errors not addressed when you run synthesis appear in the SRR log file, but does not stop synthesis from running.
5. Open the FDC file resulting from translation in the FPGA SCOPE editor to check these constraints and make any changes to them.
 6. Run the constraints checker.
 7. Save this version of the FDC to run synthesis.

For information about the FDC file, see [FDC Constraints, on page 132](#).

For details about the translated files and troubleshooting guidelines, see [sdc2fdc Conversion, on page 135](#).

Using the SCOPE Editor (Legacy)

You can use the Legacy SCOPE editor for the SDC constraint files created before release version G-2012.09. However, it is recommended that you translate your SDC files to FDC files to enable the latest version of the SCOPE editor and to utilize the enhanced timing constraint handling in the tool. The latest version of the SCOPE editor automatically formats timing constraints using Synopsys Standard syntax (such as `create_clock`, and `set_multicycle_path`).

To do this, add your SDC constraint files to your project and run the following at the command line:

```
% sdc2fdc
```

This feature translates all SDC files in your project.

If you choose to do so, the following procedure shows you how to use the legacy SCOPE editor to create constraints for the constraint file (SDC).

- 1. Open an existing file for editing.
 - Make sure you have closed the SCOPE window, or you could overwrite previous constraints.
 - Double-click on an existing constraint file (`sdc`) in the project.
 - Select File->Open, set the Files of Type filter to Constraint Files (`sdc`) and open the file you want.
- 2. Enter the timing or design constraints you need.

Use SCOPE...	To Define...
Clocks	Clock frequencies define_clock. See Defining Clocks , on page 185 for additional information.
	Clock frequency other than the one implied by the signal on the clock pin syn_reference_clock (attribute). See Defining Clocks , on page 185 for additional information
	Clock domains with asymmetric duty cycles define_clock. See Defining Clocks , on page 185 for additional information

Use SCOPE...	To Define...
Clock to Clock	Edge-to-edge clock delays define_clock_delay. See Defining Clocks , on page 185 for additional information
Collections	Set constraints for a group of objects you have defined as a collection with the Tcl command.
Inputs/Outputs	Speed up paths feeding into a register define_reg_input_delay. Speed up paths coming from a register define_reg_output_delay.
Registers	Input delays from outside the FPGA define_input_delay. See Defining Input and Output Constraints (Legacy) , on page 195 for additional information Output delays from your FPGA define_output_delay. See Defining Input and Output Constraints (Legacy) , on page 195 for additional information
Delay Paths	Paths with multiple clock cycles define_multicycle_path. See Defining Multicycle Paths , on page 155 for additional information False paths (certain technologies) define_false_path. See Defining False Paths (Legacy) , on page 196 for additional information. Path delays define_path_delay. See Defining From/ To/ Through Points for Timing Exceptions , on page 151 for additional information
Attributes	Assign attributes for objects specifying their values
I/O Standards	Define an I/O standard for ports
Compile Points	Specify compile points for your design
Other	Enter newly-supported constraints for advanced users.

Entering and Editing SCOPE Constraints (Legacy)

Enter constraints directly in the SCOPE window. You can use the Initialize Constraint panel to enter default constraints, and then use the direct method to modify, add, or delete constraints.

The tool also lets you add constraints automatically. For information about auto constraints, see [Using Auto Constraints, on page 377](#).

1. Click the appropriate tab at the bottom of the window to enter the kind of constraint you want to create:

To define...	Click...
Clock frequency for a clock signal output of clock divider logic A specific clock frequency that overrides the global frequency	Clocks
Edge-to-edge clock delay that overrides the automatically calculated delay.	Clock to Clock
Constraints for a group of objects you have defined as a collection with the Tcl command. For details, see Creating and Using SCOPE Collections , on page 168.	Collections
Input/output delays that model your FPGA input/output interface with the outside environment	Inputs/ Outputs
Delay constraints for paths feeding into/out of registers	Registers
Paths that require multiple clock cycles	Delay Paths
Paths to ignore for timing analysis (false paths)	Delay Paths
Maximum delay for paths	Delay Paths
Attributes, like <code>syn_reference_clock</code> , that were not entered in the source files	Attributes
I/O standards for any port in the I/O Standard panel of the SCOPE window.	I/O Standard
Compile points in a top-level constraint file. See Synthesizing Compile Points , on page 469 for more information about compile points.	Compile Points
Place and route tool constraints	Other
Other constraints not used for synthesis, but which are passed to other tools. For example, multiple clock cycles from a register or input pin to a register or output pin	

The SCOPE window displays columns appropriate to the kind of constraint you picked. You can now enter constraints using the wizard, or work directly in the SCOPE window.

2. Save the file by clicking the Save icon and naming the file.

The software creates a TCL constraint file (`sdc`). See [Working with Constraint Files, on page 47](#) for information about the commands in this file.

3. To apply the constraints to your design, you must add the file to the project now or later.
 - Add it immediately by clicking Yes in the prompt box that opens after you save the constraint file.
 - Add it later, following the procedure for adding a file described in [Making Changes to a Project, on page 58](#).

Specifying SCOPE Timing Constraints (Legacy)

You can define timing constraints in the SCOPE GUI, which automatically generates a Tcl constraints file, or manually with a text editor, as described in [Using a Text Editor for Constraint Files \(Legacy\), on page 49](#).

The SCOPE GUI is much easier to use, and you can define various timing constraints in it. For the equivalent Tcl syntax, see [Chapter 2, Tcl Synthesis Commands](#) in the *Reference Manual*. See the following for different timing constraints:

- [Entering Default Constraints, on page 182](#)
- [Setting Clock and Path Constraints, on page 182](#)
- [Defining Clocks, on page 185](#)
- [Defining Input and Output Constraints \(Legacy\), on page 195](#)
- [Specifying Standard I/O Pad Types, on page 144](#)

To set constraints for timing exceptions like false paths and multicycle paths, see [Specifying Timing Exceptions, on page 151](#).

Entering Default Constraints

To edit or set individual constraints, or to create constraints in the Other tab, work directly in the SCOPE window ([Setting Clock and Path Constraints, on page 182](#)). For auto constraints in the Synplify Pro tool, see [Using Auto Constraints, on page 377](#). To apply the constraints, add the file to the project according to the procedure described in [Making Changes to a Project, on page 58](#). The constraints file has an `sdc` extension. See [Working with Constraint Files, on page 47](#) for more information about constraint files.

Setting Clock and Path Constraints

The following table summarizes how to set different clock and path constraints from the SCOPE window. For information about setting compile point constraints or attributes, see [Synthesizing Compile Points, on page 469](#) for more information about compile points and [Specifying Attributes Using the SCOPE Editor, on page 89](#). For information about setting default constraints, see [Entering Default Constraints, on page 182](#).

To define...	Pane	Do this to set the constraint...
Clocks	Clock	Select the clock object (Clock). Specify a clock name (Clock Alias), if required. Type a frequency value (Frequency) or a period (Period). Change the default Duty Cycle or set Rise/Fall At, if needed. Change the default clock group, if needed Check the Enabled box. See Defining Clocks , on page 185 for information about clock attributes.
Virtual clocks	Clock	Set the clock constraints as described for clocks, above. Check the Virtual Clock box.
Route delay	Clock Inputs/ Outputs Registers	Specify the route delay in nanoseconds. Refer to Defining Clocks , on page 185 , Defining Input and Output Constraints (Legacy) , on page 195 and the Register Delays section of this table details.
Edge-to-edge clock delay	Clock to Clock	Select the starting edge for the delay constraint (From Clock Edge). Select the ending edge for the constraint (To Clock Edge). Enter a delay value. Mark the Enabled check box.

To define...	Pane	Do this to set the constraint...
Input/output delays	Inputs/ Outputs	See Defining Input and Output Constraints (Legacy) , on page 195 for information about setting I/O constraints.
Register delays	Registers	Select the register (Register). Select the type of delay, input or output (Type). Type a delay value (Value). Check the Enabled box. If you do not meet timing goals after place-and-route, adjust the clock constraint as follows: • In the Route column for the constraint, specify the actual route delay (in nanoseconds), as obtained from the place-and-route results. Adding this constraint is equivalent to putting a register delay on that input register. • Resynthesize your design.
Maximum path delay	Delay Path	Select the Delay Type path of Max Delay. Select the port or register (From/Through). See Defining From/ To/ Through Points for Timing Exceptions , on page 151 for more information. Select another port or register if needed (To/Through). Set the delay value (Max Delay). Check the Enabled box.
Multi-cycle paths	Delay Paths	See Defining Multicycle Paths , on page 155.
False paths	Delay Paths Clock to Clock	See Defining False Paths (Legacy) , on page 196 for details.

To define...	Pane	Do this to set the constraint...
Global attributes	Attributes	Set Object Type to <global>. Select the object (Object). Set the attribute (Attribute) and its value (Value). Check the Enabled box.
Attributes	Attributes	Do either of the following: • Select the type of object (Object Type). Select the object (Object). Set the attribute (Attribute) and its value (Value). Check the Enabled box. • Set the attribute (Attribute) and its value (Value). Select the object (Object). Check the Enabled box.
Other	Other	Type the TCL command for the constraint (Command). Enter the arguments for the command (Arguments). Check the Enabled box.

Defining Clocks

Clock frequency is the most important timing constraint, and must be set accurately. If you are planning to auto constrain your design ([Using Auto Constraints, on page 377](#)), do not define any clocks. The following procedures show you how to define clocks and set clock groups and other constraints that affect timing:

- [Defining Clock Frequency](#), on page 186
- [Constraining Clock Enable Paths](#), on page 190
- [Defining Other Clock Requirements](#), on page 191

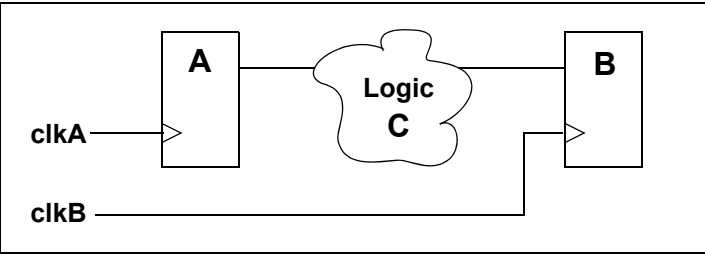
Defining Clock Frequency

This section shows you how to define clock frequency either through the GUI or in a constraint file. See [Defining Other Clock Requirements, on page 191](#) for other clock constraints. If you want to use auto constraints, do not define your clocks.

1. Define a realistic global frequency for the entire design, either in the Project view or the Constraints tab of the Implementation Options dialog box.

This target frequency applies to all clocks that do not have specified clock frequencies. If you do not specify any value, a default value of 1 MHz (or 1000 ns clock period) applies to all timing paths whenever the clock associated with both start and end points of the path is not specified. Each clock that uses the global frequency is assigned to its own clock group. See [Defining Other Clock Requirements, on page 191](#) for more information about clock group settings.

The global frequency also applies to any purely combinational paths. The following figure shows how the software determines constraints for specified and unspecified start or end clocks on a path:



If clkA is...	And clkB is...	The effect for logic C is...
Undefined	Defined	The path is unconstrained unless you specify that clkB be constrained to the inferred clock domain for clkA
Defined	Undefined	The path is unconstrained unless you specify that clkA be constrained to the inferred clock domain for clkB.
Defined	Defined	For related clocks in the same clock group, the relationship between clocks is calculated; all other paths between the clocks are treated as false paths.
Undefined	Undefined	The path is unconstrained.

2. Define frequency for individual clocks on the Clocks tab of the SCOPE window (define_clock constraint).
 - Specify the frequency as either a frequency in the Frequency column (-freq Tcl option) or a time period in the Period column (-period Tcl option). When you enter a value in one column, the other is calculated automatically.
 - For asymmetrical clocks, specify values in the Rise At (-rise) and Fall At (-fall) columns. The software automatically calculates and fills out the Duty Cycle value.

The software infers all clocks, whether declared or undeclared, by tracing the clock pins of the flip-flops. However, it is recommended that you specify frequencies for all the clocks in your design. The defined frequency overrides the global frequency. Any undefined clocks default to the global frequency.

3. Define internal clock frequencies (clocks generated internally) on the SCOPE Clocks tab (define_clock constraint). Apply the constraint according to the source of the internal clock.

Source	Add SCOPE constraint/define_clock to...
Register	Register.
Instance, like a PLL or clock DLL	Instance. If the instance has more than one clock output, apply the clock constraints to each of the output nets, making sure to use the n: prefix (to signify a net) in the SCOPE table.
Combinational logic	Net. Make sure to use the n: prefix in the SCOPE interface.

4. For signals other than clocks, define frequencies with the syn_reference_clock attribute. You can add this attribute on the SCOPE Attributes tab, as follows:
 - Define a dummy clock on the Clocks tab (define_clock constraint).
 - Add the syn_reference_clock attribute (Attributes tab) to the affected registers to apply the clock. In the constraint file, you can use the Find command to find all registers enabled by a particular signal and then apply the attribute:

```
define_clock -virtual dummy -period 40.0
define_attribute {find -seq * -hier -filter @(enable == en40)}
syn_reference_clock dummy
```

In earlier releases, limited clocking resources might have forced you to use an enable signal as a clocking signal, and use the `syn_reference_clock` attribute to define an enable frequency. However, because of changes in the reporting of clock start and end points, it is recommended that you use a multicycle path constraint instead for designs that use an enable signal and a global clock, and where paths need to take longer than one clock cycle. See [Constraining Clock Enable Paths, on page 190](#) for a detailed explanation.

Note: This method is often used for designs that have an enable signal and a global clock, and where paths need to take longer than one clock cycle. The registers in the design are actually connected to the global clock; however, the tool treats the registers as having a virtual clock at the frequency of the enable signal.

Using this method to constrain paths for technologies with clock buffer delays requires careful analysis with the Timing Analysis Reports (STA). The virtual clock does not include clock buffer delays. However, non-virtual clocks that pass through clock buffers do include clock buffer delays. The register that generates the enable signal is on the non-virtual clock domain, whereas the registers connected to the enable signal are on the virtual clock domain. Timing analysis shows that the enable signal is on the path between the non-virtual and virtual clock domains. For the actual design, the enable signal is on a path in the non-virtual clock domain. Any paths between virtual and non-virtual clocks are reported with a clock buffer delay on the non-virtual clock. This may result in the critical path reporting negative slack.

In the following example, the path comes from a register on a non-virtual clock and goes to a register on a virtual clock.

Path information for path number 1:

```
Requested Period: 3.125
- Setup time: 0.229
= Required time: 2.896

- Propagation time: 1.448
- Clock delay at starting point: 1.857
= Slack (critical): -0.409
```

```
Number of logic level(s): 0
```

```
Starting point: SourceFlop / Q
Ending point: DestinationFlop / CE
The start point is clocked by Non-VirtualClock [rising] on pin C
The end point is clocked by VirtualClock [rising] on pin C
```

The path is reported with a negative slack of -0.49.

Timing analysis specifies a Clock delay at starting point that is the delay in the clock buffers of the non-virtual clock, but not a Clock delay at ending point. In the actual design, this delay exists at the end point. Since the clock end point is a virtual clock, the clock buffer delay creates a negative slack that does not exist in the actual design.

It is recommended that you use a multicycle path constraint instead to constrain all registers driven by the enable signal in the design.

5. After synthesis, check the Performance Summary section of the log file for a list of all the defined and inferred clocks in the design.
6. If you do not meet timing goals after place-and-route, adjust the clock constraint as follows:
 - Open the SCOPE window with the clock constraint.
 - In the Route column for the constraint, specify the actual route delay (in nanoseconds), as obtained from the place-and-route results. Adding this constraint is equivalent to putting a register delay on all the input registers for that clock.
 - Resynthesize your design.

Constraining Clock Enable Paths

You might use an enable signal as a clocking signal if you have limited clocking resources. If the enable is slower than the clock, you can ensure more accuracy by defining the enable frequency separately, instead of slowing down the clock frequency. If you slow down the clock frequency, it affects all other registers driven by the clock, and can result in longer run times as the tool tries to optimize a non-critical path.

There are two ways to define clock enables:

- By setting a multicycle path constraint to constrain all flip-flops driven by the clock enable signal (see [Defining Multicycle Paths, on page 155](#)). This is the recommended method.
- Using the `syn_reference_clock` attribute, as described in step 4 of [Defining Clock Frequency, on page 186](#). Although this method was used in earlier releases, it is not recommended any more because of changes in the way the clock start and end points are reported. An explanation of the clock start and end points reporting follows.

Clock Domains for Clock Enables Defined with `syn_reference_clock`

When you use the `syn_reference_clock` attribute to constrain an enable signal, you are telling the tool to treat the flip-flops as if they had a virtual clock at the frequency of the enable signal, when the flip-flops are actually connected to the global clock. This could result in critical paths being reported with negative slack.

The flip-flop that generates the enable signals is in the non-virtual clock domain. The flip-flops that are connected to the enable signal are in the virtual clock domain. The timing analyst considers the enable signal to be on a path that goes between a non-virtual clock domain and a virtual clock domain. In the actual circuit, the enable signal is on a path within a non-virtual clock domain. The timing analyst reports any paths between virtual and non-virtual clocks with a clock buffer delay on the non-virtual clock. This is why critical paths might be reported with negative slack.

If you use this method to constrain paths in a technology that includes clock buffer delays, you must carefully analyze the timing analysis reports. The virtual clock does not include clock buffer delays, but any non-virtual clock that passes through clock buffers will include clock buffer delays.

The following is an example report of a path from a clock enable, starting from a flip-flop on a non-virtual clock to a flip-flop on a virtual clock. The path is reported with a negative slack of -0.49.

```
Path information for path number 1:
  Requested Period: 3.125
  - Setup time: 0.229
  = Required time:2.896

  - Propagation time: 1.448
  - Clock delay at starting point:      1.857
  = Slack (critical) : -0.409

  Number of logic level(s):      0
  Starting point:SourceFlop/ Q
  Ending point:DestinationFlop / CE
```

The start point is clocked by Non-VirtualClock [rising]on pin C

The end point is clocked by VirtualClock [rising] on pin C

This timing analysis report includes a Clock delay at starting point, but does not include Clock delay at ending point. The clock delay at the starting point is the delay in the clock buffers of the non-virtual clock. In the actual circuit, this delay would also be at the ending point and not affect the calculation of slack. However as the ending clock is a virtual clock, the clock buffer delay ends up creating a negative slack that does not exist in the actual circuit.

This report is a result of defining the clock enables with the `syn_reference_clock` attribute. This is why it is recommended that you use multicyle paths to constrain all the flip-flops driven by the enable signal.

Defining Other Clock Requirements

Besides clock frequency (described in [Defining Clock Frequency, on page 186](#)), you can also set other clock requirements, as follows:

- If you have limited clock resources, define clocks that do not need a clock buffer by attaching the `syn_noclockbuf` attribute to an individual port, or the entire module/architecture.
- Define the relationship between clocks by setting clock domains. By default, each clock is in a separate clock group named `default_clkgroup<n>` with a sequential number suffix.

- On the SCOPE Clocks tab, group related clocks by putting them into the same clock group. Use the Clock Group field to assign all related clocks to the same clock group.
- Make sure that unrelated clocks are in different clock groups. If you do not, the software calculates timing paths between unrelated clocks in the same clock group, instead of treating them as false paths.
- Input and output ports that belong to the System clock domain are considered a part of every clock group and will be timed. See [Defining Input and Output Constraints \(Legacy\)](#), on page 195 for more information.

The software does not check design rules, so it is best to define the relationship between clocks as completely as possible.

- Define all gated clocks with the `define_clock` constraint.

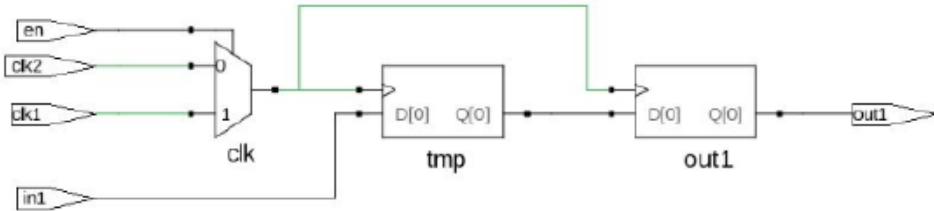
Avoid using gated clocks to eliminate clock skew. If possible, move the logic to the data pin instead of using gated clocks. If you do use gated clocks, you must define them explicitly, because the software does not propagate the frequency of clock ports to gated clocks.

To define a gated clock, attach the `define_clock` constraint to the clock source, as described above for internal clocks. To attach the constraint to a keepbuf (a keepbuf is a placeholder instance for clocks generated from combinational logic), do the following:

- Attach the `syn_keep` attribute to the gated clock to ensure that it retains the same name through changes to the RTL code.
 - Attach the `define_clock` constraint to the net or pin connected to the keepbuf instance generated for the gated clock.
- Specify edge-to-edge clock delays on the Clock to Clock tab (`define_clock_delay`).
 - After synthesis, check the Performance Summary section of the log file for a list of all the defined and inferred clocks in the design.

Defining Clocks for Clock Muxes

The definition of clocks that are to be muxed together varies slightly, depending on whether the clocks have the same frequency or not. The following procedures use this example as an illustration:



Defining Muxed Clocks with Different Frequencies

If the clocks are asynchronous, separate clock paths must be defined, as described below.

1. Define the clocks with `create_clock` constraints.

For the example, two clocks are defined:

```
create_clock -name {clk1} [get_nets {clk1}] -period 10.0
  -waveform {0 5.0}
create_clock -name {clk2} [get_nets {clk2}] -period 10.0 -waveform {0 5.0}
```

2. Use multiple `set_clock_groups` constraints to mark them as asynchronous to each other:

```
set_clock_groups -derive -asynchronous -name {default_clkgroup_0} -group
  [get_clocks {clk1}]
set_clock_groups -derive -asynchronous -name {default_clkgroup_1} -group
  [get_clocks {clk2}]
```

3. Check the timing report.

For the example, the tool reports two separate clock paths, one for each clock.

Defining Muxed Clocks with the Same Frequency

If the clocks have the same phase and frequency, follow this procedure to define the clocks.

1. Define the clock at the net connected to the output pin of the mux.

For example:

```
create_clock -name {clk} [get_nets {clk}] -period 10.0  
-waveform {0 5.0}
```

2. Define the mux output clock as asynchronous to all other clocks, using a `set_clock_groups` constraint:

```
set_clock_groups -derive -asynchronous -name {default_clkgroup_2} -group  
[get_clocks {clk}]
```

3. Check the timing report.

In this case, there should be a single clock path, instead of separate paths.

Defining Input and Output Constraints (Legacy)

In addition to setting I/O delays in the SCOPE window as described in [Setting Clock and Path Constraints, on page 182](#), you can also set the Use clock period for unconstrained IO option.

- Open the SCOPE window, click Inputs/Outputs, and select the port (Port). You can set the constraint for
 - All inputs and outputs (globally in the top-level netlist)
 - For a whole bus
 - For single bits

You can specify multiple constraints for the same port. The software applies all the constraints; the tightest constraint determines the worst slack. If there are multiple constraints from different levels, the most specific overrides the more global. For example, if there are two bit constraints and two port constraints, the two bit constraints override the two port constraints for that bit. The other bits get the two port constraints.

- Specify the constraint value in the SCOPE window:
 - Select the type of delay: input or output (Type).
 - Type a delay value (Value).
 - Check the Enabled box, and save the constraint file in the project.

Make sure to specify explicit constraints for each I/O path you want to constrain.

- To determine how the I/O constraints are used during synthesis, do the following:
 - Select Project->Implementation Options, and click Constraints.
 - To use only the explicitly defined constraints disable Use clock period for unconstrained IO.
 - To synthesize with all the constraints, using the clock period for all I/O paths that do not have an explicit constraint enable Use clock period for unconstrained IO.
 - Synthesize the design. When you forward-annotate the constraints, the constraints used for synthesis are forward-annotated for place-and-route.

- Input or output ports with explicitly defined constraints, but without a reference clock (-ref option) are included in the System clock domain and are considered to belong to every defined or inferred clock group.
- If you do not meet timing goals after place-and-route and you need to adjust the input constraints; do the following:
 - Open the SCOPE window with the input constraint.
 - In the Route column for the input constraint, specify the actual route delay in nanoseconds, as obtained from the place-and-route results. Adding this constraint is equivalent to putting a register delay on the input register.
 - Resynthesize your design.

Defining False Paths (Legacy)

You define false paths by setting constraints explicitly on the Delay Paths tab or implicitly on the Clock and Clock to Clock tabs. See [Defining From/To/Through Points for Timing Exceptions, on page 151](#) for object naming and specifying through points.

- To define a false path between ports or registers, select the SCOPE Delay Paths tab, and do the following:
 - From the Delay Type pull-down menu, select False.
 - Use the pull-down to select the port or register from the appropriate column (From/To/Through).
 - Check the Enabled box.

The software treats this as an explicit false constraint and assigns it the highest priority. Any other constraints on this path are ignored.

- To define a false path between two clocks, select the SCOPE Clocks tab, and assign the clocks to different clock groups:

The software implicitly assumes a false path between clocks in different clock groups. This false path constraint can be overridden by a maximum path delay constraint, or with an explicit constraint.

- To define a false path between two clock edges, select the SCOPE Clock to Clock tab, and do the following:
 - Specify one clock as the starting clock edge (From Clock Edge).
 - Specify the other clock as the ending clock edge (To Clock Edge).

- Click in the Delay column, and select false.
- Mark the Enabled check box.

Use this technique to specify a false path between any two clocks, regardless of clock groups. This constraint can be overridden by a maximum delay constraint on the same path

- To override an implicit false path between any two clocks described previously, set an explicit constraint between the clocks by selecting the SCOPE Clock to Clock tab, and doing the following:
 - Specify the starting (From Clock Edge) and ending clock edges (To Clock Edge).
 - Specify a value in the Delay column.
 - Mark the Enabled check box.

The software treats this as an explicit constraint. You can use this method to constrain a path between any two clocks, regardless of whether they belong to the same clock group.

- To set an implicit false path on a path to or from an I/O port, do the following:
 - Select Project->Implementation Options->Constraints.
 - Disable Use clock period for unconstrained IO.

Generating Constraint Files for Technologies Supported in Radiant Software

The tool automatically generates vendor-specific constraint files that you can use for forward-annotation. The synthesis constraints are mapped to the appropriate vendor constraints. You can control this process with the attributes given below.

The software converts the following synthesis and global frequency constraints into the corresponding commands in the *.ldc file for technologies supported in the Radiant software.

<code>create_clock</code>	<code>create_generated_clock</code>
<code>set_clock_groups</code>	<code>set_clock_latency</code>
<code>set_clock_uncertainty</code>	<code>set_input_delay</code>
<code>set_output_delay</code>	<code>set_max_delay</code>
<code>set_multicycle_path</code>	<code>set_false_path</code>

CHAPTER 6

Synthesizing and Analyzing the Results

This chapter describes how to run synthesis, and how to analyze the log file generated after synthesis. See the following:

- [Synthesizing Your Design](#), on page 198
- [Checking Log File Results](#), on page 203
- [Handling Messages](#), on page 218

Synthesizing Your Design

Once you have set your constraints, options, and attributes, running synthesis is a simple one-click operation. See the following:

- [Running Logic Synthesis](#)
- [Using Up-to-date Checking for Job Management](#)

Running Logic Synthesis

When you run logic synthesis, the tool compiles the design and then maps it to the technology target you selected.

1. If you want to compile your design without mapping it, select Run-> Compile Only or press F7.

A compiled design has the RTL mapping, and you can view the RTL view. You might want to just compile the design when you are not ready to synthesize the design, but when you need to use a tool that requires a compiled design, like the SCOPE interface.

2. To synthesize the logic, set all the options and attributes you want, and then click Run.

Using Up-to-date Checking for Job Management

Synthesis is becoming more complex and consists of running many jobs. Often, part or all of the job flow is already up-to-date and rerunning the job may not be necessary. For large designs that may take hours to run, up-to-date checking can reduce the time for rerunning jobs.

Up-to-date checking is run for all synthesis design flows. However, for the Hierarchical Project Management flows, up-to-date checking is an essential feature. For example, if a project contains four sub-projects and only one project is modified, then the other three projects do not need to be rerun. This saves in overall runtime.

Up-to-date checking includes the following:

- The GUI launches mapper modules (pre-mapping and technology mapping) and saves the intermediate netlists and log files in the synwork and synlog folders, respectively.

- After each individual module run completes, the GUI optionally copies the contents of these intermediate log files from the synlog folder and adds them to the Project log file (rev_1/*projectName.srr*). To set this option, see [Copy Individual Job Logs to the SRR Log File, on page 201](#).
- If you re-synthesize the design and there are no changes to the inputs (HDL, constraints, and Project options):
 - The GUI does not rerun pre-mapping and technology mapping and no new netlist files are created.
 - In the HTML log file, the GUI adds a link that points to the existing pre-mapping and mapping log files from the previous run. Double-click on this link (@L: indicates the link) to open the new text file window.

If you open the text log file, the link is a relative path to the implementation folder for the pre-mapping and mapping log files from the previous run.

Note: Also, the GUI adds a note that indicates mapping will not be re-run and to use the Run->Resynthesize All option in the Project view to force synthesis to be run again.

```
#####]

Synopsys Pre-mapping Report                                ersion map610dev, Build
@N: : | premapping output is up to date. No run necessary. 11 Rights Reserved
To force a re-synthesis, select [Resynthesize All] in menu [Run]. d\gated_clocks\test.sdc

Log file from previous run:
@L:top_premapping.srr                                     le <C:\builds\syn201103_
#####]                                                  table from file <C:\buil

@W:"c:\syn_hier\fixed\gated_clocks\myreg.v":18:14:18:22|Net U_reg
@W:"c:\syn_hier\fixed\gated_clocks\top.v":17:14:17:24|Net U_gt_re

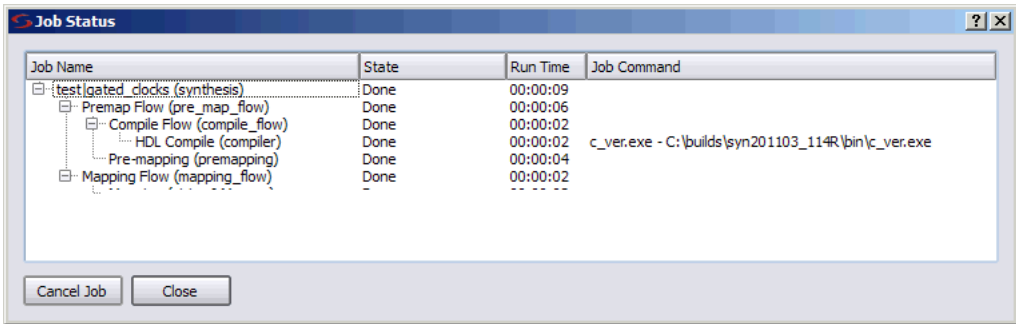
Finished Pre Mapping Phase. (Time elapsed 0h:00m:00s; Memory used
@N: BN225 |Writing default property annotation file C:\syn_hier\f
Pre Mapping successful!
Process took 0h:00m:01s realtime, 0h:00m:01s cputime
# Mon Jan 24 18:39:06 2011

#####]
```



As the job is running, you can click in the job status field of the Project view to bring up the Job Status display. When you rerun synthesis, the job status identifies which modules (pre-mapping or mapping) are up-to-date.

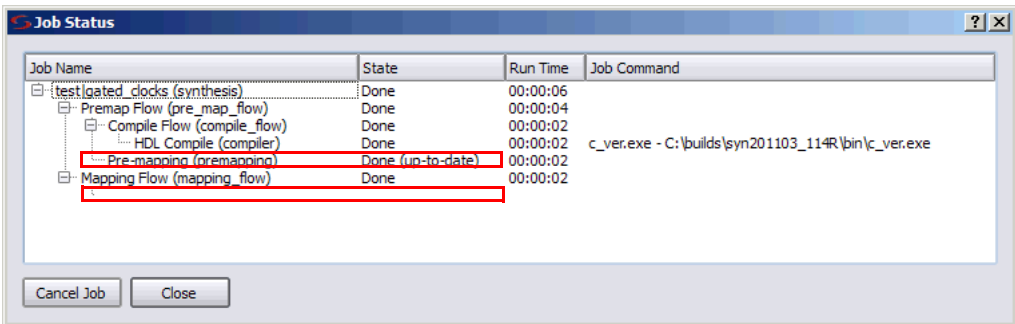
Job Status for Synthesis Run



Job Name	State	Run Time	Job Command
test_lgated_clocks (synthesis)	Done	00:00:09	
Premap Flow (pre_map_flow)	Done	00:00:06	
Compile Flow (compile_flow)	Done	00:00:02	
HDL Compile (compiler)	Done	00:00:02	c_ver.exe - C:\builds\syn201103_114R\bin\c_ver.exe
Pre-mapping (premapping)	Done	00:00:04	
Mapping Flow (mapping_flow)	Done	00:00:02	

Buttons: Cancel Job, Close

Job Status for Re-synthesis Run



Job Name	State	Run Time	Job Command
test_lgated_clocks (synthesis)	Done	00:00:06	
Premap Flow (pre_map_flow)	Done	00:00:04	
Compile Flow (compile_flow)	Done	00:00:02	
HDL Compile (compiler)	Done	00:00:02	c_ver.exe - C:\builds\syn201103_114R\bin\c_ver.exe
Pre-mapping (premapping)	Done (up-to-date)	00:00:02	
Mapping Flow (mapping_flow)	Done	00:00:02	

Buttons: Cancel Job, Close

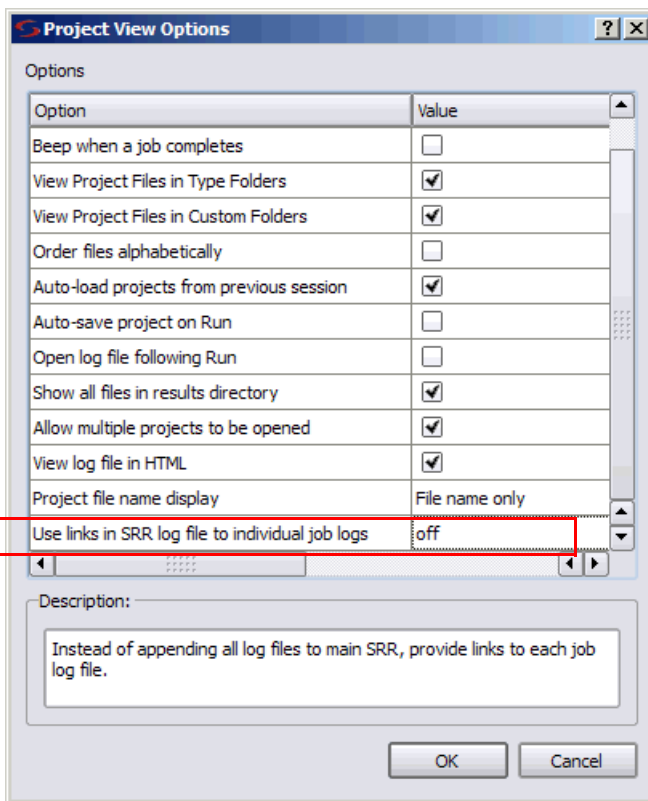
See also:

- [Copy Individual Job Logs to the SRR Log File](#)
- [Limitations and Risks](#)

Copy Individual Job Logs to the SRR Log File

By default, up-to-date checking uses links in the log file (srr) to individual job logs. To change this option so that individual job logs are always appended to the main log file (srr), do the following:

1. Select Options->Project View Options from the Project menu.
2. On the Project View Options dialog box, scroll down to the Use links in SRR log file to individual job logs option.
3. Use the pull-down menu, and select off.



Limitations and Risks

Up-to-date checking limitations and risks include the following:

- Compiler up-to-date checks are done internally by the compiler and with no changes to the compiler reporting structure.
- GUI up-to-date checks use timestamp information of its input files to decide when mapping is re-run. Be aware that:
 - The GUI uses netlist files (`vm` and `srd`) from the `synwork` folder for timestamp checks. If you delete a `vm` file from the implementation folder, this does not trigger compiler or mapper re-runs. You must delete netlist files from the `synwork` folder instead.
 - The copy command behaves differently on Windows and Linux. On Windows, the timestamp does not change if you copy a file from one directory to another. But on Linux (and MKS shell), the timestamp information gets changed.
- When running a design, the up-to-date checking feature automatically determines if the design needs to be re-synthesized. However, when you modify constraints in a Tcl file sourced within the constraints file, the software is not aware of these changes and does not force the design to be re-synthesized.

Checking Log File Results

You can check the log file for information about the synthesis run. In addition, the user interface has a Tcl Script window, that echoes each command as it is run. The following describe different ways to check the results of your run:

- [Viewing and Working with the Log File](#), on page 203
- [Accessing Specific Reports Quickly](#), on page 207
- [Accessing Results Remotely](#), on page 209
- [Analyzing Results Using the Log File Reports](#), on page 213
- [Using the Watch Window](#), on page 214
- [Checking Resource Usage](#), on page 215
- [Querying Metrics for a Design](#), on page 216

Viewing and Working with the Log File

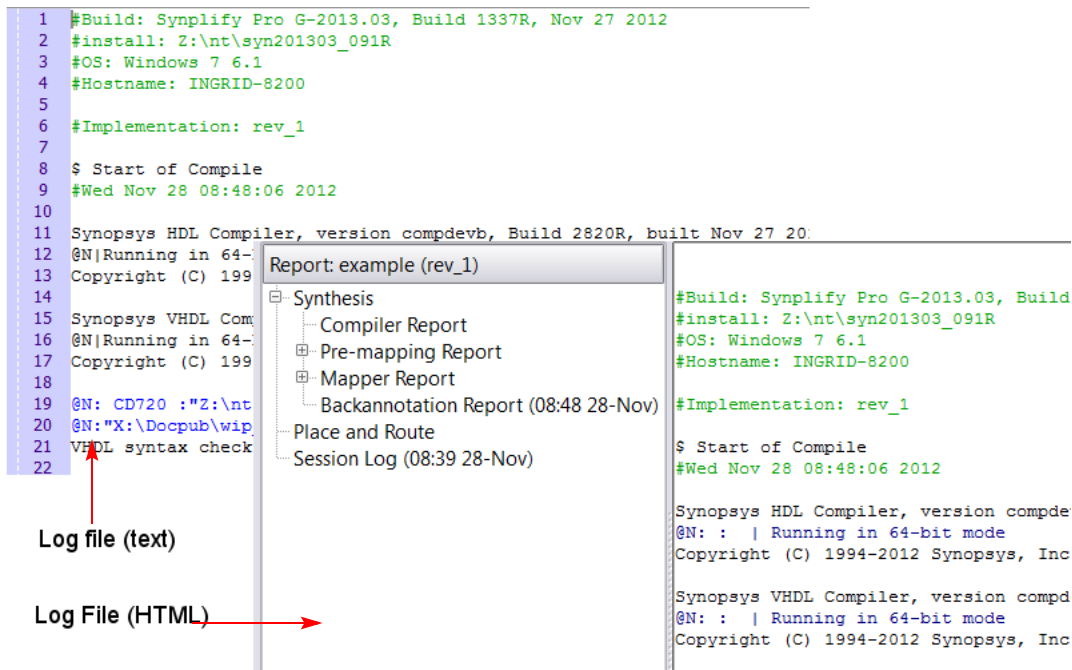
The log file contains the most comprehensive results and information about a synthesis run. The default log file is in HTML format, but there is a text version available too.

For users who only want to check a few critical performance criteria, it is easier to use the Watch Window (see [Using the Watch Window, on page 214](#)) instead of the log file. For details, read through the log file.

1. To open the log file, use one of these listed methods, according to the format you want:

HTML	• Select View->Log File. • Click the View Log button in the Project window. • Double-click the <i>designName</i>.htm file in the Implementation Results view.
Text	Double-click the <i>designName</i> .srr file in the Implementation Results view. To set the text file version to open by default instead of the HTML version, select Options->Project View Options, and toggle off the View log file in HTML option.

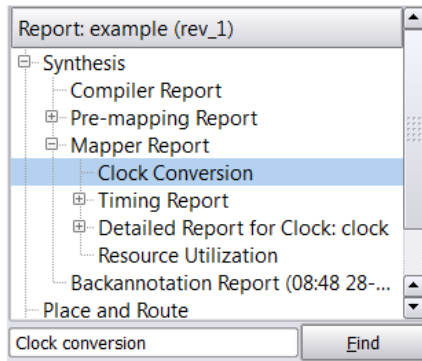
The log file lists the compiled files, details of the synthesis run, and includes color-coded errors, warnings and notes, and a number of reports. For information about the reports, see [Analyzing Results Using the Log File Reports](#), on page 213.



2. Navigate the log file to view specific pieces of information.

For quicker access to specific log information, use alternative access methods, described in [Accessing Specific Reports Quickly](#), on page 207 instead of the ones described here.

- Use the panel on the left of the HTML log file to navigate to the section you want. You can use the Find button and the search field at the bottom of this panel to search the headings.



- To search the body of the log file, use Control-f or the Edit->Find command. See [Viewing and Working with the Log File, on page 203](#) for details.
- To add bookmarks or for general information about working in an editing window, see [Editing HDL Source Files with the Built-in Text Editor, on page 32](#).

The areas of the log file that are most important are the warning messages and the timing report. The log file includes a timing report that lists the most critical paths. The synthesis products also let you generate a report for a path between any two designated points, see [Generating Custom Timing Reports with STA, on page 367](#). The following table lists places in the log file you can use when searching for information.

To find...	Search for...
Notes	@N or look for blue text
Warnings and errors	@W and @E, or look for purple and red text respectively
Performance summary	Performance Summary
The beginning of the timing report	START TIMING REPORT

To find...	Search for...
Detailed information about slack times, constraints, arrival times, etc.	Interface Information
Resource usage	Resource Usage Report. See Checking Resource Usage , on page 215.
Gated clock conversions	Gated clock report

3. Resolve any errors and check all warnings.

You must fix errors, because you cannot synthesize a design with errors. Check the warnings and make sure you understand them. See [Checking Results in the Message Viewer, on page 218](#) for information. Notes are informational and usually can be ignored. For details about crossprobing and fixing errors, see [Working with Downgradable Errors and Critical Warnings, on page 228](#), [Editing HDL Source Files with the Built-in Text Editor, on page 32](#), and [Crossprobing from the Text Editor Window, on page 331](#).

If you see Automatic dissolve at startup messages, you can usually ignore them. They indicate that the mapper has optimized away hierarchy because there were only a few instances at the lower level.

4. If you are trying to find and resolve warnings, you can bookmark them as shown in this procedure:
 - Select Edit->Find or press Ctrl-f.
 - Type @W as the criteria on the Find form and click Mark All. The software inserts bookmarks at every line with a warning. You can now page through the file from bookmark to bookmark using the commands in the Edit menu or the icons in the Edit toolbar. For more information on using bookmarks, see [Editing HDL Source Files with the Built-in Text Editor, on page 32](#).
5. To crossprobe from the log file to the source code, click on the file name in the HTML log file or double-click on the warning text (not the ID code) in the ASCII text log file.

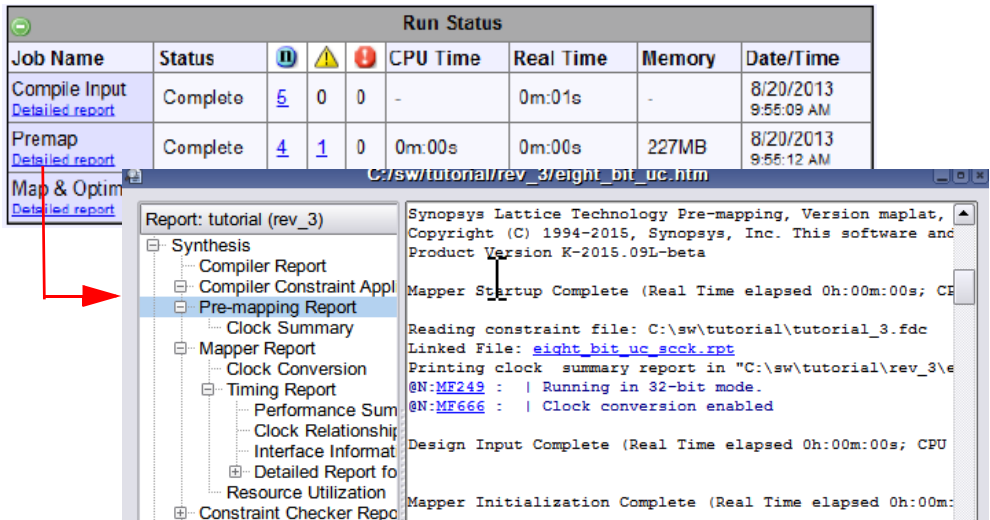
Accessing Specific Reports Quickly

The log file contains all the results from the synthesis run, but you might want to hone in on specific information. Instead of browsing the log file to find the information you need, you can use the techniques described below:

1. To quickly view specific pieces of log information, go to the Project Status window and click the appropriate links to display the corresponding reports or specific parts of the log file.

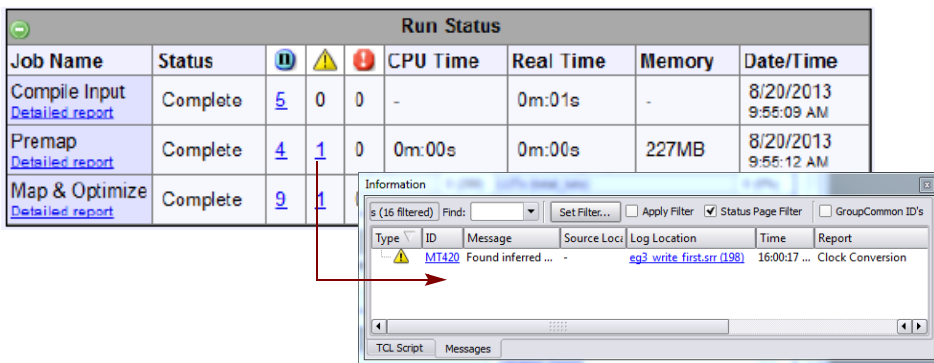
Timing reports	Click Detailed Report or Timing Report View in the Timing Summary panel.
Log at different stages	Click Detailed Report in the Run Status panel.
Area reports	Click Detailed Report or Hierarchical Area Report in the Area Summary panel.
High reliability reports	Click Detailed Report in the High Reliability Report panel.
Optimizations	Click Detailed Report in the Optimizations Summary panel.

The Detailed Report links display parts of the log file, and the other links go to special view windows for different kinds of reports. See [The Project Results View, on page 25](#) for more information about different reports that can be accessed from the Project Results view.



2. To view timing information, use one of these methods:
 - Set important timing parameters to monitor in the Watch window, like slack and frequency. See [Using the Watch Window, on page 214](#) for details.
 - Click View Log in the Project view and navigate to the appropriate section in the log file.
3. To view messages, use any of the following methods
 - From the Run Status panel in the Project Status window, click the link that lists the number of errors, warnings, or notes at different design stages. The Message window opens. Click the message ID to get more information about the error and how to fix it.

This is the quickest method to narrow down the list of messages and access the one you want.



The numbers of notes, errors, and warnings reported in the Run Status panel might not match the numbers displayed in the Messages window if the design contains compile points. The numbers reported are for the top level.

- Click the Messages tab at the bottom of the Project view to open a window with a list of all the notes, errors and warnings. See [Checking Results in the Message Viewer, on page 218](#) for more information about using this window.

Type	ID	Message	Source Location	Log Location	Time	Report
Warning	CD233	Using sequential encoding for type aluop_type	const_plq.vhd	spcl_reqs.srr	07:37:47 Wed May 09	Vhdl Compiler
Warning	CD630	Synthesizing work.reg_file.first	-	spcl_reqs.srr	07:37:47 Wed May 09	Vhdl Compiler
Warning	FX271	Instance "DECODE.ALUOP[3]" with 30 loads has been...	-	spcl_reqs.srr	07:37:47 Wed May 09	SPARTAN3 Mapper
Warning	FX271	Instance "SPECIAL_REGS.INST[9]" with 19 loads has been...	spcl_reqs.vhd (44)	spcl_reqs.srr (194)	07:37:47 Wed May 09	SPARTAN3 Mapper
Warning	FX271	Instance "SPECIAL_REGS.INST[11]" with 30 loads has been...	spcl_reqs.vhd (44)	spcl_reqs.srr (167)	07:37:47 Wed May 09	SPARTAN3 Mapper
Warning	FX271	Instance "DECODE.ALUOP[1]" with 15 loads has been...	ins_decode.vhd (296)	spcl_reqs.srr (193)	07:37:47 Wed May 09	SPARTAN3 Mapper
Warning	FX271	Instance "DECODE.ALUOP[3]" with 30 loads has been...	ins_decode.vhd (296)	spcl_reqs.srr (168)	07:37:47 Wed May 09	SPARTAN3 Mapper
Warning	FX271	Instance "DECODE.ALUOP[0]" with 36 loads has been...	ins_decode.vhd (296)	spcl_reqs.srr (166)	07:37:47 Wed May 09	SPARTAN3 Mapper
Warning	CD720	Setting time resolution to ns	std.vhd (123)	spcl_reqs.srr (16)	07:37:47 Wed May 09	Vhdl Compiler
Warning	FX107	No read/write conflict check. Simulation mismatch pos...	reg_file.vhd (23)	spcl_reqs.srr (112)	07:37:47 Wed May 09	Mapper Report
Warning	CL134	Found RAM mem, depth=32, width=8	reg_file.vhd (23)	spcl_reqs.srr (80)	07:37:47 Wed May 09	Vhdl Compiler
Warning	CL201	Trying to extract state machine for register STACKLEV...	pc.vhd (34)	spcl_reqs.srr (82)	07:37:47 Wed May 09	Vhdl Compiler
Warning	FX214	Generating ROM ROM.Data_1[11:0]	ins_rom.vhd (22)	spcl_reqs.srr (145)	07:37:47 Wed May 09	Mapper Report
Warning	MT206	Autoconstrain Mode is ON	-	spcl_reqs.srr (108)	07:37:47 Wed May 09	Mapper Report
Warning	MT197	Clock constraints cover only FF-to-FF paths associate...	-	spcl_reqs.srr (246)	07:37:47 Wed May 09	Timing Report

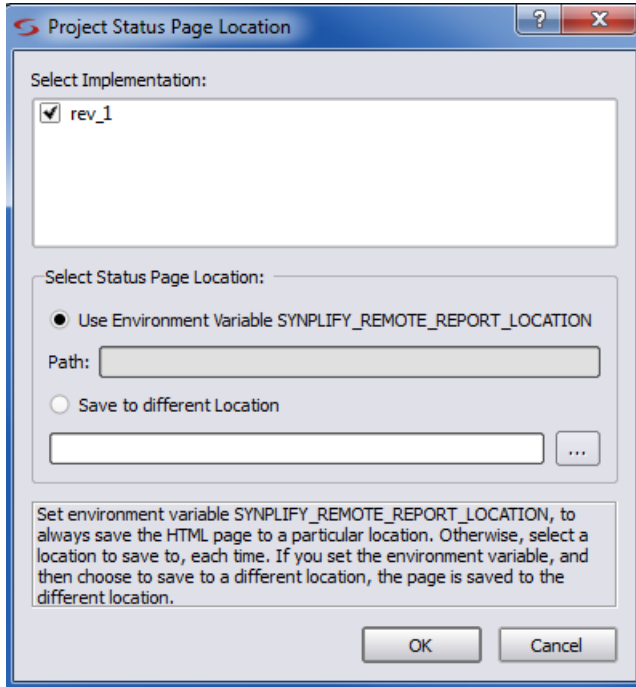
- Open the log file, locate the message, and click the message ID. The log file includes all the results from the run, so it could be harder to locate the message you want.

Accessing Results Remotely

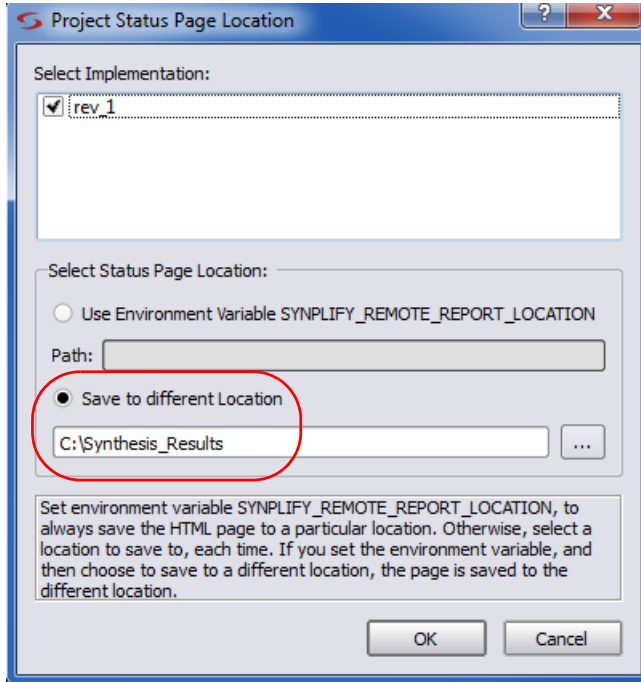
You can access the log file results remotely from various mobile devices. For example, you can use this feature to run synthesis for jobs with long run times and then check the results of the synthesis run later from anywhere. The Project Status report files can be accessed from any browser without bringing up the synthesis tool.

To access the log file remotely, do the following:

1. Select Options->Project Status Page Location from the Project menu and select the implementation for which you want the reports.



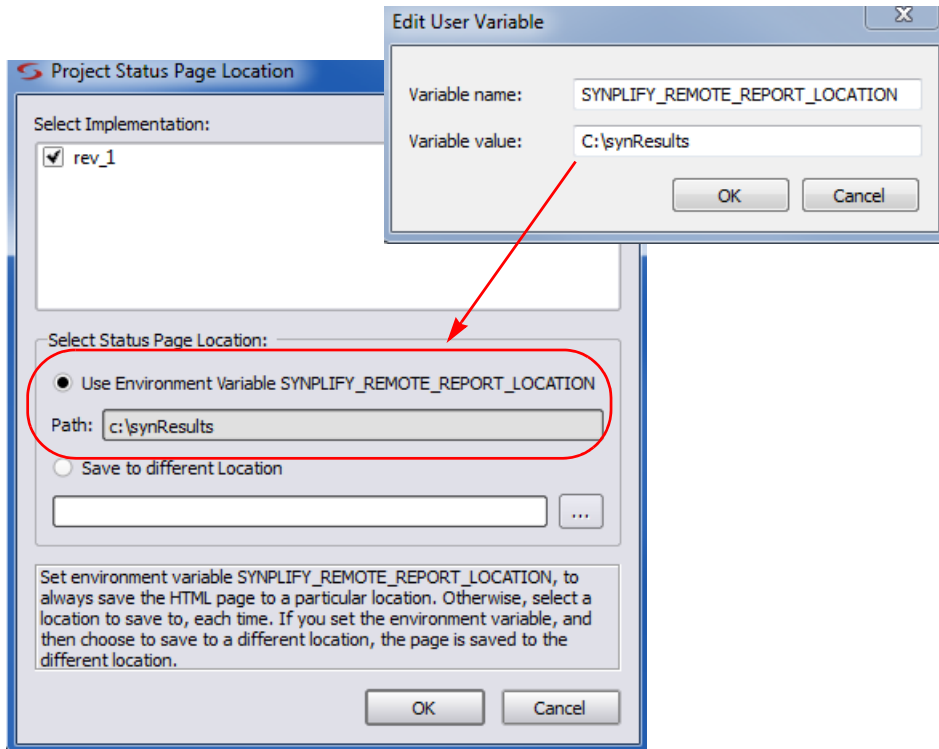
2. Set the location for storing the project status page, using either of these methods:
 - Enable Save to different location and specify a path for the location of the status page. This allows you to save the status reports in different locations.



- Use an environment variable by enabling Use Environment Variable SYNPLIFY_REMOTE_REPORT_LOCATION.

Windows	Enable Use Environment Variable SYNPLIFY_REMOTE_REPORT_LOCATION. Specify the variable name SYNPLIFY_REMOTE_REPORT_LOCATION and the location you want from the Control Panel on the Edit User Variable dialog box.
Linux	Specify <code>setenv SYNPLIFY_REMOTE_REPORT_LOCATION <i>pathLocation</i></code> in the <code>.cshrc</code> file. Enable Use Environment Variable SYNPLIFY_REMOTE_REPORT_LOCATION.

If you use this option, you must restart the tool the first time, since the environment variable is not applied dynamically. This option always saves the status report to the location indicated by the variable.



3. Click OK.
4. Run synthesis.

The status reports are saved to the location you specified for your project. For example:

C:\synResults\tutorial\rev_1

5. Access the location you set up from any browser on a mobile device (for example, a smart phone or tablet).
 - Access the location you set in the previous steps.
 - Open the *projectName/implementationName/index.html* file with any browser.

Your company may need to set up a location on its internal internet, where the status reports can be saved and later accessed with a URL address.

Analyzing Results Using the Log File Reports

The log file contains technology-appropriate reports like timing reports, resource usage reports, and net buffering reports, in addition to any notes, errors, and warning messages.

1. To analyze timing results, do the following:

- View the Timing Report (Performance Summary section of the log file) and check the slack times. See [Handling Negative Slack, on page 366](#) for details.
- Check the detailed information for the critical paths, including the setup requirements at the end of the detailed critical path description. You can crossprobe and view the information graphically and determine how to improve the timing.
- In the HTML log file, click the link to open up the HDL Analyst view for the path with the worst slack.

To generate timing information for a path between any two designated points, see [Generating Custom Timing Reports with STA, on page 367](#).

2. To check buffers, do the following:

- Check the report by going to the Net Buffering Report section of the log file.
- Check the number of buffers or registers added or replicated and determine whether this fits into your design optimization strategy.

3. To check logic resources, check the Resource Usage Report section at the end of the log file, as described in [Checking Resource Usage, on page 215](#).

Using the Watch Window

The Watch window provides a more convenient viewing mechanism than the log file for quickly checking key performance criteria or comparing results from different runs. Its limitation is that it only displays certain criteria. If you need details, use the log file, as described in [Viewing and Working with the Log File, on page 203](#).

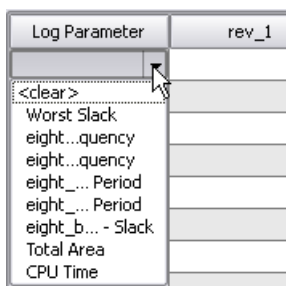
1. Open the Watch window, if needed, by checking View->Watch Window.

If you open an existing project, the Watch window shows the parameters set the last time you opened the window.

2. If you need a larger window, either resize the window or move the Watch Window as described below.
 - Hold down Ctrl or Shift, click on the window, and move it to a position you want. This makes the Watch window an independent window, separate from the Project view.
 - To move the window to another position within the Project view, right-click in the window border and select Float in Main Window. Then move the window to the position you want, as described above.

See [Watch Window, on page 34](#) in the *Reference Manual* for information about the popup menu commands.

3. Select the log parameter you want to monitor by clicking on a line and selecting a parameter from the resulting popup menu.



The software automatically fills in the appropriate value from the last synthesis run. You can check the clock requested and estimated frequencies, the clock requested and estimated periods, the slack, and some resource usage criteria.

4. To compare the results of two or more synthesis runs, do the following:
 - If needed, resize or move the window as described above.
 - Click the right mouse button in the window and select **Configure Watch** from the popup.
 - Click **Watch Selected Implementations** and either check the implementations you want to compare or click **Watch All Implementations**. Click **OK**. The Watch window now shows a column for each implementation you selected.
 - In the Watch window, set the parameters you want to compare.

The software shows the values for the selected implementations side by side. For more information about multiple implementations, see [Tips for Optimization](#), on page 414.

Log Parameter	rev_1	rev_2	rev_3
Worst Slack	989.029	995.811	997.109
system clk_inferred_clock - Requested Frequency	1.0 MHz	1.0 MHz	1.0 MHz
system clk_inferred_clock - Estimated Frequency	91.1 MHz	238.7 MHz	345.9 MHz

Checking Resource Usage

Each FPGA architecture has a certain number of dedicated FPGA resources. Use the Resource Usage section of the log file to check whether you are exceeding the available resources.

1. Go to the Resource Usage report at the end of the log file (`srr`).
2. Check the number and types of components used to determine if you have used too much of your resources.

If your design is over-utilized, you can manage usage with resource-specific attributes like `syn_ramstyle`, `syn_dspstyle`, and so on. For hierarchical designs you can set limits with attributes like `syn_allowed_resources` or the **Allocate Timing and Resource Budgets** command.

Querying Metrics for a Design

Keeping track of metrics is important for measuring and tuning the QoR of a design. Metrics include data from various steps in the design flow; the data is saved and can be retrieved anytime. The design metrics you can query include the number of LUTs, the runtime for each process step, the worst slack, the slack for specific clocks, and the number of unconverted gated clocks

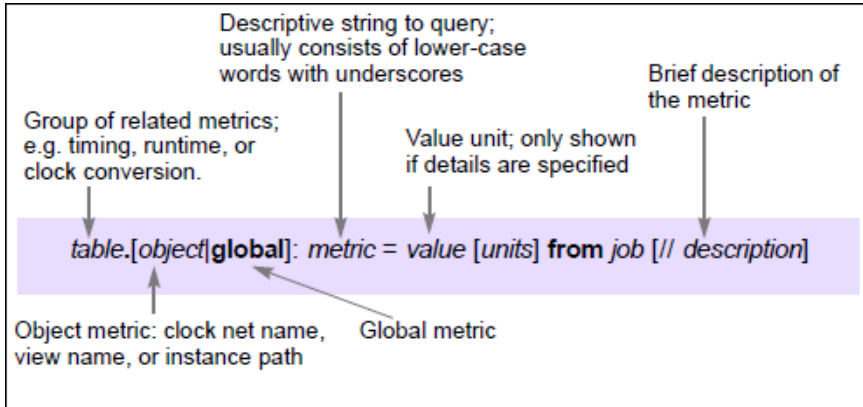
1. Start from the Tcl window and make sure you are located in the current implementation directory.
2. To find the names of the metrics available for the design, use one of the following command:

Use the Command...	To...
<code>dump_metrics</code>	Show metrics and values available for the current implementation of a design.
<code>query_available_metrics</code>	Show metrics that can be queried for the design. You should use this command primarily for scripting, since it returns a Tcl list.

3. Use the metric names with one of the following commands, according to the level of detail you want to see.

Use the Command...	To...
<code>query_metric</code>	Query specific QoR metrics. For example: % query_metric runtime.realtime -jobname compiler 3.154000
<code>query_metric_details</code>	Query information about a QoR metric. For example: % query_metric_details clock_conversion.clean_clock_pins -jobname fpga_mapper 271 {} {Number of clock pins driven by non-gated/non-generated clock trees}

Metrics can be global for the entire design, or specific to an object, such as a clock, module, or net. The command returns values in the default output format shown below:



For example, clock conversion metrics can be specified as follows:

Table	Metric Name	Description
clock_conversion	clean_clock_trees	Number of non-gated/non-generated clock trees
clock_conversion	clean_clock_pins	Number of clock pins driven by non-gated/non-generated clock trees
clock_conversion	gated_clock_trees	Number of gated/generated clock trees
clock_conversion	instances_converted	Number of sequential instances converted
clock_conversion	instances_notconverted	Number of sequential instances left unconverted

The following is an example of reported metrics:

```
clock_conversion.global: instances_converted = 0 from fpga_mapper
//Number of sequential instances converted

runtime.global: realtime = 4.160000 seconds from compiler
runtime.global: cputime = 3.073220 seconds from compiler
```

Handling Messages

This section describes how to work with the error messages, notes, and warnings that result after a run. See the following for details:

- [Checking Results in the Message Viewer](#), on page 218
- [Filtering Messages in the Message Viewer](#), on page 220
- [Filtering Messages from the Command Line](#), on page 222
- [Automating Message Filtering with a Tcl Script](#), on page 223
- [Log File Message Controls](#), on page 225
- [Working with Downgradable Errors and Critical Warnings](#), on page 228

Checking Results in the Message Viewer

The Tcl Script window includes a Message Viewer. By default, the Tcl window is in the lower left corner of the main window. This procedure shows you how to check results in the message viewer.

1. If you need a larger window, either resize the window or move the Tcl window. Click in the window border and move it to a position you want. You can float it outside the main window or move it to another position within the main window.
2. Click the Messages tab to open the message viewer.

The window lists the errors, warnings, and notes in a spreadsheet format. See [Message Viewer, on page 38](#) in the *Reference Manual* for a full description of the window.

1 warning, 37 notes

Find: Set Filter... ☐ Apply Filter ☒ Group Common

Type	ID	Message	Source Location	Log Location	Time	Report
U	9	CD233 Using sequential encoding for type aluop_type	const_pkq.vhd	spcl_reqs.srr	07:37:47 Wed May 09	Vhdl Compiler
U	9	CD630 Synthesizing work.reg_file.first	-	spcl_reqs.srr	07:37:47 Wed May 09	Vhdl Compiler
U	8	FX271 Instance "DECODE.ALUOP[3]" with 30 loads has been...	-	spcl_reqs.srr	07:37:47 Wed May 09	SPARTAN3 Mapper
U	[4]	FX271 Instance "SPECIAL_REGS.INST[9]" with 19 loads has...	spcl_reqs.vhd (44)	spcl_reqs.srr (194)	07:37:47 Wed May 09	SPARTAN3 Mapper
U	FX271	Instance "SPECIAL_REGS.INST[11]" with 30 loads ha...	spcl_reqs.vhd (44)	spcl_reqs.srr (167)	07:37:47 Wed May 09	SPARTAN3 Mapper
U	FX271	Instance "DECODE.ALUOP[1]" with 15 loads has been...	ins_decode.vhd (296)	spcl_reqs.srr (193)	07:37:47 Wed May 09	SPARTAN3 Mapper
U	FX271	Instance "DECODE.ALUOP[3]" with 30 loads has been...	ins_decode.vhd (296)	spcl_reqs.srr (168)	07:37:47 Wed May 09	SPARTAN3 Mapper
U	FX271	Instance "DECODE.ALUOP[0]" with 36 loads has been...	ins_decode.vhd (296)	spcl_reqs.srr (166)	07:37:47 Wed May 09	SPARTAN3 Mapper
U	CD720	Setting time resolution to ns	std.vhd (123)	spcl_reqs.srr (16)	07:37:47 Wed May 09	Vhdl Compiler
U	FX107	No read/write conflict check. Simulation mismatch pos...	reg_file.vhd (23)	spcl_reqs.srr (112)	07:37:47 Wed May 09	Mapper Report
U	CL134	Found RAM mem, depth=32, width=8	reg_file.vhd (23)	spcl_reqs.srr (80)	07:37:47 Wed May 09	Vhdl Compiler
U	CL201	Trying to extract state machine for register STACKLEV...	pc.vhd (34)	spcl_reqs.srr (82)	07:37:47 Wed May 09	Vhdl Compiler
U	FX214	Generating ROM ROM.Data_I[11:0]	ins_rom.vhd (22)	spcl_reqs.srr (145)	07:37:47 Wed May 09	Mapper Report
U	MT206	Autoconstrain Mode is ON	-	spcl_reqs.srr (108)	07:37:47 Wed May 09	Mapper Report
U	MT197	Clock constraints cover only FF-to-FF paths associate...	-	spcl_reqs.srr (246)	07:37:47 Wed May 09	Timing Report

TCL Script Messages

3. To reduce the clutter in the window and make messages easier to find and understand, use the following techniques:

- Use the color cues. For example, when you have multiple synthesis runs, messages that have not changed from the previous run are in black; new messages are in red.
- Enable the Group Common IDs option in the upper right. This option groups all messages with the same ID and puts a plus symbol next to the ID. You can click the plus sign to expand grouped messages and see individual messages.
 - There are two types of message groups. The same warning or note ID appears in multiple source files indicated by a dash in the source files column. Multiple warnings or notes in the same line of source code indicated by a bracketed number.
- Sort the messages. To sort by a column header, click that column heading. For example, click Type to sort the messages by type. For example, you can use this to organize the messages and work through the warnings before you look at the notes.
- To find a particular message, type text in the Find field. The tool finds the next occurrence. You can also click the F3 key to search forward, and the Shift-F3 key combination to search backwards.

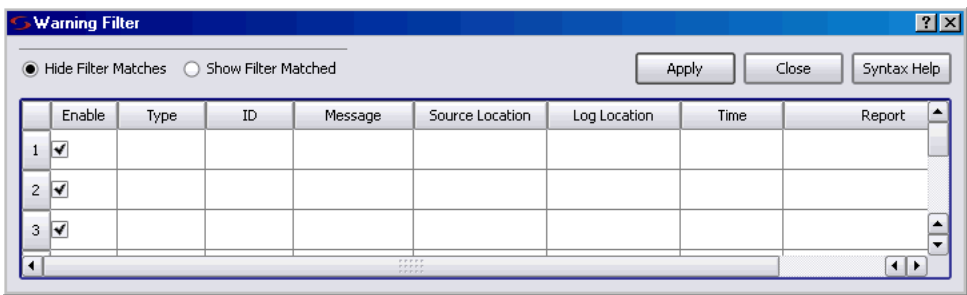
4. To filter the messages, use the procedure described in [Filtering Messages in the Message Viewer, on page 220](#). Crossprobe errors from the message window:
 - If you need more information about how to handle a particular message, click the message ID in the ID column. This opens the documentation for that message.
 - To open the corresponding source code file, click the link in the Source Location column. Correct any errors and rerun synthesis.
 - To view the message in the context of the log file, click the link in the Log Location column.

Filtering Messages in the Message Viewer

The Message viewer lists all the notes, warnings, and errors. The following procedure shows you how to filter out the unwanted messages from the display, instead of just sorting it as described in [Checking Results in the Message Viewer, on page 218](#). For the command line equivalent of this procedure, see [Filtering Messages from the Command Line, on page 222](#).

1. Open the message viewer by clicking the Messages tab in the Tcl window as previously described.
2. Click Filter in the message window.

The Warning Filter spreadsheet opens, where you can set up filtering expressions. Each line is one filter expression.



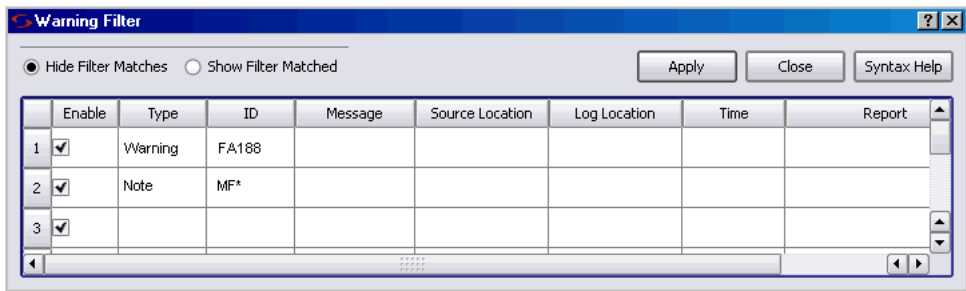
3. Set your display preferences.
 - To hide your filtered choices from the list of messages, click Hide Filter Matches in the Warning Filter window.

- To display your filtered choices, click Show Filter Matches.

4. Set the filtering criteria.

- Set the columns to reflect the criteria you want to filter. You can either select from the pull-down menus or type your criteria. If you have multiple synthesis runs, the pull-down menu might contain selections that are not relevant to your design.

The first line in the following example sets the criteria to show all warnings (Type column) with message ID FA188 (ID). The second set of criteria displays all notes that begin with MF.



- Use multiple fields and operators to refine filtering. You can use wildcards in the field, as in line 2 of the example. Wildcards are case-sensitive and space-sensitive. You can also use ! as a negative operator. For example, if you set the ID in line 2 to !MF*, the message list would show all notes except those that begin with MF.
- Click Apply when you have finished setting the criteria. This automatically enables the Apply Filter button in the messages window, and the list of messages is updated to match the criteria.

The synthesis tool interprets the criteria on each line in the Warning Filter window as a set of AND operations (Warning and FA188), and the lines as a set of OR operations (Warning and FA188 or Note and MF*).

- To close the Warning Filter window, click Close.

5. To save your message filters and reuse them, do the following:

- Save the project. The synthesis tool generates a Tcl file called **projectName.pfl** (Project Filter Log) in the same location as the main project file. The following is an example of the information in this file:

```
log_filter -hide_matches
log_filter -field type==Warning
    -field message==*Una*
    -field source_loc==sendpacket.v
    -field log_loc==usbHostSlave.srr
    -field report=="Compiler Report"
log_filter -field type==Note
log_filter -field id==BN132
log_filter -field id==CL169
log_filter -field message=="Input *"
log_filter -field report=="Compiler Report"
```

- When you want to reuse the filters, source the *projectName.pfl* file.

You can also include this file in a synhooks Tcl script to automate your process.

Filtering Messages from the Command Line

The following procedure shows you how to use Tcl commands to filter out unwanted messages. If you want to use the GUI, see [Filtering Messages in the Message Viewer, on page 220](#).

1. Type your filter expressions in the Tcl window using the `log_filter` command. For details of the syntax, see [log_filter, on page 75](#) in the *Command Reference Manual*.

For example, to hide all the notes and print only errors and warnings, type the following:

```
log_filter -enable
log_filter -hide_matches
log_filter -field type==Note
```

2. To save and reuse the filter commands, do the following:
 - Type the `log_filter` commands in a Tcl file.
 - Source the file when you want to reuse the filters you set up.
3. To print the results of the `log_filter` commands to a file, add the `log_report` command at the end of a list of `log_filter` commands.

```
log_report -print filteredMsg.txt
```


This command prints the results of the preceding `log_filter` commands to the specified text file, and puts the file in the same directory as the main project file. The file contains the filtered messages, for example:

```
@N MF138 Rom slaveControlSel_1 mapped in logic. Mapper Report
wishbonebi.v (156) usbHostSlave.srr (819) 05:22:06 Mon Oct 18
@N(2) MO106 Found ROM, 'slaveControlSel_1', 15 words by 1 bits
Mapper Report wishbonebi.v (156) usbHostSlave.srr (820)
05:22:06 Mon Oct 18
@N MO106 Found ROM, 'slaveControlSel_1', 15 words by 1 bits Mapper
Report wishbonebi.v (156) usbHostSlave.srr (820) 05:22:06 Mon
Oct 18
@N MF138 Rom hostControlSel_1 mapped in logic. Mapper Report
wishbonebi.v (156) usbHostSlave.srr (821) 05:22:06 Mon Oct 18
@N MO106 Found ROM, 'hostControlSel_1', 15 words by 1 bits Mapper
Report wishbonebi.v (156) usbHostSlave.srr (822) 05:22:06 Mon
Oct 18
@N Synthesizing module writeUSBWireData Compiler Report
writeusbwiredata.v (59) usbHostSlave.srr (704) 05:22:06 Mon Oct 18
```

Automating Message Filtering with a Tcl Script

The following example shows you how to use a synhooks Tcl script to automatically load a message filter file when a project opens and to send email with the messages after a run.

1. Create a message filter file like the following. (See [Filtering Messages in the Message Viewer, on page 220](#) or [Filtering Messages from the Command Line, on page 222](#) for details about creating this file.)

```
log_filter -clear
log_filter -hide_matches
log_filter -field report=="LIFCL MAPPER"
log_filter -field type==NOTE
log_filter -field message=="Input *"
log_filter -field message=="Pruning *"
puts "DONE!"
```

2. Copy the `synhooks.tcl` file and set the environment variable as described in [Automating Flows with synhooks.tcl, on page 571](#).
3. Edit the `synhooks.tcl` file so that it reads like the following example. For syntax details, see [synhooks File Syntax, on page 278](#) in the *Reference Manual*.

- The following loads the message filter file when the project is opened. Specify the name of the message filter file you created in step 1. Note that you must source the file.

```
proc syn_on_open_project {project_path} {
  set filterFilename
  puts "FILTER $filter IS BEING APPLIED"
  source d:/tcl/filters/$filterFilename
}
```

- Add the following to print messages to a file after synthesis is done:

```
proc syn_on_end_run {runName run_dir implName} {
  set warningFileName "messageFilename"

  if {$runName == "synthesis"} {
    puts "Mapper Done!"
    log_report -print $warningFileName
    set f [open [lindex $warningFileName] r]
    set msg ""
    while {[gets $f warningLine]>=0} {
      puts $warningLine
      append msg $warningLine\n
    }
    close $f
  }
```

- Continue by specifying that the messages be sent in email. You can obtain the `smtp` email packages off the web.

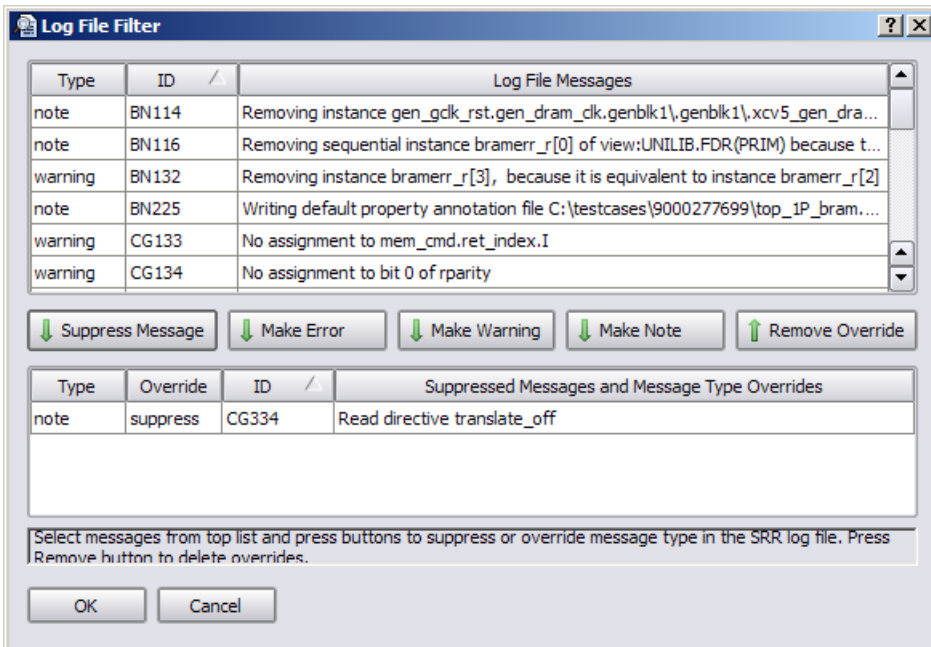
```
source "d:/tcl/smtp_setup.tcl"
proc send_simple_message {recipient email_server subject body}{
  set token [mime::initialize -canonical text/plain -string
    $body]
  mime::setheader $token Subject $subject
  smtp::sendmessage $token -recipients $recipient -servers
    $email_server
  mime::finalize $token
}
puts "Sending email..."

send_simple_message {address1,address2}
  yourEmailServer subjectText> emailText
}
```

When the script runs, an email with all the warnings from the synthesis run is automatically sent to the specified email addresses.

Log File Message Controls

The log file message control feature allows messages in the current session to be elevated in severity (for example, promoted to an error from a warning), lowered in severity (for example, demoting a warning to a note), or suppressed from the log file after the next run through the Log File Filter dialog box. This dialog box is displayed by selecting Set Filter from the Messages window and clicking Log File Filter from the GUI.



Log File Filter Dialog Box

The Log File Filter dialog box is the primary control for changing a message priority or suppressing a message. When you initially open the dialog box, all of the messages from the log (srr) file for the active implementation are displayed in the upper section and the lower section is empty. To use the dialog box:

1. Select (highlight) the message to be promoted, demoted, or suppressed from the messages displayed in the upper section.

2. Select the Suppress Message, Make Error, Make Warning, or Make Note button to move the selected message from the upper section to the lower section. The selected message is repopulated in the lower section with the Override column reflecting the disposition of the message according to the button selected.

Allowed Severity Changes

Allowed severity levels and preference settings for warning, note, and advisory messages are:

- Promote – warning to error, note to warning, note to error
- Demote – warning to note
- Suppress – suppress warning, suppress note, suppress advisory

Note: Normal error messages (messages generated by default) cannot be suppressed or changed to a lesser severity level.

When using the dialog box:

- Use the control and shift keys to select multiple messages.
- If an `srr` file is not present (for example, if you are starting a new project) the table will be empty. Run the design at least once to generate an `srr` file.
- Clicking the OK button saves the message status changes to the *project-Name.pfl* file in the project directory.

Message Reporting

The compiler and mapper must be rerun before the impact of the message status changes can be seen in the updated log file.

When a *projectName.pfl* input file is present at the start of the run, the message-status changes in the file are forwarded to the mapper and compiler which generate an updated log file. Depending on the changes specified:

- If an ID is promoted to an error, the mapper/compiler stops execution at the first occurrence of the message and prints the message in the `@E:msgID :messageText` format

- If an ID is promoted to a warning, the mapper/compiler prints the message in the `@W:msgID :messageText` format.
- If an ID is demoted to a note, the mapper/compiler prints the message in the `@N:msgID :messageText` format.
- If an ID is suppressed, the mapper/compiler excludes the message from the `srr` file.

Note: The online, error-message help documentation is unchanged by any message modification performed by the filtering mechanism. If a message is initially categorized as a warning in the synthesis tool, it continues to be reported as a warning in error-message help irrespective its promotion/demotion status.

Updating the *projectName.pfl* file

The *projectName.pfl* file in the top-level project directory stores the user message filter settings from the Log File Filter dialog box for that project. This file can be edited with a text editor. The file entry syntax is:

```
message_override -suppress ID [ID ...] | -error ID [ID ...] | -warning ID [ID ...]  
| -note ID [ID ...]
```

For example, to override the default message definition for note FX702 as a warning, enter:

```
message_override -warning FX702
```

You can also limit the number of occurrences for specified message IDs with the following syntax:

```
message_override [-limit value] [-count value]
```

For example, limit messages with IDs FX214 and FX271 to 100 each in each log file as follows:

```
message_override -limit {FX214 FX271} -count 100
```

Then, select the message filter file (*.pfl*) to be read for the project with the Read Message File option.

Note: After editing the `pfl` file, close and reopen the project to update the overrides.

messagefilter.txt File

A `messagefilter.txt` file in the *implementation*/`syntmp` directory lists any changes made to message priority or suppression through the Log File Filter dialog box. This file, which is only generated when changes are made to the default status of a message, can be accessed outside of the GUI without consuming a license.

Working with Downgradable Errors and Critical Warnings

You can temporarily change the classification for certain kinds of messages:

Downgradable errors (@DE)	Can be downgraded to warnings. This is a small set of <i>non-fatal</i> errors where you can temporarily postpone addressing the error and continue with the design flow and verification of other aspects of the design.
Critical warnings (@CW)	Can be upgraded to errors. This small set of warnings represent critical problems. Elevating them to errors ensures that they are recognized and dealt with, because the error status forces the tool to stop.

You can downgrade or upgrade these messages from the GUI or through a Tcl command.

Downgrading or Upgrading Messages from the GUI

1. Open the log file.
2. Right click within the log file and select Log File Message Filter from the pop-up menu to display the Log File Filter dialog box.
3. Highlight the DE or CW message from the list of messages displayed at the top of the dialog box. You can only downgrade messages with a DE prefix, or upgrade messages with a CE prefix.
4. Depending on the message type selected:

- Click the Make Warning button to downgrade the error to a warning. This action moves the message from the top section of the message filter to the bottom section. The Override column in the bottom section displays the updated status for the message (Warning).
 - Click the Make Error button to upgrade the critical warning to an error. This action moves the message from the top section of the message filter to the bottom section. The Override column in the bottom section displays the updated status for the message (Error).
5. To see the changes reflected, click the Run button. You can verify that the message is now treated as:
- A warning (the mapping operation continues past the initial point of the error condition)
 - An error (the operation stops at the initial point of the error).
6. To revert the DE or CW message:
- Click on one of the downgraded DE warning IDs in the report and select Log File Message Filter from the pop-up menu to display the Log File Filter dialog box. This reverts the warning message to an error.
 - Click on one of the upgraded CW warning IDs in the report and select Log File Message Filter from the pop-up menu to display the Log File Filter dialog box. This reverts the error message to a warning.
7. From the dialog box, highlight the DE or CW message from the list of messages displayed at the top of the dialog box. Click either the:
- Make Error button to return the message to its original error status.
 - Make Warning button to return the message to its original critical warning status.

Modifying the status of a message does not affect the message string. A message originally categorized as an error continues to be reported as an error regardless of its user-assigned status.

Downgrading or Upgrading Messages with a Tcl Command

Type the `message_override` command in the Tcl script window to change the classification of DE and CW messages.

1. To downgrade DE errors or upgrade CW warnings, use the appropriate `message_override` command:

```
message_override -warning DEmessageID  
message_override -error CWmessageID
```

Enter the message ID without the @ prefix, as shown above.

2. To see the changes reflected, rerun the state from the database where you changed the message classification.

After you have fixed the cause of the upgraded warning (critical warning) or completed the rest of the flow (downgradable error), you must change the message classification back to its original status.

3. To revert the messages back to their original status, use the appropriate `message_override` command:

```
message_override -error DEmessageID  
message_override -warning CWmessageID
```

4. Rerun again to confirm that the message status has reverted to the original.

CHAPTER 7

Analyzing with HDL Analyst

This chapter describes how to analyze logic in the HDL Analyst and FSM Viewer.

See the following for detailed procedures:

- [Working in the Schematic](#), on page 232
- [Exploring Design Hierarchy](#), on page 247
- [Finding Objects](#), on page 255
- [Crossprobing](#), on page 264
- [Analyzing With the HDL Analyst Tool](#), on page 271
- [Working in the Schematic](#), on page 232
- [Exploring Design Hierarchy](#), on page 307
- [Finding Objects \(Standard\)](#), on page 315
- [Crossprobing \(Standard\)](#), on page 328
- [Analyzing With the Standard HDL Analyst Tool](#), on page 336
- [Using the FSM Viewer \(Standard\)](#), on page 353

Working in the Schematic

The HDL Analyst tool includes single-page schematics, which can help you graphically analyze and navigate your entire design easier. This section describes basic procedures you use in the schematics. These procedures include the following topics:

- Clone Schematic – See [Cloning Schematics](#), on page 235
- Instance Groups – See [Grouping Objects in the Schematic](#), on page 241
- Partial Dissolve – See [Grouping Objects in the Schematic](#), on page 241
- Net Based Filtering – See [Filtering Schematics](#), on page 275
- Unfilter – See [Filtering Schematics](#), on page 275
- Multi-threaded Find – See [Browsing to Find Objects in HDL Analyst Views](#), on page 255
- New Mouse Strokes with Cancel Display– See [Mouse Stroke Conventions](#), on page 235
- New Push View Tab – See [Cloning Schematics](#), on page 235
- Peek – See [Viewing Design Hierarchy and Context](#), on page 271
- Show Contents – See [Viewing Design Hierarchy and Context](#), on page 271
- Improved Bus Display and Handling – See [Dissolving and Partial Dissolving of Buses and Pins](#), on page 283

For information on specific tasks like analyzing critical paths, see the following sections:

- [Traversing Design Hierarchy with the Hierarchy Browser](#), on page 247
- [Exploring Object Hierarchy with Push/Pop Commands](#), on page 250
- [Crossprobing](#), on page 264
- [Analyzing With the HDL Analyst Tool](#), on page 271

Opening the Views

The procedure for opening a view is the same at different design stages; the main difference is the content that is available at the different design database states.

1. Start at the database state you want.

RTL view	Start with a compiled design.
Technology view	Start with a mapped (synthesized) design.

2. To enable the new HDL Analyst tool, use one of the following methods:

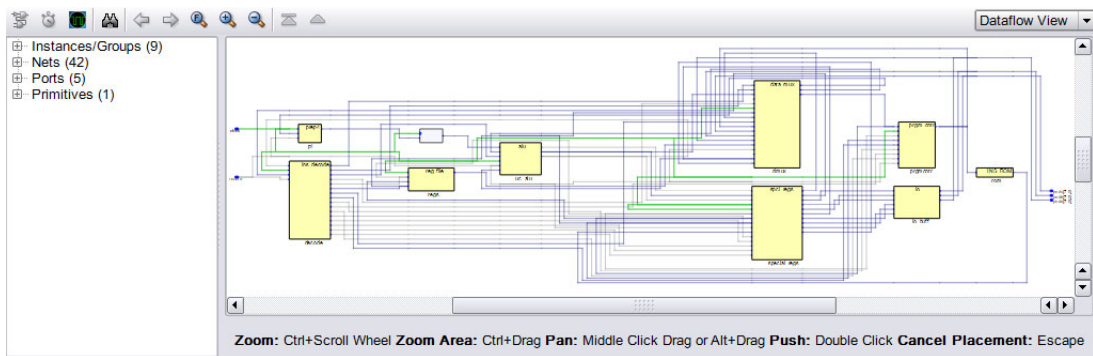
- From the UI, Select HDL Analyst->Use New HDL Analyst option.
- From the UI: Select Options->Use New HDL Analyst option.
- By default, this option is enabled.

3. Open the schematic using one of the following commands:

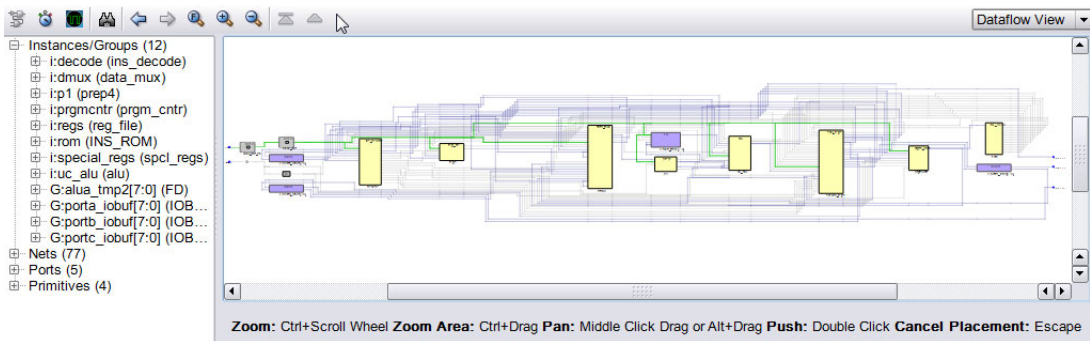
Hierarchical RTL or Technology view	Use one of these methods: • Select HDL Analyst->RTL->Hierarchical View. • Click the RTL View icon ( (a plus sign inside a circle)). • Double-click the <code>vm</code> file in the Implementation Results view. To open a flattened RTL view, select HDL Analyst->RTL->Flattened View.
Hierarchical Technology view	Use one of these methods: • Select HDL Analyst ->Technology->Hierarchical View. • Click the Technology View icon (NAND gate icon ). • Double-click the <code>srnm</code> file in the Implementation Results view.
Flattened RTL or Technology view	Select HDL Analyst->RTL->Flattened View or HDL Analyst->Technology->Flattened View

All schematic views have the schematic on the right and a pane on the left that contains a hierarchical list of the objects in the design. This pane is called the Hierarchy Browser.

Compiled View



Mapped View

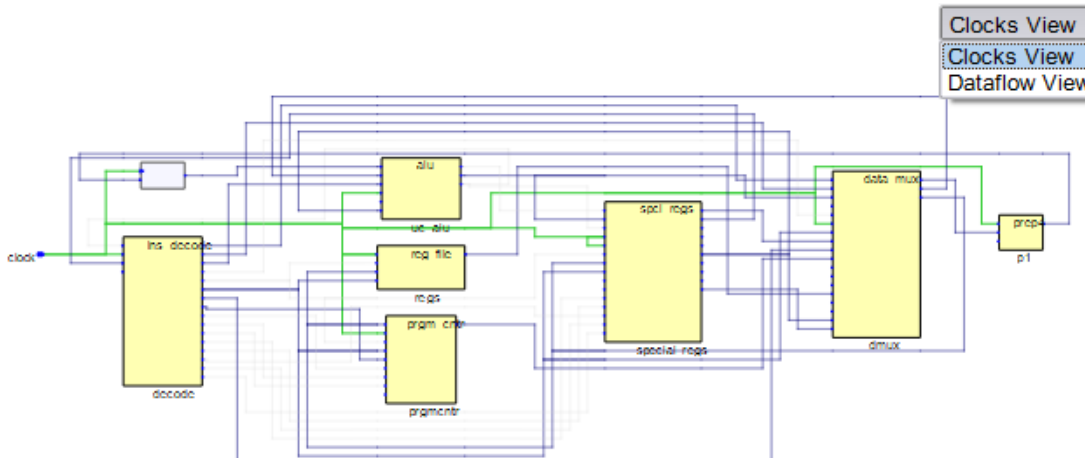


Dataflow View

Both the compiled and mapped views have a Dataflow View. Use this view to display objects from a left to right datapath flow as shown above. You can display a Clock View as well.

Clock View

To display all sequential elements connected to clock nets and debug the clocks in the design use this view. Select Clocks View from the drop-down menu in the upper right corner of the schematic view. Clock nets are displayed with the color green.



Mouse Stroke Conventions

Use the mouse strokes to control navigation and the display, which are listed at the bottom of the schematic window. They include:

- Zoom – Ctrl-scroll wheel
- Zoom Area – Ctrl-drag
- Pan – Middle-click drag or Alt-drag
- Push – Double-click
- Pop – Double-click an empty space
- Cancel Display – Press Esc
(For large designs, you can cancel displaying the netlist but still use the netlist from the hierarchy browser, Tcl window, and Find dialog box.)

Cloning Schematics

Most operations performed in any of the HDL Analyst views (Clock View or Dataflow View) are displayed in the current view. To create a new view, use the clone commands.

1. To clone the current view displayed, right-click and select Clone Schematic from the drop-down menu. This view opens in a new window. You can open multiple clone views.

Tcl equivalent: `analyst clone`

To close a clone view: `analyst close_design designID`.

For example: `analyst close_design d:3`

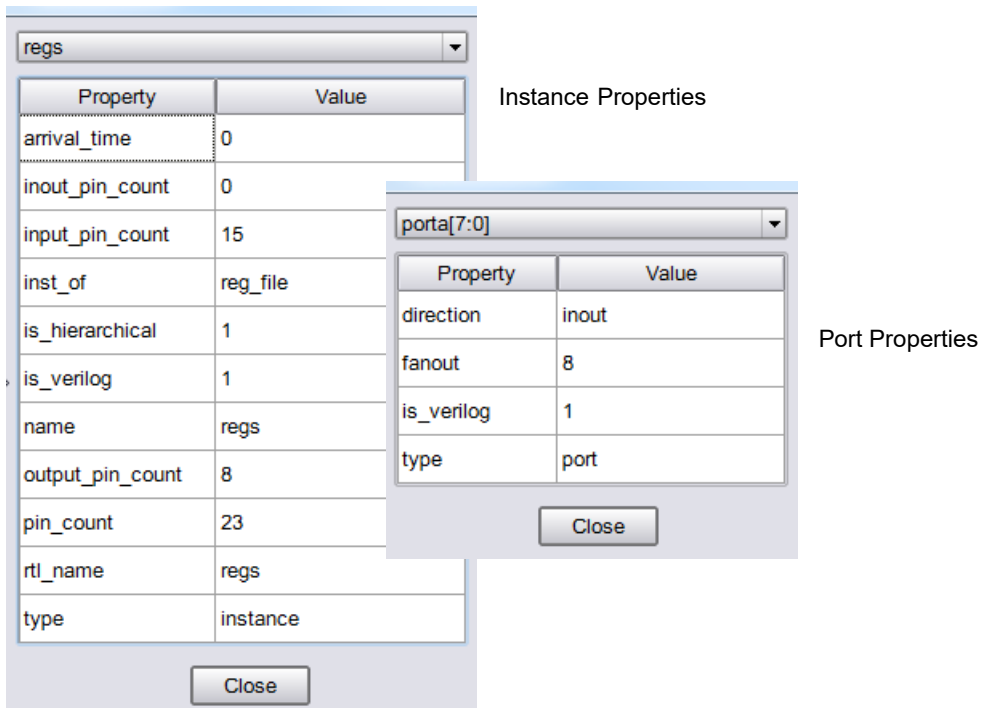
2. To push into an object and create a new view, select an object then right-click and select Push in New Tab from the drop-down menu. For more information, see [Exploring Object Hierarchy with Push/Pop Commands](#), on page 250.
3. To filter objects and create a new view, select the objects then right-click and select Filter in a New Tab from the drop-down menu. For more information, see [Filtering Schematics](#), on page 275.

Viewing Object Properties

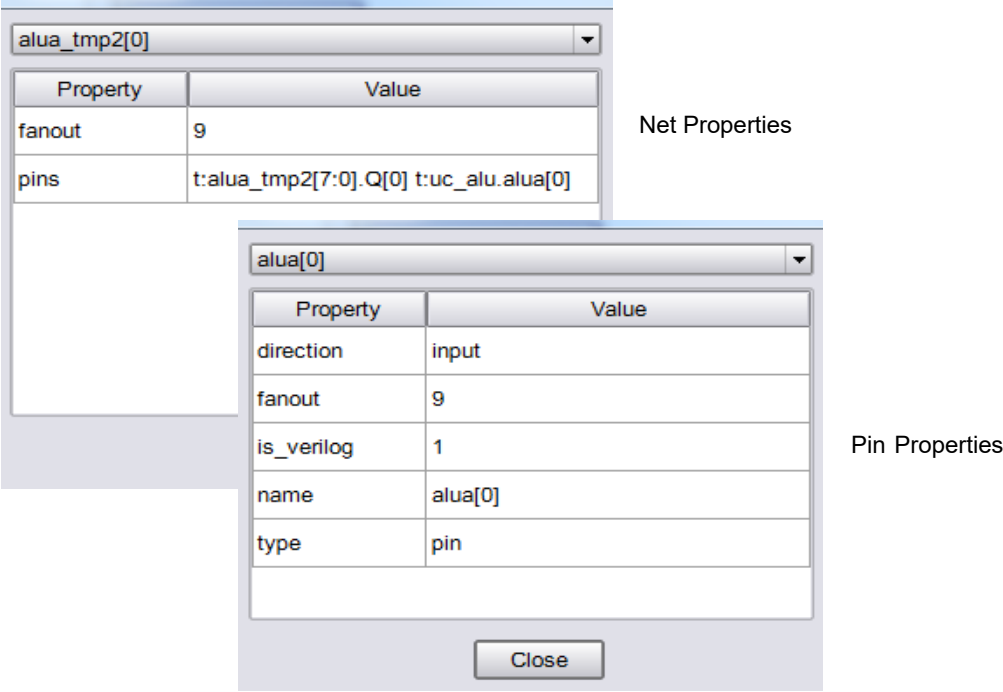
There are a few ways in which you can view the properties of objects.

1. To temporarily display the properties of a particular object, hold the cursor over the object.
2. Select the object, right-click, and select Properties. The properties and their values are displayed in a table.

For example, you can view the properties for instances and ports as shown below.



Similarly, you can view the properties for pins and nets.



3. You can copy one field at a time from the Properties dialog box and paste this property to the Tcl window or a text file from within the tool.

For example, use this field with the collection commands to identify groups of objects in the schematic.

Selecting Objects in the Schematic

For mouse selection, standard object selection rules apply:

To select ...	Do this ...
Single objects	Click on the object in the schematic, or click the object name in the Hierarchy Browser. Tcl equivalent: select {!:<i>instanceName</i>}

To select ...	Do this ...
Multiple objects	Use one of these methods: • Draw a rectangle around the objects. • Select an object, press Ctrl, and click other objects you want to select. • Select multiple objects in the Hierarchy Browser. See Browsing With the Hierarchy Browser , on page 255. • Use Find to select the objects you want. See Finding Objects , on page 255. Tcl equivalent: select {i:instance1} {i:instance2} • Select all instances in the current view: Tcl equivalent: select -instances
Objects by type (instances, ports, nets)	Use Find to select the objects (see Browsing With the Find Command , on page 260), or use the Hierarchy Browser, which lists objects by type.
No objects (deselect all currently selected objects)	Click the left mouse button in a blank area of the schematic. Deselected objects are no longer highlighted. Tcl equivalent: select -clear

The HDL Analyst view highlights selected objects in red. If you have other windows that are cloned, the selected object is highlighted in the other windows as well (crossprobing).

Selecting a Sequence of Objects in the Schematic

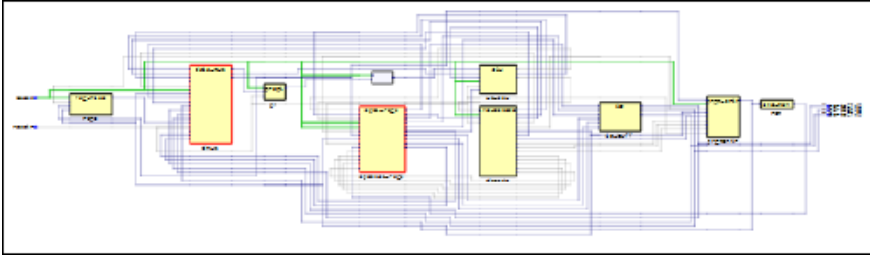
You can select a series of objects in the schematic, then traverse back and forth through these selections in the order they were chosen. Use the backwards and forwards icons () to move between each selection. This is handy to help you undo or redo any changes with your selections. Once you select an operation, the selections becomes unavailable.

When an object is selected in the schematic, you can use the following command to print the name of the object (instance, port, pin, or net):

```
analyst get_selected [-inst] [-net] [-port] [-pin]
```

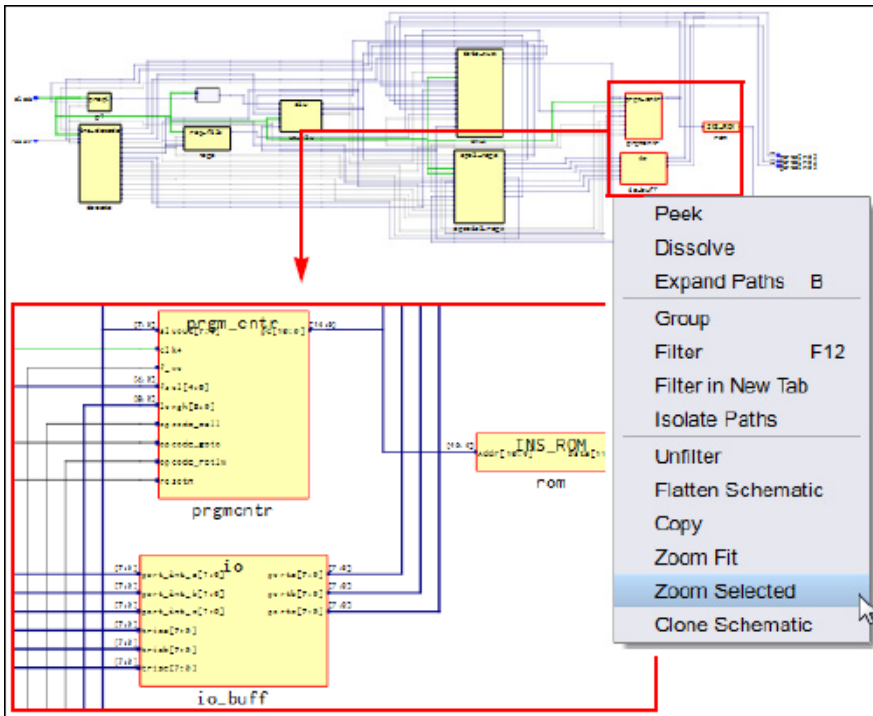
In the following example, two instances are selected. Specify the following command to display their names in the Tcl window:

```
% analyst get_selected -inst
{i:dmux} {i:special_regs}
```



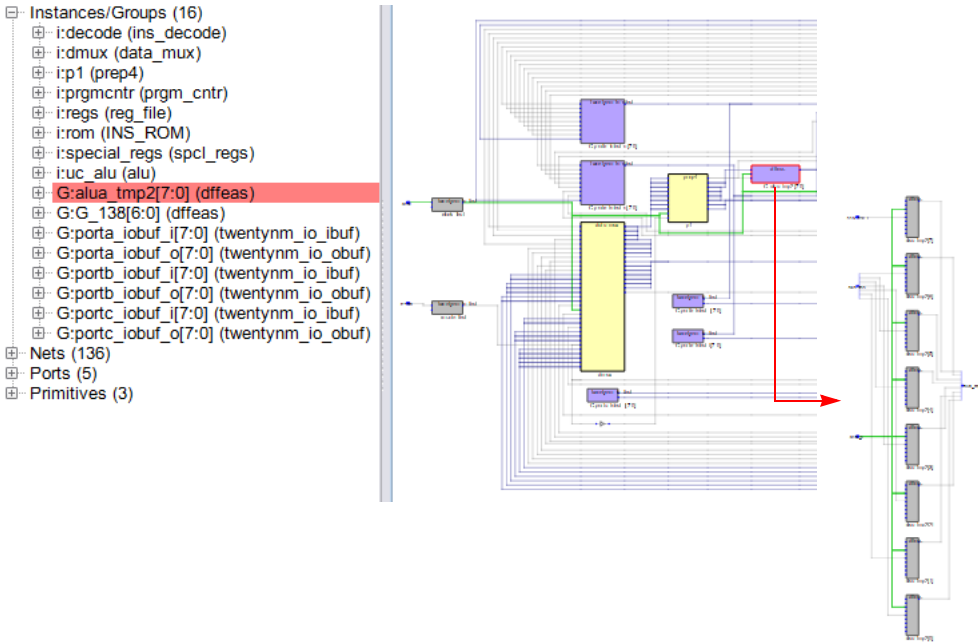
Zooming in on Selected Objects

Once you have selected objects in the schematic view, you can automatically zoom in on these objects. To do this, highlight the required objects, then right-click and select **Zoom Selected** from the drop-down menu.



Grouping Objects in the Schematic

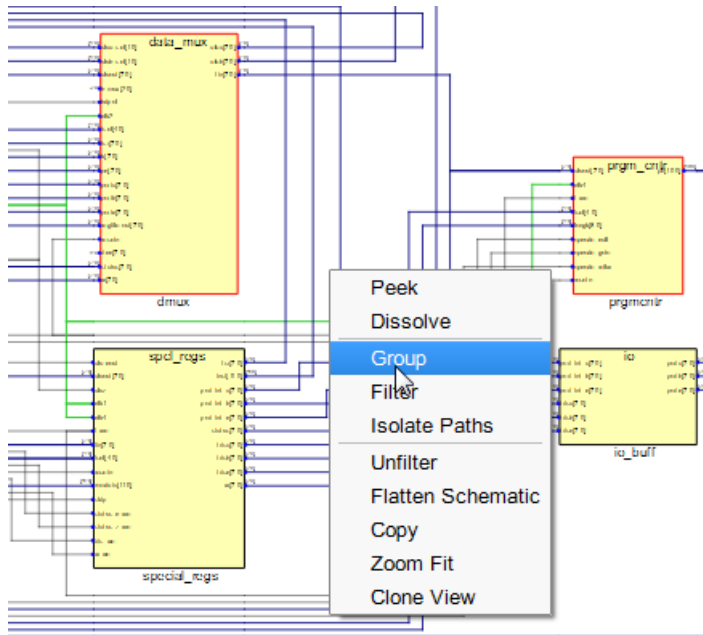
You can group objects in the schematic. Sometimes the HDL Analyst tool automatically determines groups shown with the color purple below. When you push into the group block, the content of its objects is displayed.



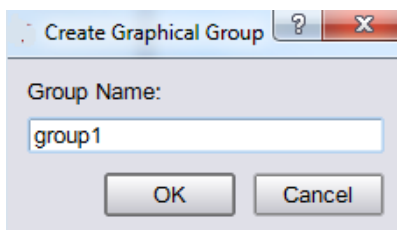
The HDL Analyst tool automatically groups instances with similar names at all levels of hierarchy, when you enable the Allow Automatic Grouping option on the HDL Analyst Options dialog box. For example, suppose there are three registers with the names `out_reg[1]`, `out_reg[2]`, and `out_reg[3]`. A group will be created with the registers having the name `out_reg[3:1]`.

To create your own groups:

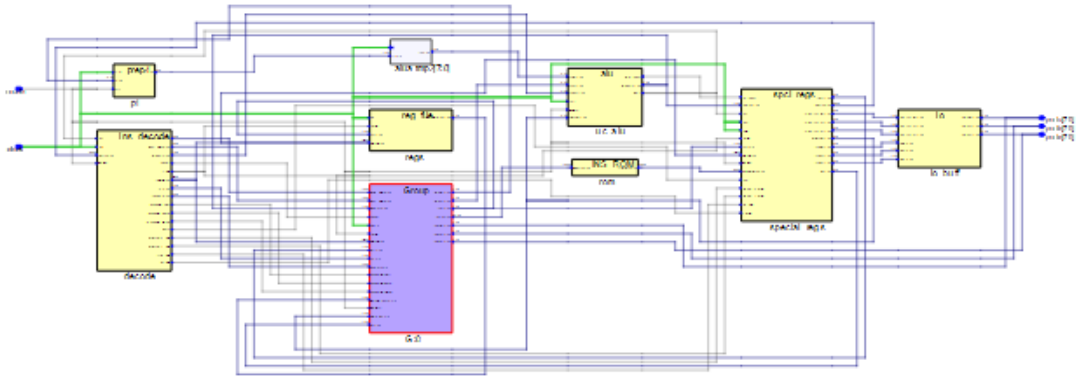
1. Select the specified objects in the view.
2. Right-click and select the Group option from the pop-up menu.



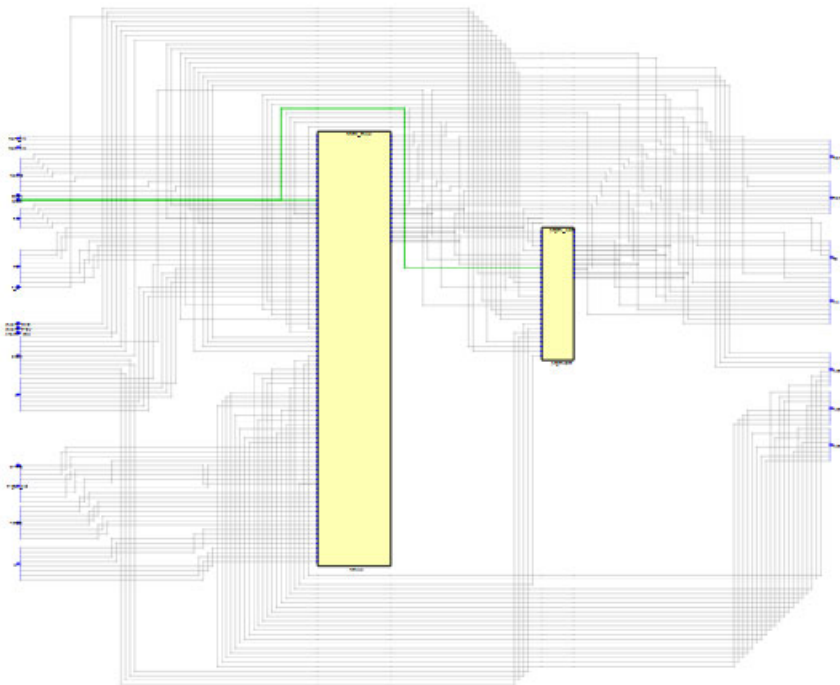
3. Specify a group name in the dialog box that pops up.



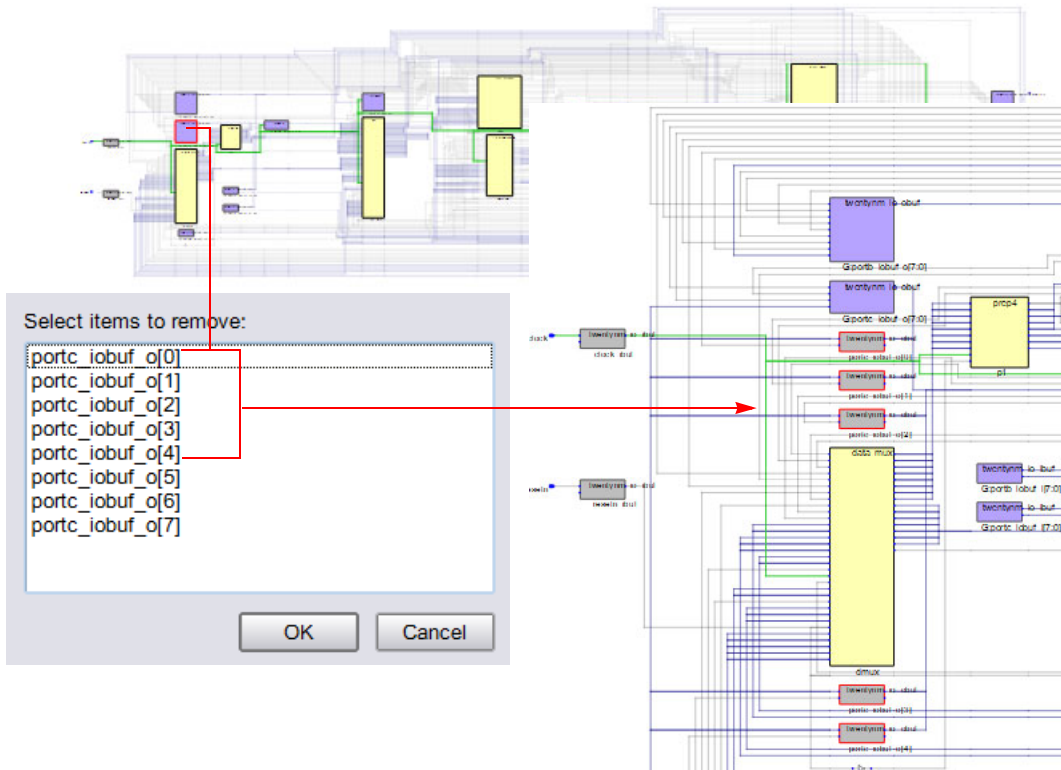
The objects are now grouped together in the purple block.



4. Push into the purple block to see the grouped objects.



5. To ungroup selections, right-click and select Dissolve from the drop-down menu.
6. To remove individual instances of a group, right-click and select Partial Dissolve from drop-down menu. A dialog box is displayed where you can select the items to remove.



7. User-created groups are not saved when you close and re-open the same netlist.

Moving Between Views in a Schematic Window

When you filter or expand your design, you move through a number of different design views in the same schematic window. For example, you might start with a view of the entire design, then filter an object and finally expand a connection in the filtered view, for a total of three views. You can also move back after flattening a view.

1. To move back to the previous view, click the Back icon ().

The software displays the last view.

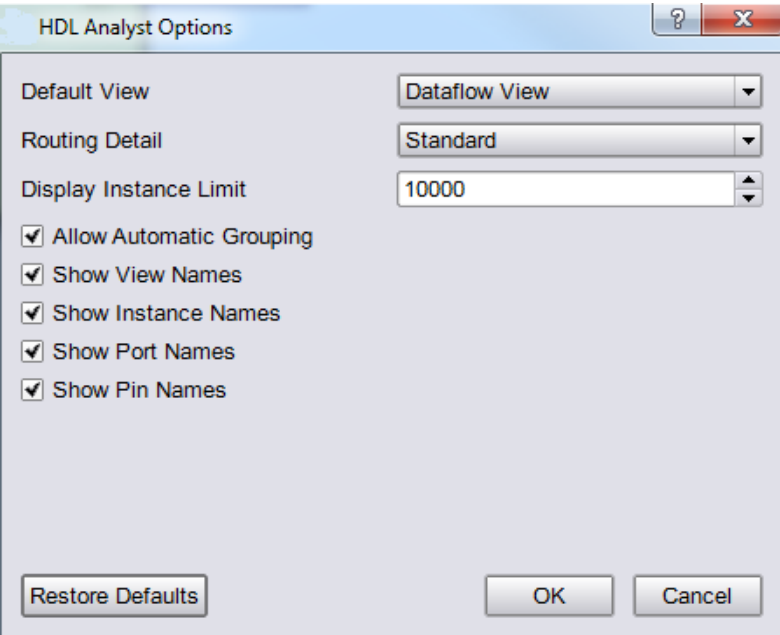
2. To move forward again, click the Forward icon ().

The software displays the next view in the display history.

Setting Schematic Preferences

You can set various preferences for the schematic from the user interface.

1. Select either:
 - HDL Analyst->Schematic Options
 - Options->Schematics Options



For a description of all the options on this form, see [HDL Analyst Options Command](#), on page 475 in the *Command Reference Manual*.

2. The table details the following operations:

To...	Do this...
Specify how you want the schematic to display.	Select Clock View or Dataflow View (default).
Specify how the tool determines the detailed routing for the design.	Select Optimized, Detailed, Standard (default), or Quick (direct connection). The time to route the design increases as you go from Quick to Optimized mode. The schematic is the most clean and uncluttered for the Optimized mode.
Determine if you want automatic grouping for the design.	When Allow Automatic Grouping is enabled, the tool automatically groups instances with similar names at every level in the design.

Exploring Design Hierarchy

Schematics generally have a certain amount of design hierarchy. You can move between hierarchical levels using the Hierarchy Browser mode. For additional information, see [Analyzing With the HDL Analyst Tool, on page 271](#). See [Traversing Design Hierarchy with the Hierarchy Browser, on page 247](#).

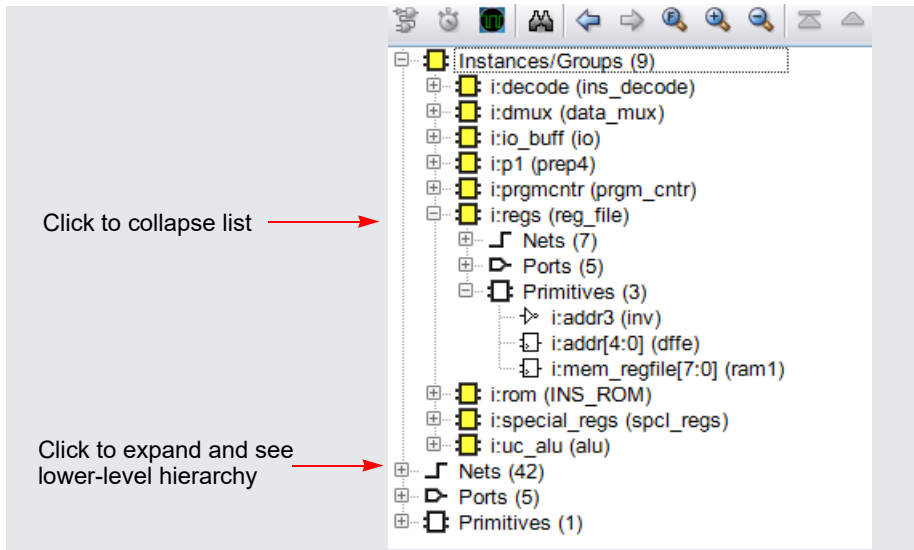
Traversing Design Hierarchy with the Hierarchy Browser

The Hierarchy Browser is the list of objects on the left side of the schematic view. It is best used to get an overview, or when you need to browse and find an object. If you want to move between design levels of a particular object, using the Push command is more direct. Refer to [Exploring Object Hierarchy with Push/Pop Commands, on page 250](#) for details.

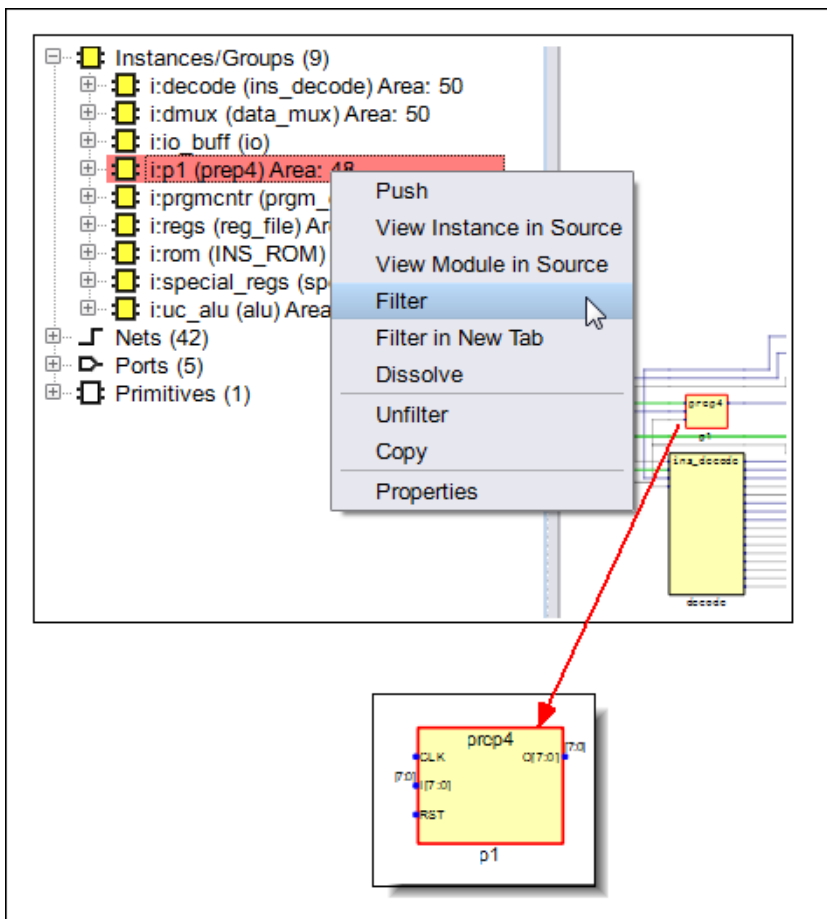
The hierarchy browser allows you to traverse and select the following:

- Instances or Groups
- Ports
- Internal nets

The browser lists the objects by type. Use the expand (⊕) and collapse (⊖) signs to ascend or descend the hierarchy.



1. You can perform some similar operations as done in the schematic view, such as filtering an object from the Hierarchy Browser. For example, highlight an instance, then right-click and select Filter as shown below.



2. You can also crossprobe to instances and modules in the source file from the Hierarchy Browser. Right-click and select either:
 - View Instance in Source
 - View Module in Source

For details, see [Crossprobing, on page 264](#).

Exploring Object Hierarchy with Push/Pop Commands

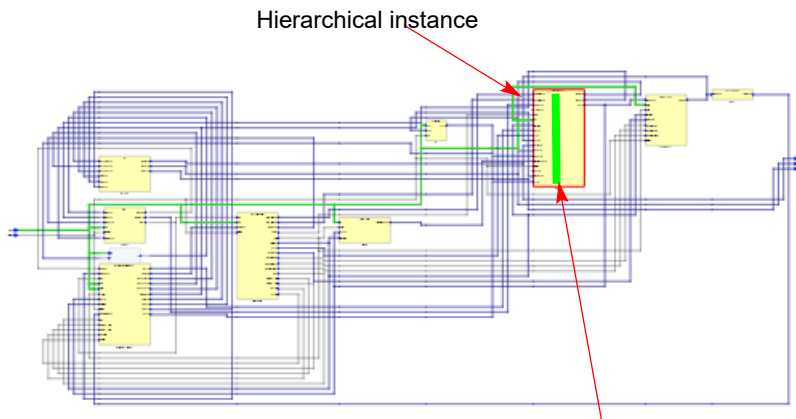
To view the internal hierarchy of a specific instance, use the Push/Pop commands from the drop-down menu or mouse strokes. When combined with other commands like filtering and expansion commands, Push/Pop can be a very powerful tool for isolating and analyzing logic. See [Filtering Schematics, on page 275](#) and [Expanding Pin and Net Logic, on page 277](#) for details about filtering and expansion. See the following sections for information about pushing down and popping up in hierarchical design objects:

- [Pushing into Objects, on page 250](#)
- [Popping up a Hierarchical Level, on page 253](#)

Pushing into Objects

In the schematic, you can push into instances and view the lower-level hierarchy. You can use a mouse stroke or the command to push into objects:

1. To move down a level (push into an object) with a mouse stroke, put your cursor near the top of the object, hold down the right mouse button, and draw a vertical stroke from top to bottom. You can push into the following objects; see step 3 for examples of pushing into different types of objects.
 - Hierarchical instances. They can be displayed as pale yellow boxes (opaque instances).



Press right mouse button and draw downward to push into an object

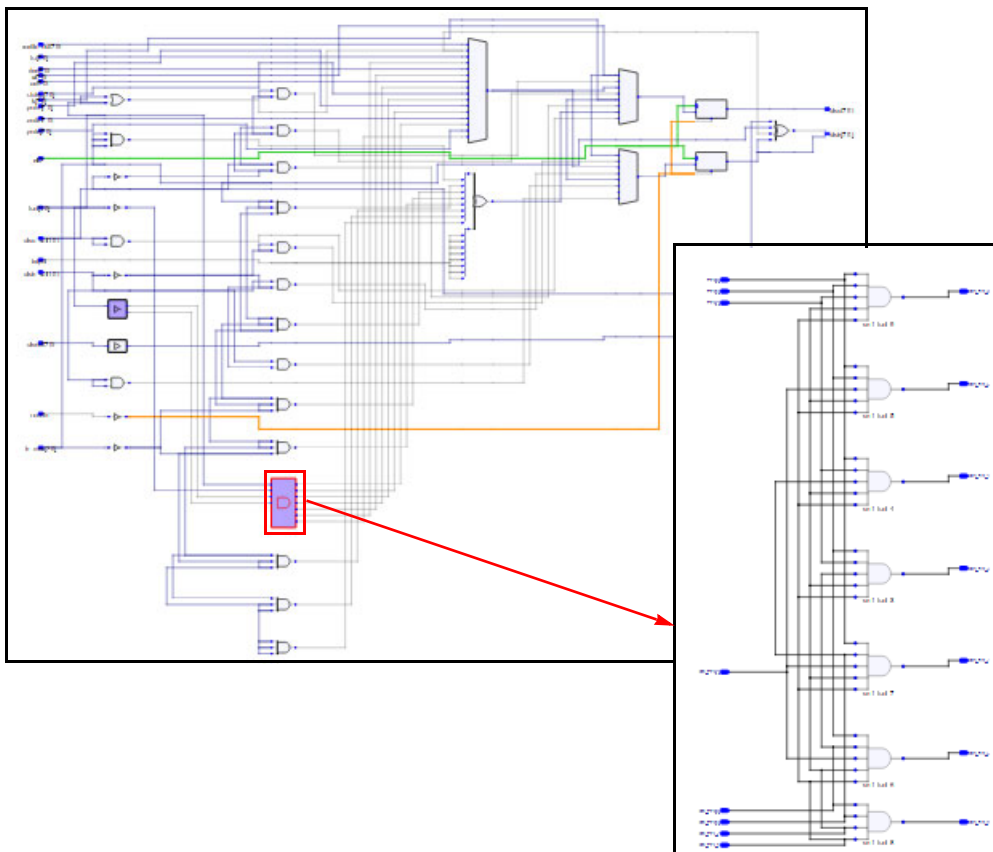
- Technology-specific primitives. The primitives are listed in the Hierarchy Browser in the schematic, under `i:instanceNames->Primitives`.
- Inferred ROMs and state machines.

The remaining steps show you how to use the icon or command to push into an object.

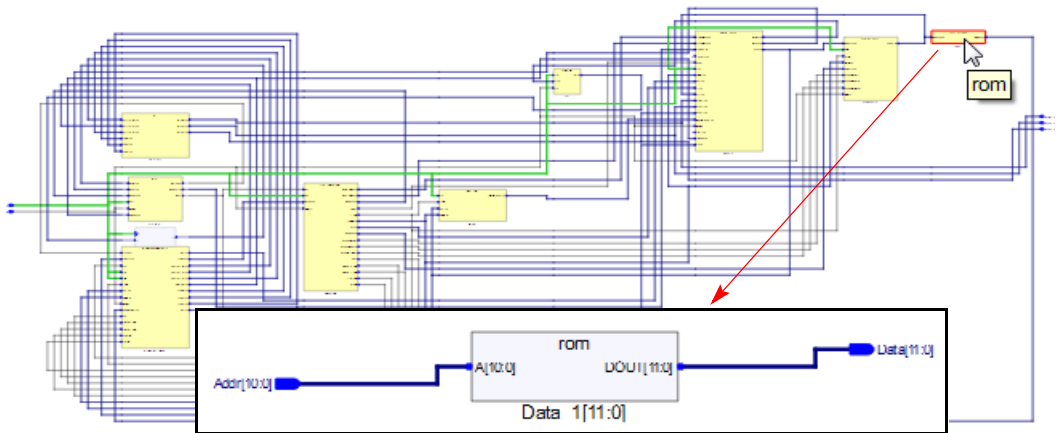
2. Enable the Push/Pop command by doing one of the following:

- Double-click on the object.
- Right-click in the view and select Push/Pop from the drop-down menu.
- Use the mouse strokes.

After pushing into an instance, the following schematic is displayed. Notice the purple block that groups gates with similar names together. You can push into this block as well.



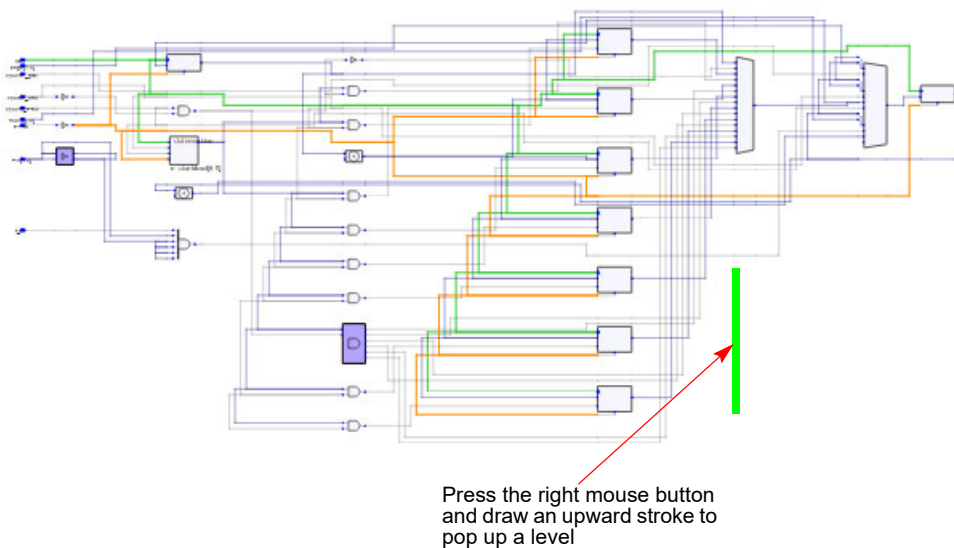
3. To push (descend) into an object, double-click on the hierarchical object. The following figure shows the result of pushing into a ROM.



Popping up a Hierarchical Level

1. To move up a level (pop up a level), put your cursor anywhere in the design,
 - Use the Pop Hierarchy icon ().

Hold down the right mouse button, and draw a vertical mouse stroke, moving from the bottom upwards.



The software moves up a level, and displays the next level of hierarchy.

2. Alternatively, you can double-click on any whitespace in the view to pop up a level from where you pushed into the hierarchy.

Finding Objects

In the schematics, you can use the Hierarchy Browser or the Find command to find objects, as explained in these sections:

- [Browsing to Find Objects in HDL Analyst Views](#), on page 255
- [Using Wildcards with the Find Command](#), on page 263

Browsing to Find Objects in HDL Analyst Views

You can always zoom in to find an object in the schematic. Use Zoom Fit to quickly fit all objects into the schematic. The following procedure shows you how to browse through design objects and find an object at any level of the design hierarchy. You can use the Hierarchy Browser or the Find command to do this. If you are familiar with the design hierarchy, the Hierarchy Browser can be the quickest method to locate an object. The Find command is best used to graphically browse and locate the object you want.

Browsing With the Hierarchy Browser

1. In the Hierarchy Browser, click the name of the net, port, or instance you want to select.

The object is highlighted in the schematic.

2. To select a range of objects, you can press and hold the Shift key while clicking the selected objects in the range.

The software selects and highlights all the objects in the range.

3. If the object is on a lower hierarchical level, do either of the following:
 - Expand the appropriate higher-level object by clicking the collapsed symbol next to it, and then select the object you want.
 - Push down into the higher-level object, and then select the object from the Hierarchy Browser.

The selected object is highlighted in the schematic. However, you may have to filter the object to view it in the design hierarchy.

4. To select all objects of the same type, select them from the Hierarchy Browser. For example, you can find all the nets in your design.

Browsing With the New HDL Analyst

Using the HDL Analyst tool, you can display a hierarchy of design objects in the Hierarchy Browser. Typically, the time taken to display a design hierarchy depends on the size of the design.

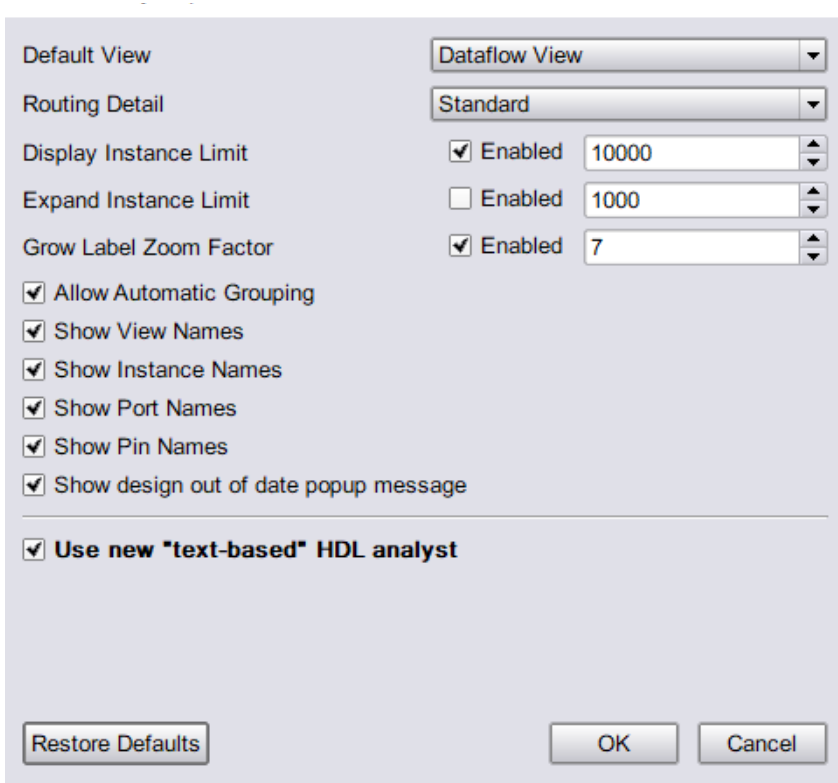
To speed up this process, the new HDL Analyst traverses the entire (text-based) netlist to quickly extract hierarchical instance data. This helps to display the entire netlist hierarchy quickly and also facilitates the viewing of custom instances on demand, instead of traversing down the design hierarchy. This flow is advantageous in large designs, to display the hierarchy and view any instance, quickly.

To enable the new HDL Analyst, follow these steps:

1. To set preference in the schematic from the user interface, select one of the following:
 - HDL Analyst->Schematic Options
 - Options->Schematics Options

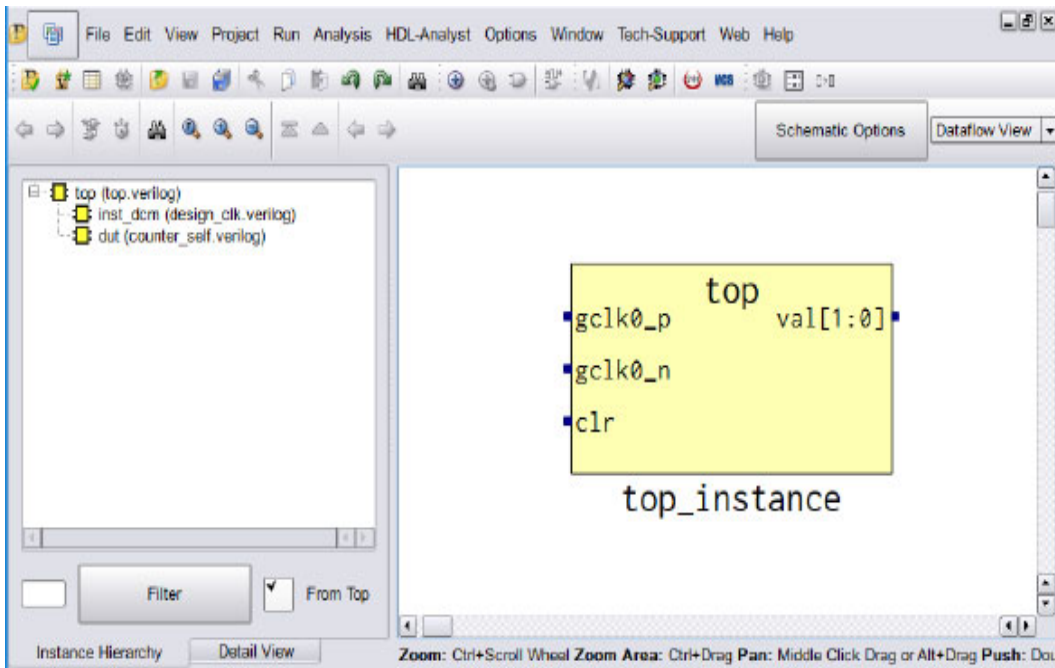
The HDL Analyst Options dialog box is displayed.

2. Enable the Use new “text-based” HDL analyst option.



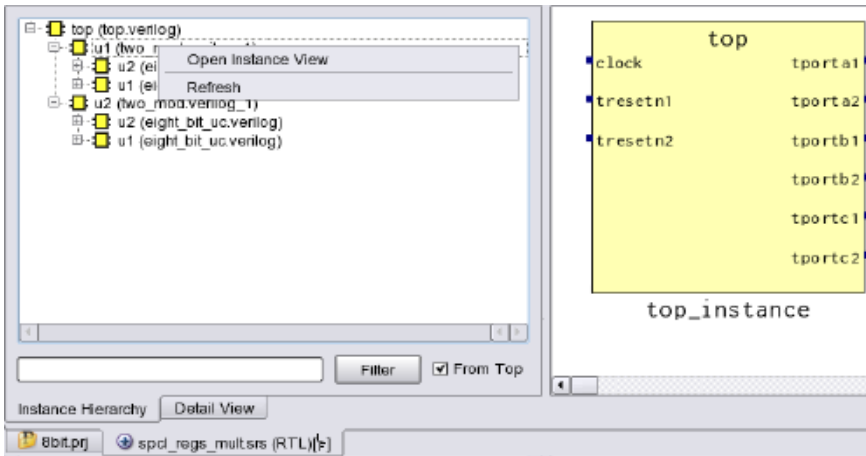
When the next schematic is displayed, if the Use new “text-based” HDL analyst option is enabled, the HDL Analyst tool displays the following:

- The design hierarchies (Instance Hierarchy tab)
 - The corresponding objects in the loaded hierarchy (Detail View tab).
3. Click OK to close the dialog box. Now each design you view will follow the updated option settings.
 4. Select the RTL View icon or select RTL->Hierarchical View from the HDL-Analyst menu, to view the RTL view of the design.



The Instance Hierarchy tab is visible only when the Use new “text-based” HDL analyst option is enabled in the Schematic Options (HDL Analyst Options) dialog box.

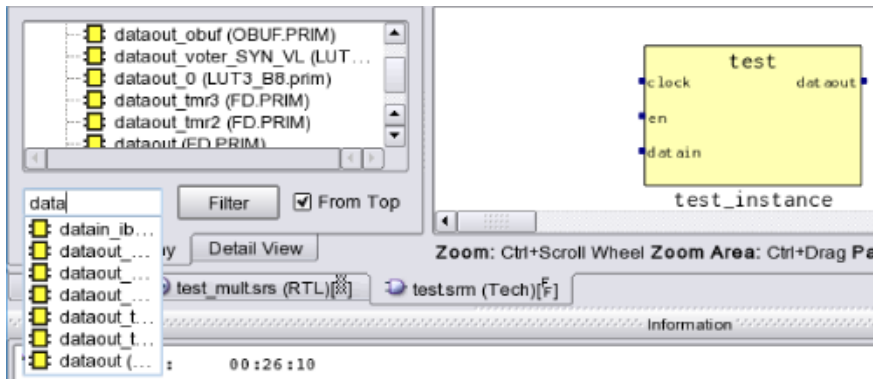
5. Open the instance by one of these methods:
 - Double-click the instance.
 - Right-click and select Open Instance View.



Browsing an Object by Filtering or Loading

1. To filter a particular instance, type the instance name into the text box available below the hierarchical browser pane as shown below.

The instance names become visible in the drop-down as you continue to type.

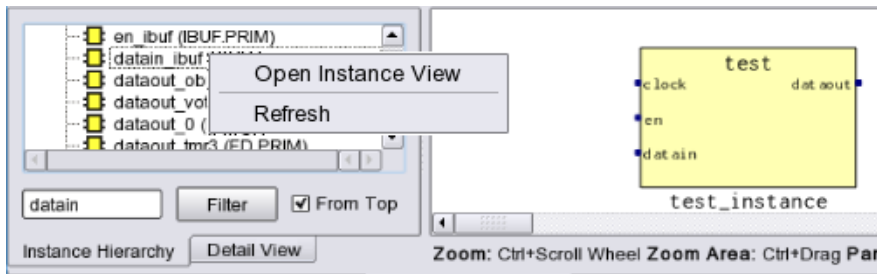


2. Click Filter and the selected hierarchical instance is highlighted in the hierarchical browser.
3. To search through a hierarchical path, select From Top.

If this is not checked, the command searches the entire design.

4. To load an instance from the Hierarchy Browser, right-click on the desired instance and select the Open Instance View option.

The instance is displayed in the RTL schematic view.



New HDL Analyst Limitations

Consider these limitations before using the new HDL Analyst flow:

- The expand operation works as intended, if the entire hierarchy is loaded from the top level. The expand operation will stop for unloaded hierarchies.
- The name of each object will be appended with the immediate hierarchy name, which was used to load the object.

Browsing With the Find Command

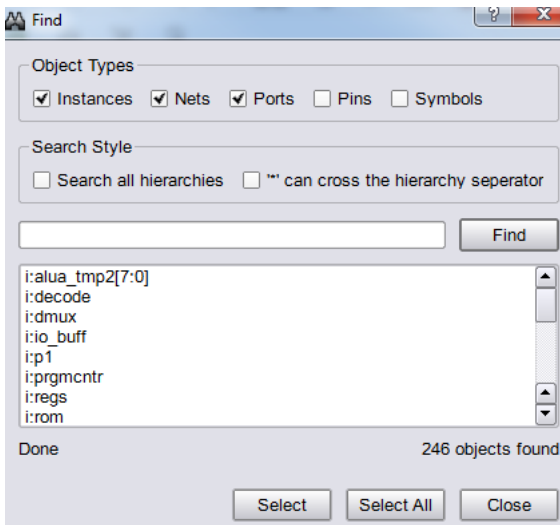
1. In a schematic, select the search icon (🔍) or press Ctrl-f to open the Find dialog box.
2. Do the following in the dialog box:
 - Select the type of objects to find: instances, nets, ports, pins, and/or symbols.
 - Specify how you want the search to occur: for all hierarchies and/or allow * to search across the hierarchy separator.

When searching, note the following:

- Double-click on a selected object from the dialog box to filter it in the schematic view.
- To search for multiple patterns, specify the objects with a single space between each selection in the search field.

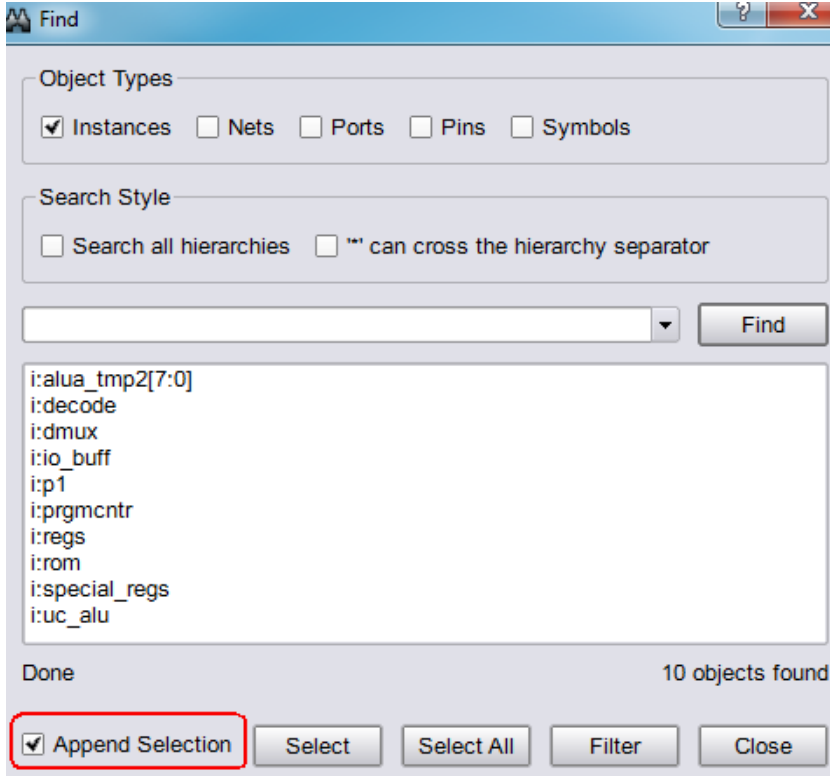
- Once multiple objects have been select from the dialog box, you can highlight them, then copy and paste them to the Tcl window or in a text file.
3. Select an object displayed in the dialog box below, then click the **Select** button. Click the **Filter** button, to select the specified objects and filter them in the HDL Analyst view.

Click the **Close** button to end the Find search. Then, you can use the **Filter** command to display the object.



When the search style options (Search all hierarchies and "",can cross the hierarchy separator) are not enabled, the software searches for objects at the top level.

4. If you enable the **Append Selection** option, objects selected in the current display window are appended to each other when you click the **Select** or **Select All** button. Otherwise, objects will be overridden after each selection.



5. When Append Selection is enabled, you can also search for multiple patterns, then filter them in the schematic view by clicking the Filter button.
6. If you determine that the search is taking too long to run, notice that the Find button changes to Stop. Click Stop.

The multi-threaded Find command can be interrupted and canceled once the search term has been identified. If you let the search complete, you will see Finishing and Done appearing under the display window.

Using Wildcards with the Find Command

Use the following wildcards when you search the schematics:

*	The asterisk matches any sequence of characters.
?	The question mark matches any single character, but not the hierarchy separator by default.
.	The dot explicitly matches a hierarchy separator, so type one dot for each level of hierarchy. To use the dot as a pattern and not a hierarchy separator, type a backslash before the dot: \.

Crossprobing

Crossprobing is the process of selecting an object in one view and having the object or the corresponding logic automatically highlighted in other views. Crossprobing helps you visualize where coding changes or timing constraints might help to reduce area or improve performance.

This section describes how to crossprobe from different views. It includes the following:

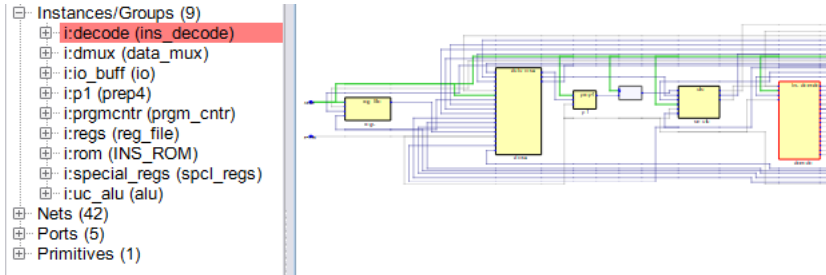
- [Crossprobing within a View, on page 264](#)
- [Crossprobing from an HDL Analyst View, on page 265](#)
- [Crossprobing from the Text Editor Window, on page 269](#)

Crossprobing within a View

Selecting an object name in the Hierarchy Browser highlights the object in the schematic, and vice versa.

Selected Object	Highlighted Object
Instance in schematic (single-click)	Module icon in Hierarchy Browser
Net in schematic	Net icon in Hierarchy Browser
Port in schematic	Port icon in Hierarchy Browser
Logic icon in Hierarchy Browser	Instance in schematic
Net icon in Hierarchy Browser	Net in schematic
Port icon in Hierarchy Browser	Port in schematic

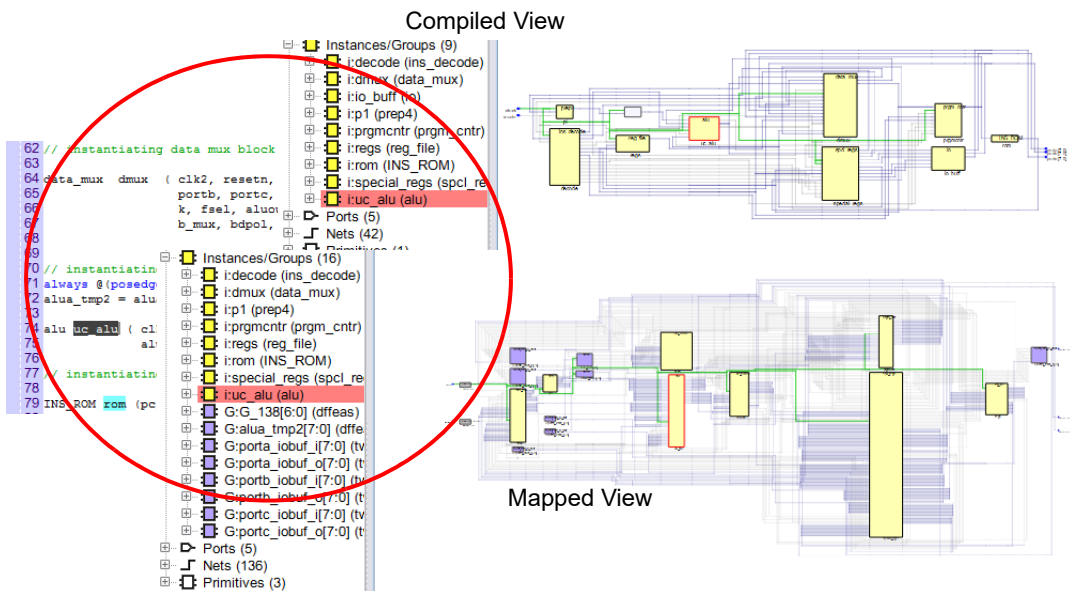
In this example, when you select the DECODE module in the Hierarchy Browser, the DECODE module is automatically selected in the view.



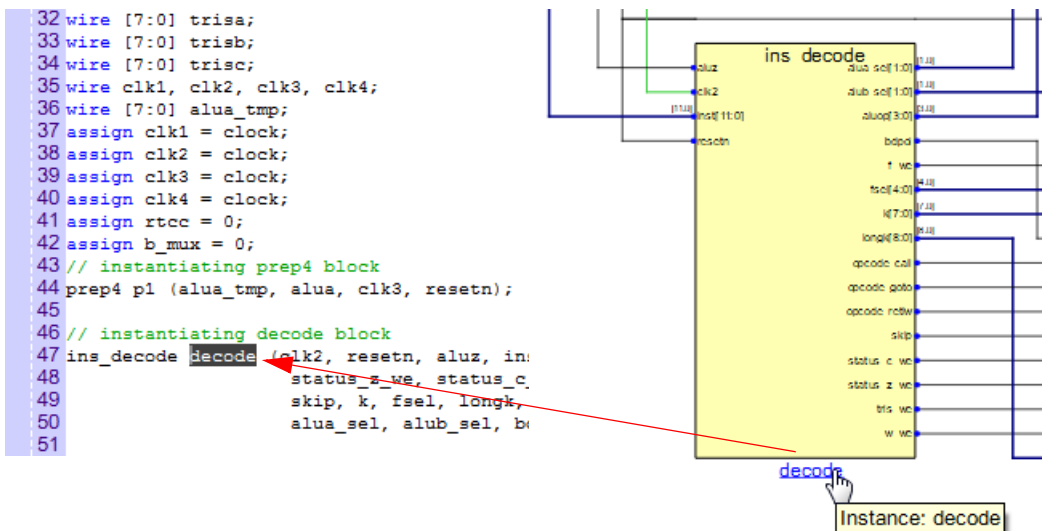
Crossprobing from an HDL Analyst View

To crossprobe from the schematic to other open views or the source code files, select the object by clicking on it.

The software automatically highlights the object in all open views. If the open view is a schematic, the software highlights the object in the Hierarchy Browser on the left as well as in the schematic. If the highlighted object is in another hierarchy of a schematic, the view does not automatically track to the page. You may have to filter the schematic.



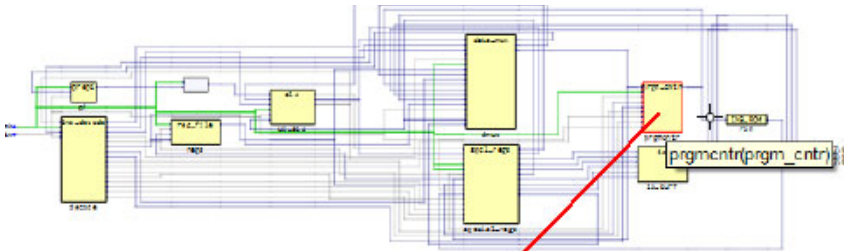
To crossprobe from the schematic to a source file when the source file is not open, the instance names must be the same. Notice that when you hover over an instance name in the schematic, it turns blue. You can click this link, to automatically open the editor window of the source code file and highlight the appropriate code as shown below. A message is generated if a match cannot be found.



Crossprobing to the Source Code

You can easily crossprobe instances or modules in the HDL Analyst view to the source code. To do this, choose either to:

- Highlight an instance in the HDL Analyst view, then right-click and select View Instance in Source from the drop-down menu. The tool automatically crossprobes to this instance in the source code.

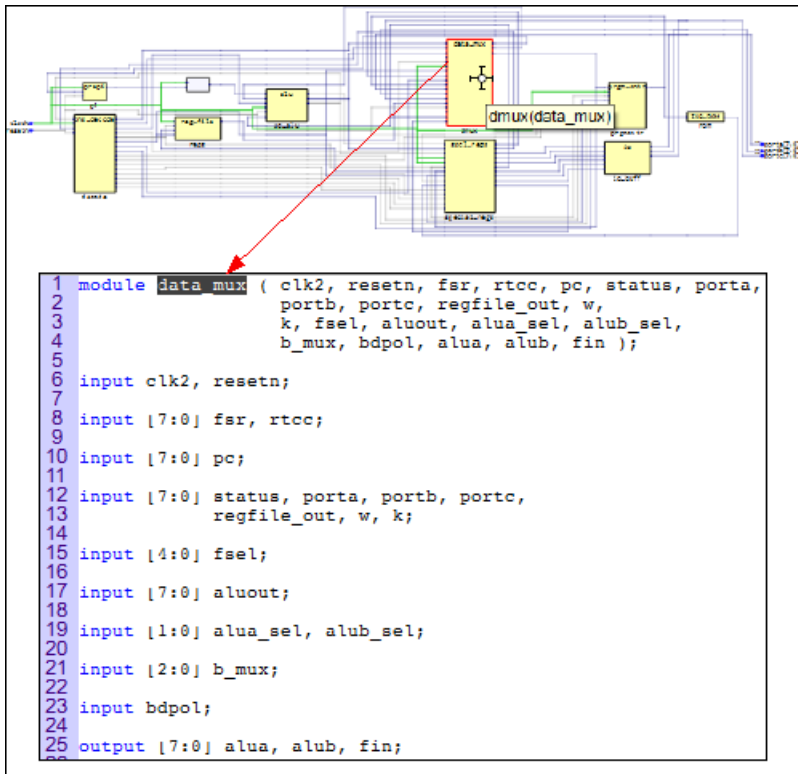


```

3 // instantiating prep4 block
4 prep4 p1 (alua_tmp, alua, clk3, resetn);
5
6 // instantiating decode block
7 ins_decode decode (clk2, resetn, aluz, inst, f_we, w_we,
8                   status_r_we, status_c_we, tris_we,
9                   skip, k, fsel, longk, aluop,
10                  alua_sel, alub_sel, bdpol, opcode_goto, opcode_c
11
12 // instantiating program counter block
13 prgm_ctr prgm_ctr ( clk4, resetn, f_we, longk, fsel,
14                   opcode_goto, opcode_call, opcode_retlw, fin, pc );
15
16 // instantiating regs block
17 reg_file regs (clk1, f_we, fsel, fin, regfile_out);
18
19 // instantiating data mux block
20
21 data_mux dmux ( clk2, resetn, fsr, rtcc, pc[7:0], status, porta,
22               portb, portc, regfile_out, w,
23               k, fsel, aluout, alua_sel, alub_sel,
24               b_mux, bdpol, alua, alub, fin );

```

- Highlight a module in the HDL Analyst view, then right-click and select View Module in Source from the drop-down menu. The tool automatically crossprobes to this module in the source code.



Note that you can crossprobe to instances and modules in the source code from the Hierarchy Browser as well. Highlight an object, then right-click and select View Instance in Source or View Module in Source from the drop-down menu.

Crossprobing from the Text Editor Window

To crossprobe source in the text editor window or from the log file to a schematic, use this procedure. You can use this method to crossprobe from any text file with objects that have the same instance names as in the synthesis software.

1. Open the schematic to which you want to crossprobe.
2. Select the appropriate portion of text in the Text Editor window. In some cases, it may be necessary to select an entire block of text to crossprobe.
3. You can choose either to:
 - Highlight the objects in the path, right-click and select **Filter** in Analyst from the drop-down menu. The tool automatically filters the schematic so that you see just the selected objects in the view.
 - Highlight the objects in the path, right-click and select **Select** in Analyst from the drop-down menu. You might have to filter the selected objects to see them displayed in the schematic.

For example:

Text Editor

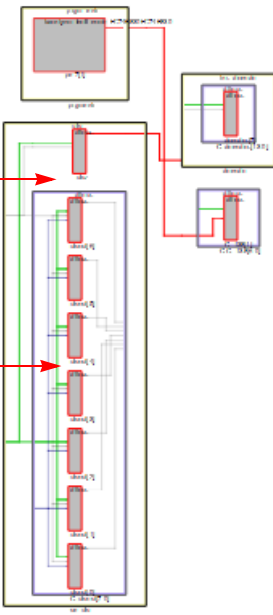
635 *****
636
637
638 Instance Starting
639 Reference
640 Clock

641 uc_alu.aluout[2] eight_bit_uc|clock dffeas d
642 uc_alu.aluout[1] eight_bit_uc|clock dffeas d
643 uc_alu.aluout[3] eight_bit_uc|clock dffeas d
644 uc_alu.aluout[0] eight_bit_uc|clock dffeas d
645 uc_alu.aluout[4] eight_bit_uc|clock dffeas d
646 uc_alu.aluout[5] eight_bit_uc|clock dffeas d
647 uc_alu.aluout[6] eight_bit_uc|clock dffeas d
648 decode.decodes[7] eight_bit_uc|clock dffeas d
649 uc_alu.aluout[1] eight_bit_uc|clock dffeas d
650
651
652
653
654
655 Worst Path Information
656 *****
657
658
659 Path information for path number 1
660 Requested Period:
661 - Setup time:
662 + Intrinsic clock delay:
663 + Clock delay at ending point:
664 = Required time:
665

Undo Ctrl+Z
Redo Ctrl+Y
Cut Ctrl+X
Copy Ctrl+C
Paste Ctrl+V
Select All Ctrl+A
Toggle bookmark Ctrl+F2
Next bookmark F2
Previous bookmark Shift+F2
Refresh Ctrl+U
Filter in Analyst
Select in Analyst
Select From...

Schematic View

Schematic View (Filtered)



Analyzing With the HDL Analyst Tool

The HDL Analyst tool is a graphical productivity tool that helps you visualize your synthesis results. It displays schematics of the design at different stages, allowing you to graphically view and analyze your design. At an early design stage, the schematic displays high-level structures like RAM, ROM, operators, and FSM as abstractions. Later in the cycle, these structures are converted to gates and mapped to technology-specific resources.

To analyze information or compare views with the log file, the FSM view, and the source code, you can use techniques like crossprobing, flattening, and filtering. See the following for more information about analysis techniques.

- [Viewing Design Hierarchy and Context](#), on page 271
- [Filtering Schematics](#), on page 275
- [Expanding Pin and Net Logic](#), on page 277
- [Dissolving and Partial Dissolving of Buses and Pins](#), on page 283
- [Flattening Schematic Hierarchy](#), on page 285
- [Using the FSM Viewer](#), on page 288

For additional information about navigating the HDL Analyst views or using other techniques like crossprobing, see the following:

- [Working in the Schematic](#), on page 232
- [Exploring Design Hierarchy](#), on page 247
- [Finding Objects](#), on page 255
- [Crossprobing](#), on page 264

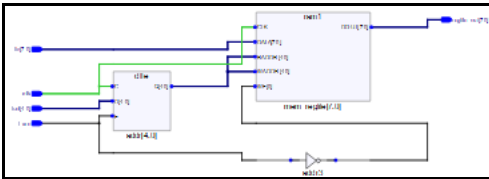
Viewing Design Hierarchy and Context

Most large designs are hierarchical, so the software provides tools that help you view hierarchy details or put the details in context. Alternatively, you can browse and navigate hierarchy with the Push/Pop command, or flatten the design to view internal hierarchy.

This section describes how to use interactive hierarchical viewing operations to better analyze your design. Automatic hierarchy viewing operations that are built into other commands are described in the context in which they appear.

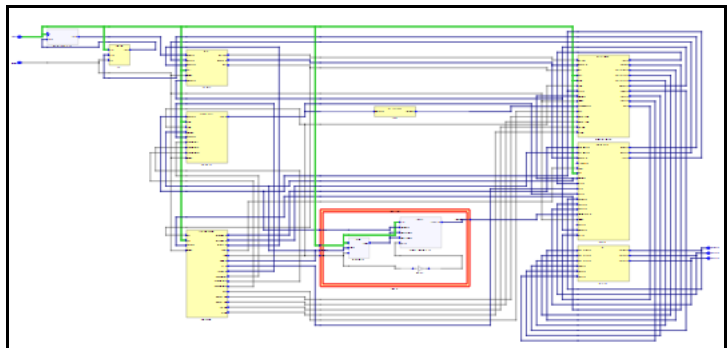
1. To view the internal logic of instances in your design, do either of the following:
 - To view the logic of an individual instance, push into it. This generates a new schematic with the internal details. Click the Back icon to return to the previous view.
 - To view the logic of all instances in the design, select all the required instances and right-click then select Peek. This command lets you see internal logic in content, by adding the internal details to the current schematic. If the view is too cluttered with this option on, filter the view (see [Filtering Schematics, on page 275](#)) or push into the primitive. Click the Back icon to return to the previous view after filtering or pushing into the object.

The following figure compares these two methods:



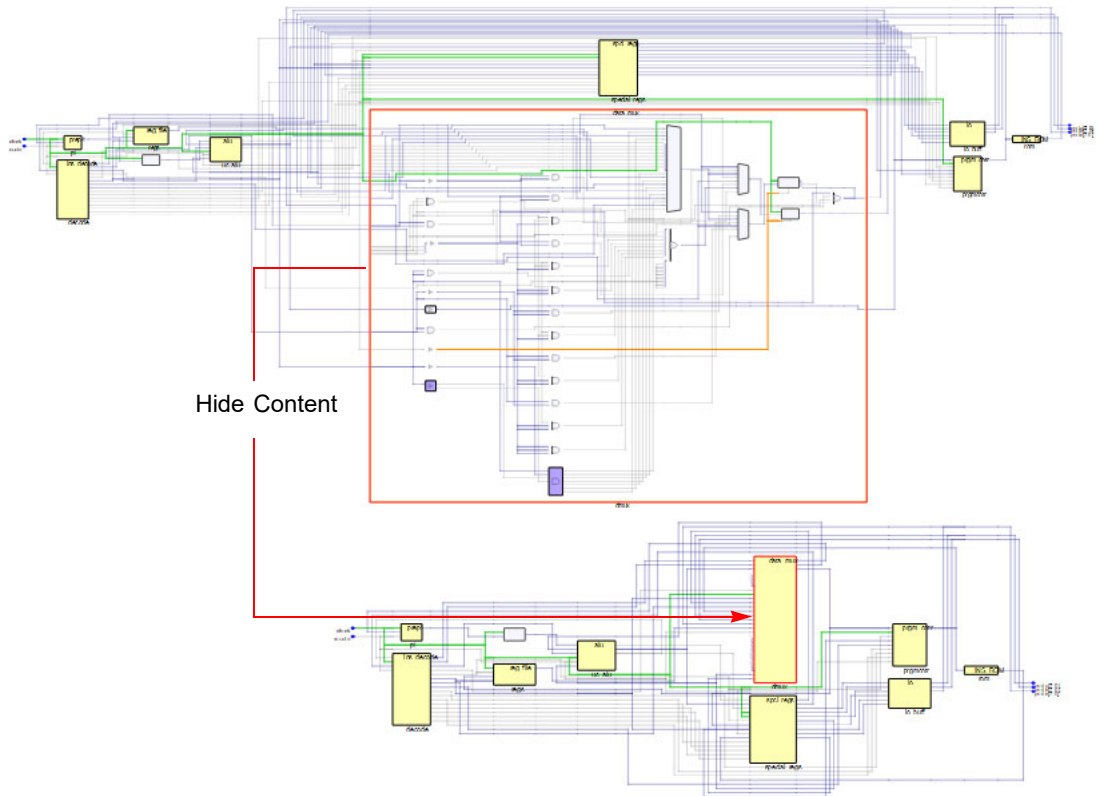
Result of pushing into a primitive (new view of lower-level logic)

Result of enabling Peek
(same view with internal logic)



The technology primitives that are contained in a block are displayed automatically when you zoom in on them in the schematic view.

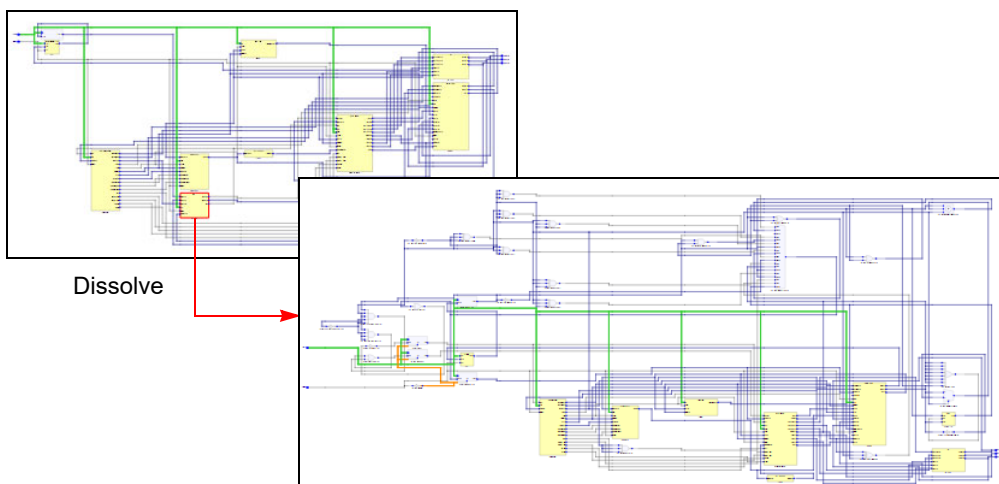
2. Suppose you just used the peek option to see the internal logic of an instance. To return back to the schematic state before using peek and while the peek objects are still highlighted, right-click and select Hide Contents from the drop-down menu.



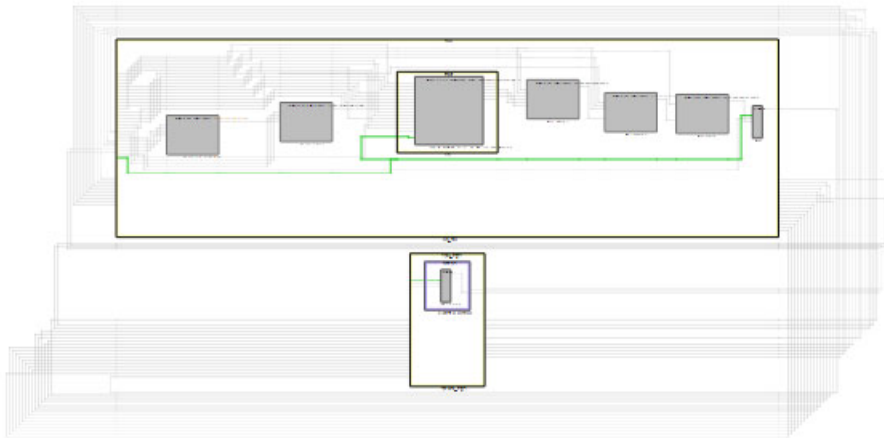
3. To view the internal logic of a hierarchical instance, you can push into the instance, dissolve the selected instance with the Dissolve command, or flatten the design.

Pushing into an instance	Generates a view that shows only the internal logic. You do not see the internal hierarchy in context. To return to the previous view, click Back. See Exploring Object Hierarchy with Push/Pop Commands , on page 250 for details.
Flattening the entire design	Opens a view where the entire design is flattened. Large flattened designs can be overwhelming. See Flattening Schematic Hierarchy , on page 285 for details about flattening designs.
Flattening an instance by dissolving	Generates a view where the hierarchy of the selected instances is flattened, but the rest of the design is unaffected. This provides context. See Flattening Schematic Hierarchy , on page 285 for details about dissolving instances.

The following schematic shows an instance that has been dissolved in the view.



4. The software automatically traces a critical path through different hierarchical levels using hollow boxes with nested internal logic (transparent instances) to indicate levels in hierarchical instances.



Filtering Schematics

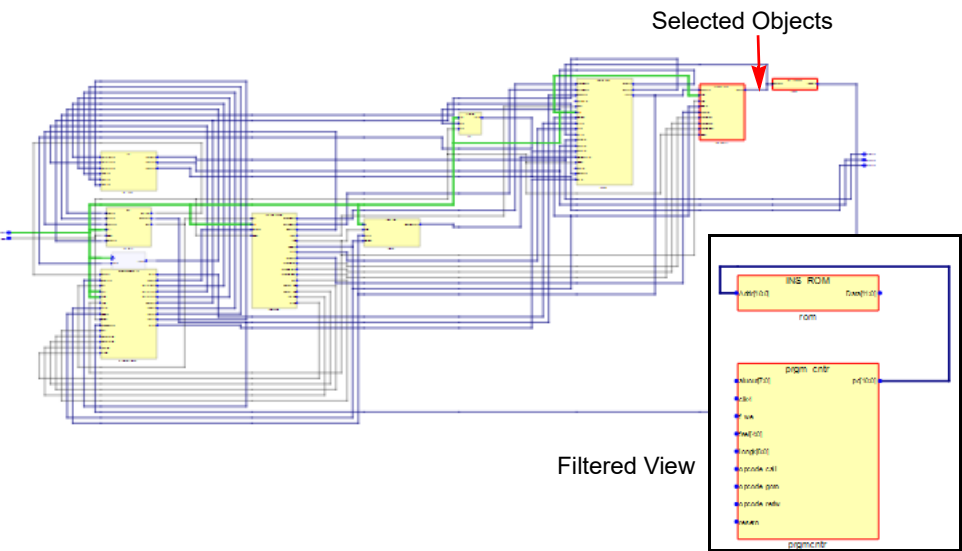
Filtering is a useful first step in analysis, because it focuses analysis on the relevant parts of the design. Some commands, like the Expand commands, automatically generate filtered views; this procedure only discusses manual filtering, where you use the Filter command to isolate selected objects.

This table lists the advantages of using filtering over flattening:

Filter Schematic Command	Flatten Commands
Loads part of the design; better memory usage	Loads entire design
Combine filtering with the Push/Pop command, and history buttons (Back and Forward) to move freely between hierarchical levels	You can use the Back arrow or Show Top View icon to return to previous view that has been flattened.

1. Select the objects that you want to isolate. For example, you can select two connected objects.
2. Select the Filter command, using one of these methods:
 - Right-click and select Filter from the popup menu.
 - Click the Filter icon (buffer gate) (.

The software filters the design and displays the selected objects in a filtered view. These objects are isolated in the schematic displayed. Select Unfilter to take you back to the view where objects are at the same level.



You can now analyze the problem, and do operations like the following:

Trace paths, build up logic	See Expanding Pin and Net Logic , on page 277
Filter further	Select objects and filter again
Find objects	See Finding Objects , on page 255
Flatten	See Flattening Schematic Hierarchy , on page 285. You can hide transparent or opaque instances.
Crossprobe from filtered view	See Crossprobing from an HDL Analyst View , on page 265

- To return to the previous schematic, click the Back arrow. If you flattened the hierarchy, right-click and select the Back arrow or the Show Top View icon to return to the top-level unflattened view.

For additional information about filtering schematics, see [Filtering Schematics](#), on page 275 and [Flattening Schematic Hierarchy](#), on page 285.

Expanding Pin and Net Logic

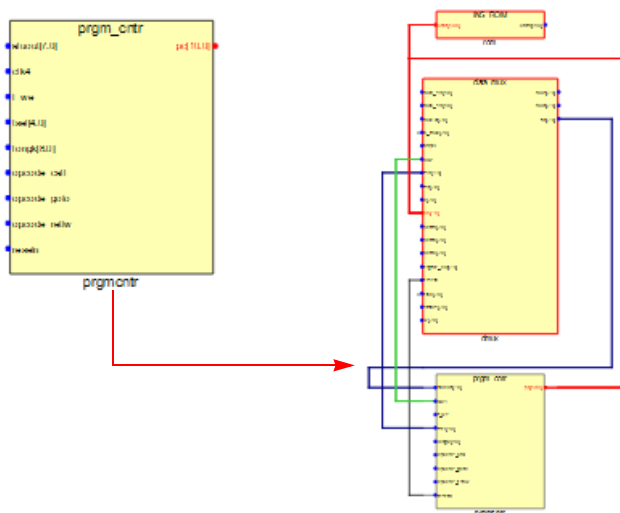
When you are working in a filtered view, you might need to include more logic in your selected set to debug your design.

Use the Expand commands with the Filter and Flatten commands to isolate just the logic that you want to examine. Filtering isolates logic, flattening removes hierarchy. See [Filtering Schematics, on page 275](#) and [Flattening Schematic Hierarchy, on page 285](#) for details.

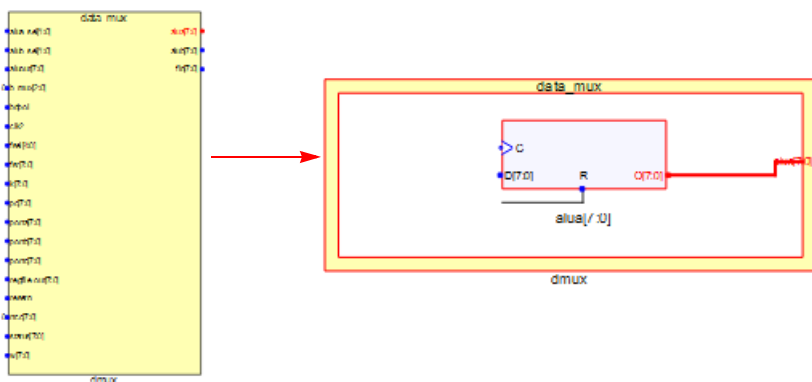
1. To expand logic from a pin hierarchically across boundaries, use the following commands.

To ...	Do this ...
See the first-level cells connected to a pin in the same hierarchy	Select a pin and select Expand. See Expanding Filtered Logic Example , on page 278 .
See the first-level cells connected to a pin at any level of hierarchy	Select a pin and select Hierarchical Expand Nets.
See all cells until a register or port is connected to the selected pin at the same level of hierarchy	Select a pin and select Expand to Reg/Port.
See all cells until a register or port is connected to the selected pin at any level of hierarchy	Select a pin and select Hierarchical Expand to Reg/Port.
See internal cells connected to a pin	Select a pin and select Expand Inwards. The software filters the schematic and displays the internal cells closest to the port. See Expanding Inwards Example , on page 278 .

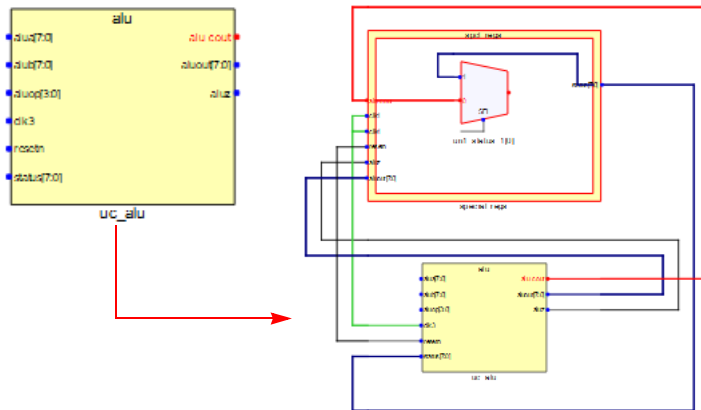
Expanding Filtered Logic Example



Expanding Inwards Example



Expanding Hierarchically



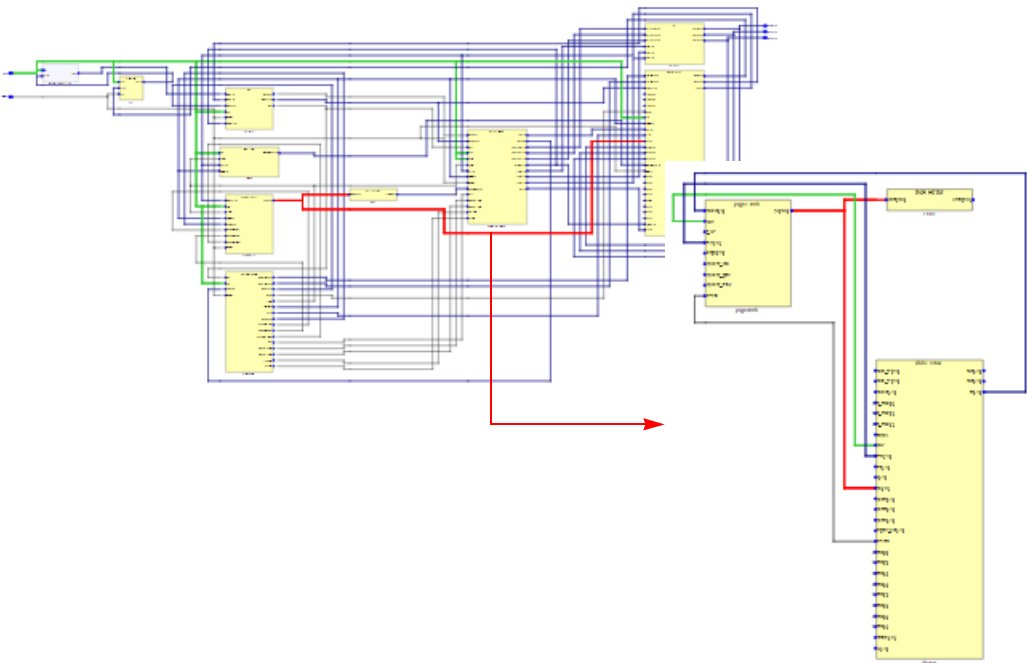
2. To expand logic from a net, do the following:

- Use the commands shown in the following table.
- Select a net, then right-click and select the command from the right-click options.

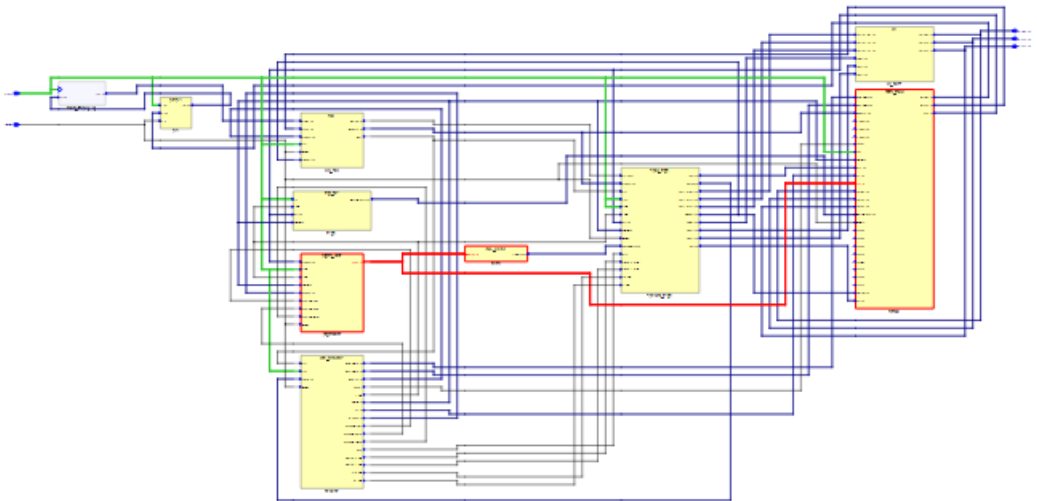
To ...	Do this ...
See all instances connected to the selected net being filtered	Select a net and select Filter by Nets.
Select the instances in the same hierarchy connected to the selected net	Select a net and select Expand Nets.
Select only one object connected to a port or pin	Select a pin or port and select Expand to One Object.
Select and show instances connected to the selected net at any level of hierarchy. The instance that drives the net and the instance which is driven by the net are shown.	Select a net and select Hierarchical Expand Nets.

The following figures illustrate this.

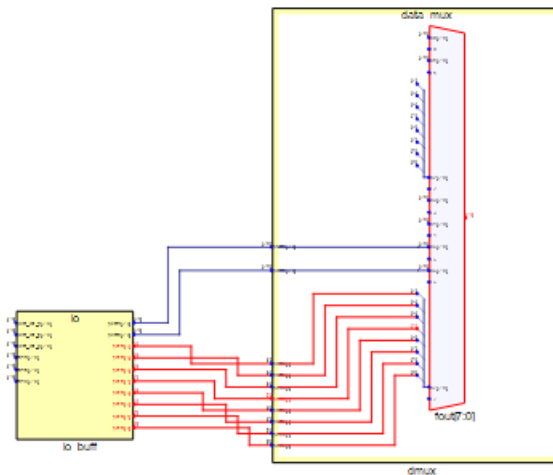
Filter by Nets



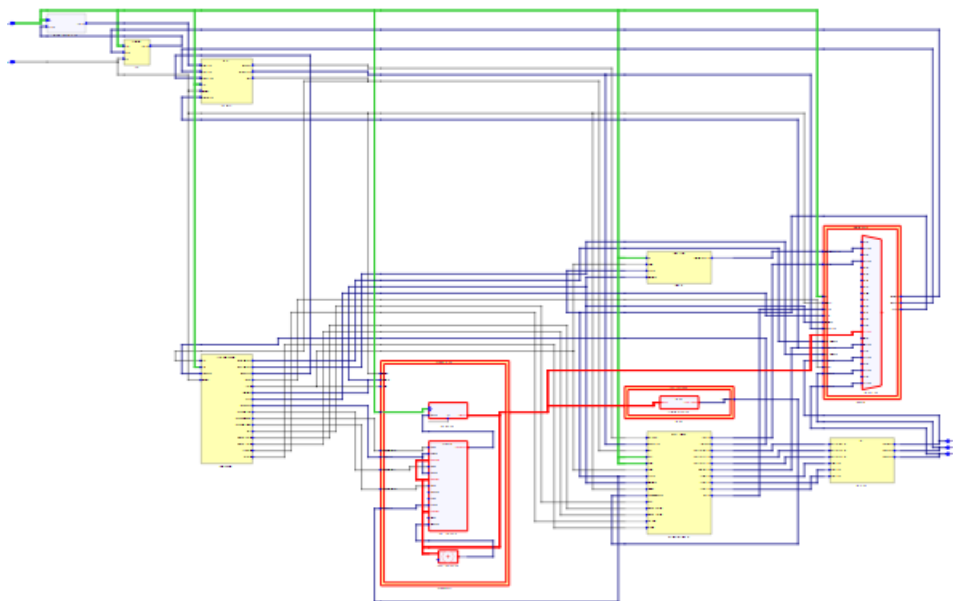
Expand Nets



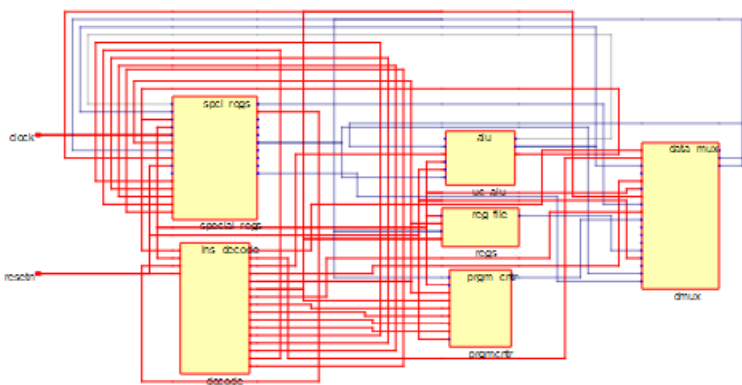
Expand to One Object



Hierarchical Expand Nets



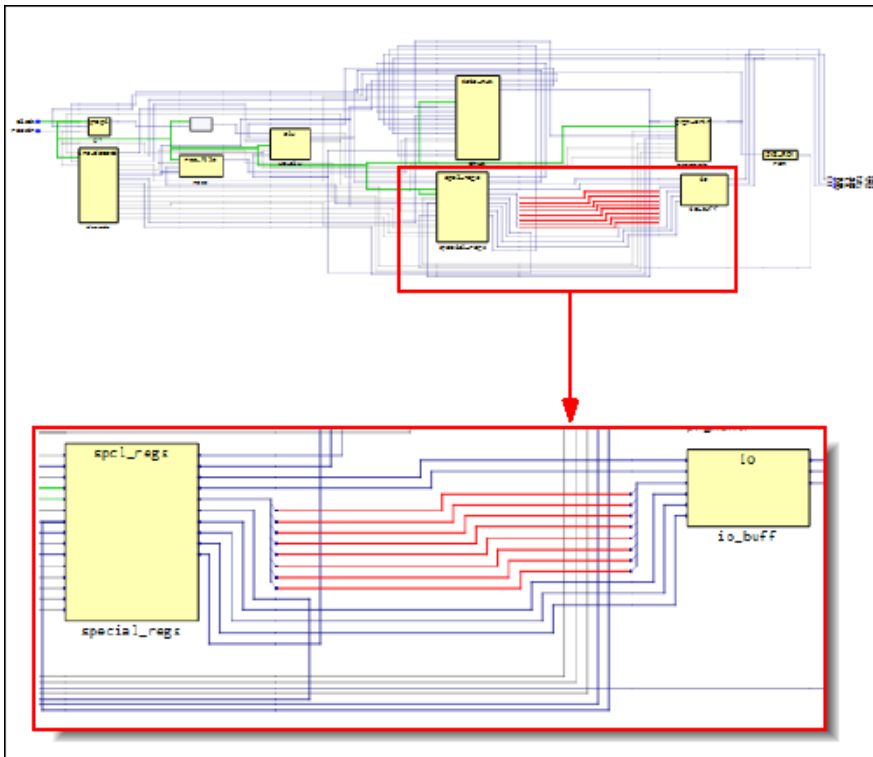
3. You can also isolate the paths to generate a schematic for a path between objects. To display connections to and from the selected instance, highlight it then right-click and select **Isolate Paths** from the drop-down menu.



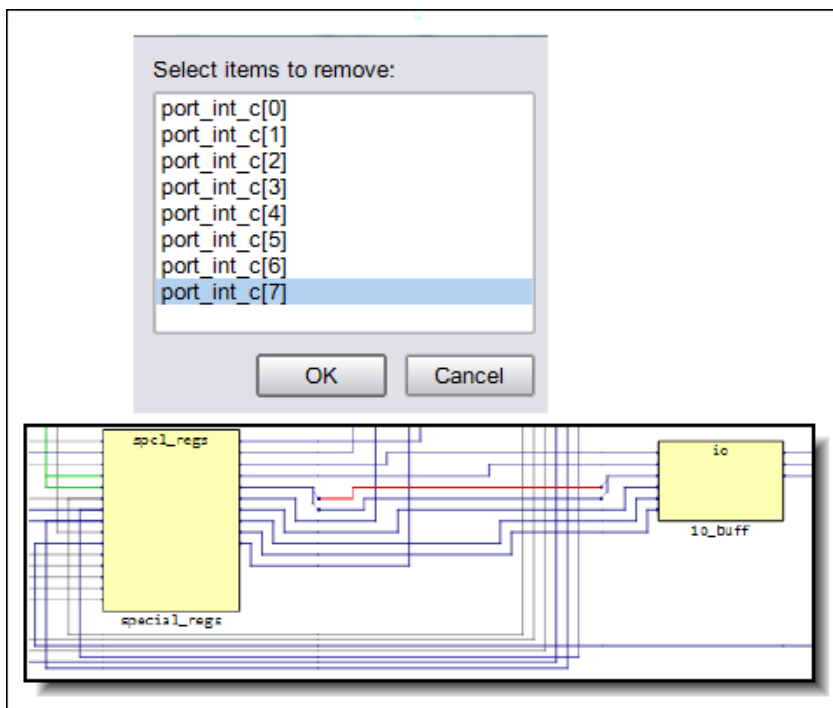
Dissolving and Partial Dissolving of Buses and Pins

The HDL Analyst tool has options for handling buses and pins in the display that can help you analyze your design easier. You can expand logic from a bus port or specific bits of a port.

1. To expand logic from a bus port to its bit ports:
 - Select a net.
 - Right-click and select Dissolve from the drop-down menu.

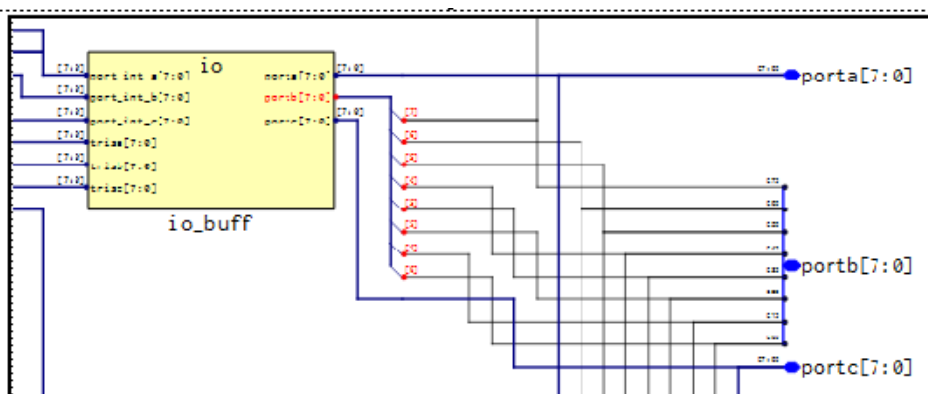


2. To expand logic from specific bits of the port:
 - Select a net.
 - Right-click and select Partial Dissolve from the drop-down menu.
 - Select the pin (port_int_c[7]) to be expanded.
 - Click OK.

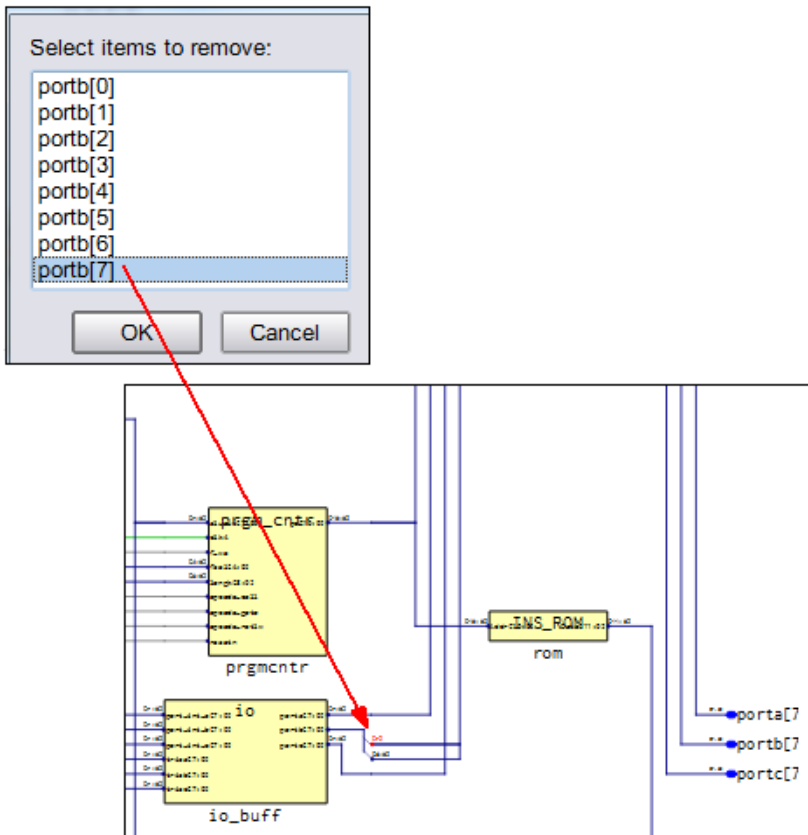


3. To expand logic from a bus port:

- Select a pin.
- Right-click and select Dissolve from the drop-down menu.



4. To expand logic from specific bits of a port:
 - Select a pin.
 - Right-click and select Partial Dissolve Pin from the drop-down menu.
 - Select the pin to be expanded.
 - Click OK.



Flattening Schematic Hierarchy

Flattening removes hierarchy so you can view the logic without hierarchical levels. In most cases, you do not have to flatten your hierarchical schematic to debug and analyze your design, because you can use a combination of

filtering and expanding to view logic at different levels. However, if you must flatten the design use the following techniques, which include flattening and dissolving instances.

1. To flatten any level of hierarchy to logic cells below the current level, right-click and select Flatten Schematic from the drop-down menu.

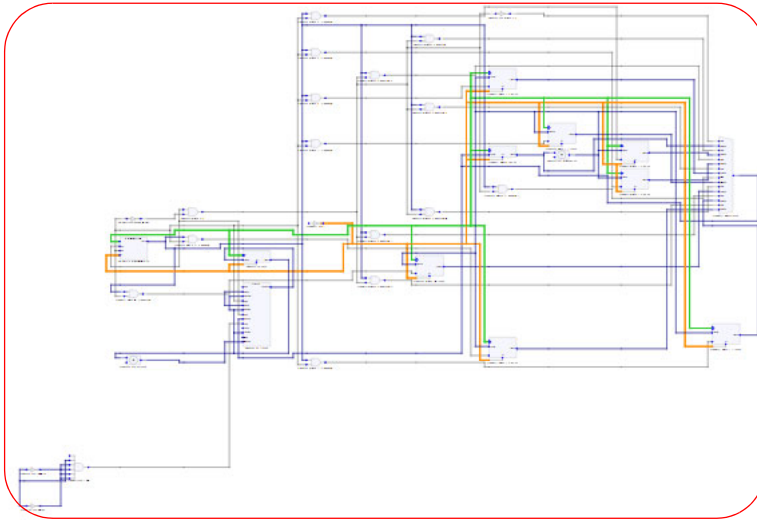
The software flattens the design hierarchy and displays it in the window. To return to the previous level, select the Back arrow.
2. To selectively flatten some hierarchical instances in your design by dissolving them, do the following:
 - Select the instances to be flattened.
 - Right-click and select Dissolve.

The results differ slightly, depending on the kind of view from which you dissolve instances.

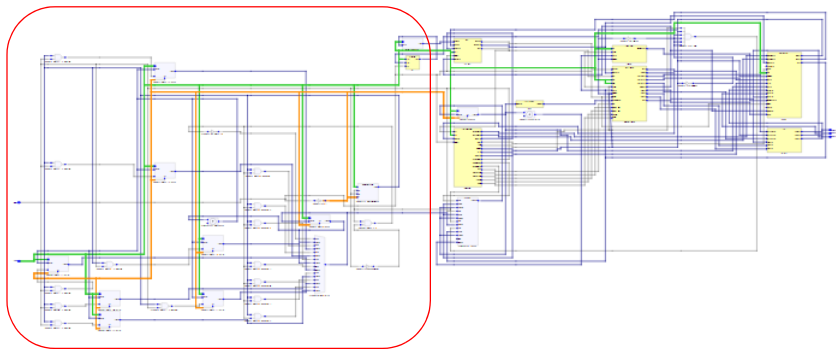
Starting View	Software Generates a ...
Filtered	Filtered view with the internal logic of dissolved instances displayed within hollow bounding boxes (transparent instances), and the hierarchy of the rest of the design unchanged. If the transparent instance does not display internal logic, use one of the alternatives described in step 4 of Viewing Design Hierarchy and Context , on page 271. Use the Back button to return to the undissolved view.
Unfiltered	New, flattened view with the dissolved instances flattened in place (no nesting) to Boolean logic, and the hierarchy of the rest of the design unchanged. You can use the Back button to return to previous or the top-level views.

The following figure illustrates this.

Dissolved logic for prgmcntr shown filtered when started from filtered view



Dissolved logic for prgmcntr shown flattened in context when you start from an unfiltered view



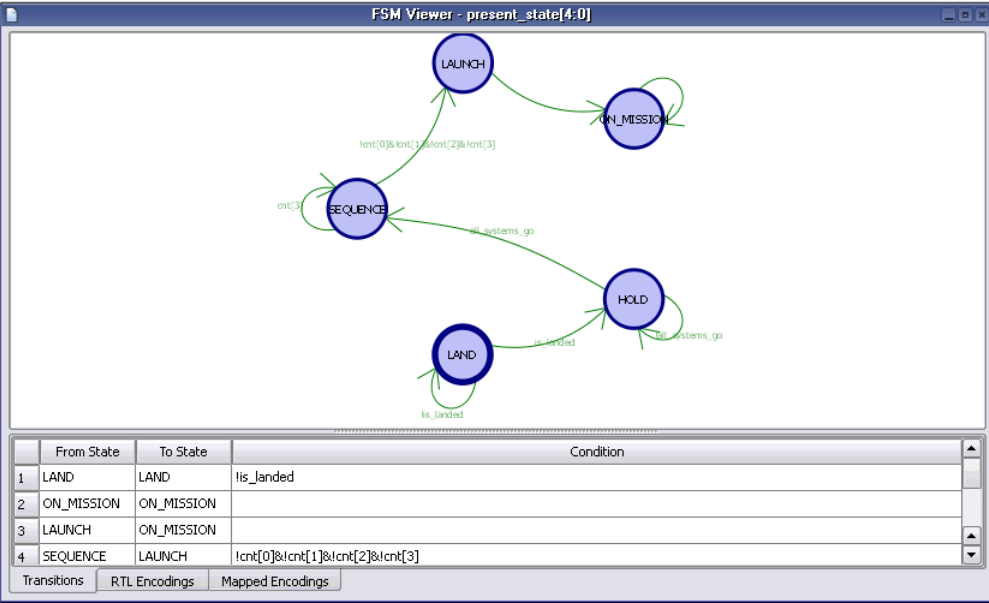
Use this technique if you only want to flatten part of your design while retaining the hierarchical context. If you want to flatten most of the design, use the technique described in the previous step. Instead of dissolving instances, you can use a combination of the filtering commands and the Push/Pop command.

Using the FSM Viewer

The FSM viewer displays state transition bubble diagrams for FSMs in the design, along with additional information about the FSM. You can use this viewer to view state machines.

1. To start the FSM viewer, open the compiled view and highlight the FSM instance, click the right mouse button and select View State Machine from the popup menu.

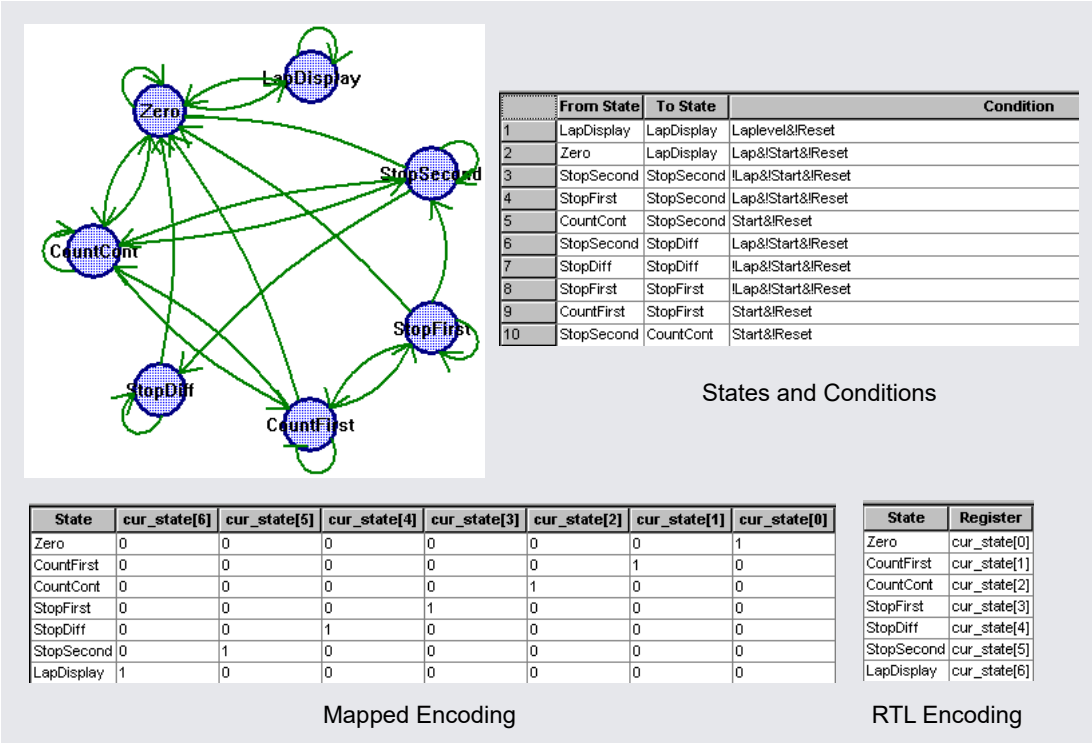
The FSM viewer opens. The viewer consists of a transition bubble diagram and a table for the encodings and transitions. If you used Verilog to define the FSMs, the viewer displays binary values for the state machines if you defined them with the 'define keyword, and actual names if you used the parameter keyword.



2. The following table summarizes basic viewing operations.

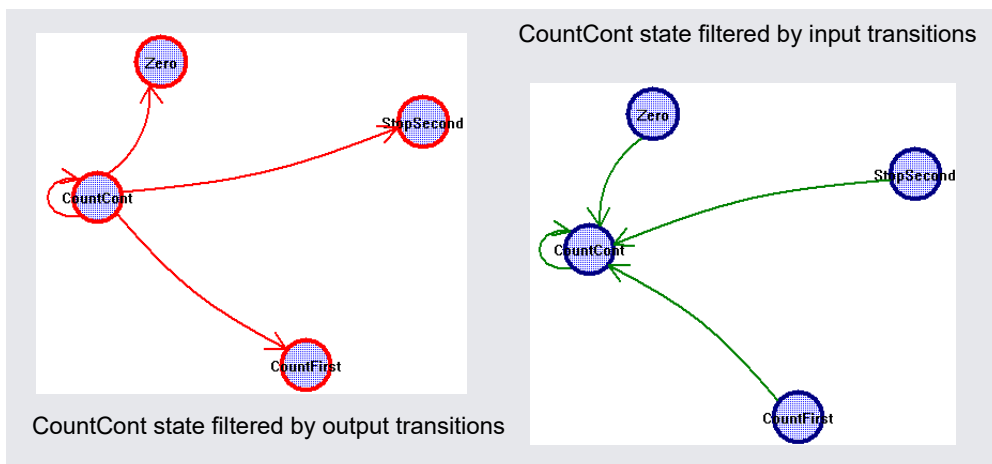
To view...	Do...
from and to states, and conditions for each transition	Click the Transitions tab at the bottom of the table.
the correspondence between the states and the FSM registers in the RTL view	Click the RTL Encoding tab.
the correspondence between the states and the registers in the Technology View	Click the Mapped Encodings tab (available after synthesis).
only the transition diagram without the table	Select View->FSM table or click the FSM Table icon. You might have to scroll to the right to see it.

This figure shows you the mapping information for a state machine. The Transitions tab shows you simple equations for conditions for each state. The RTL Encodings tab has a **State** column that shows the state names in the source code, and a **Registers** column for the corresponding RTL encoding. The Mapped Encoding tab shows the state names in the code mapped to actual values.



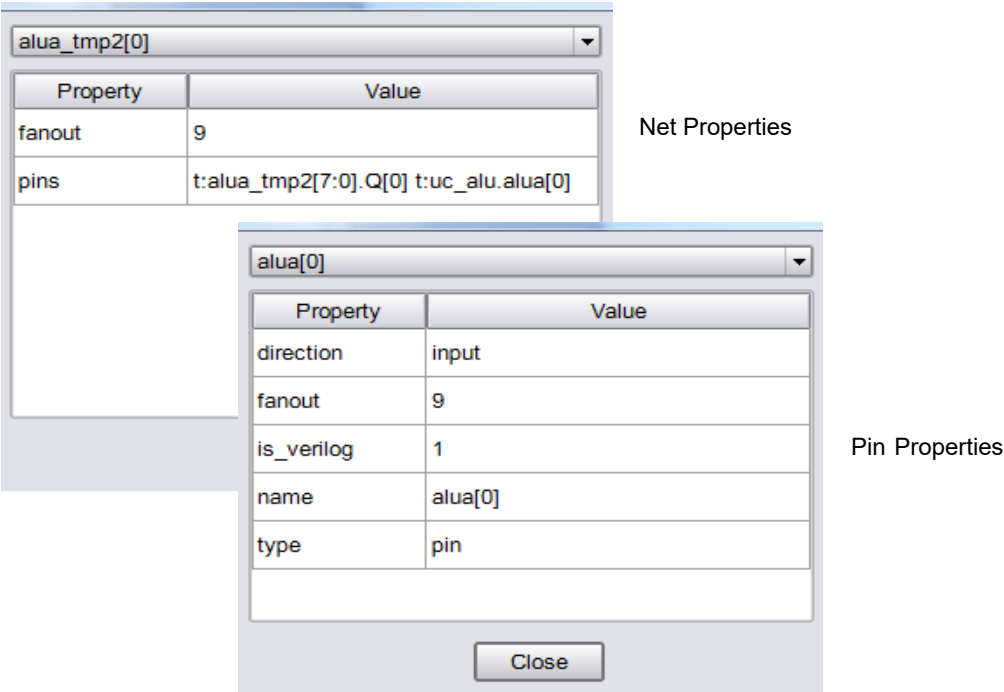
3. To view just one selected state,
- Select the state by clicking on its bubble. The state is highlighted.
 - Click the right mouse button and select the filtering criteria from the popup menu: output, input, or any transition.

The transition diagram now shows only the filtered states you set. The following figure shows filtered views for output and input transitions for one state.



Similarly, you can check the relationship between two or more states by selecting the states, filtering them, and checking their properties.

4. To view the properties for a state:



- Select the state.
- Click the right mouse button and select Properties from the popup menu. A form shows you the properties for that state.

To view the properties for the entire state machine like encoding style, number of states, and total number of transitions between states, deselect any selected states, click the right mouse button outside the diagram area, and select Properties from the popup menu.

Working in the Standard Schematic

The HDL Analyst includes the RTL and Technology views, which are schematic used to graphically analyze your design. In the Synplify product, these views are part of the optional HDL Analyst package. The RTL view is available after a design is compiled; the Technology view is available after a designed has been synthesized and contains technology-specific primitives.

The HDL Analyst includes the RTL and Technology views, which are schematics used to graphically analyze your design. In the Synplify product, these views are part of the optional HDL Analyst package. The RTL view is available after a design is compiled; the Technology view is available after a designed has been synthesized and contains technology-specific primitives.

For detailed descriptions of these views, see the *HDL Analyst Tool* section of the *Reference Manual*. This section describes basic procedures you use in the RTL and Technology views. The information is organized into these topics:

- [Opening the Views](#), on page 233
- [Opening the Views](#), on page 233
- [Viewing Object Properties](#), on page 236
- [Selecting Objects in the RTL/Technology Views](#), on page 300
- [Working with Multisheet Schematics](#), on page 301
- [Moving Between Views in a Schematic Window](#), on page 303
- [Setting Schematic Preferences](#), on page 304
- [Managing Windows](#), on page 305

For information on specific tasks like analyzing critical paths, see the following sections:

- [Exploring Object Hierarchy by Pushing/Popping](#), on page 308
- [Exploring Object Hierarchy of Transparent Instances](#), on page 314
- [Browsing to Find Objects in HDL Analyst Views](#), on page 315
- [Crossprobing \(Standard\)](#), on page 328
- [Analyzing With the Standard HDL Analyst Tool](#), on page 336

Differentiating Between the HDL Analyst Views

RTL View	Technology View
Generated after compilation.	Generated after mapping.
Technology-independent components at a high level of abstraction, like adders, registers, large muxes, and state machines.	Technology-specific primitives like look-up tables, cascade and carry chains, muxes and flip-flops.
srs database (Synopsys proprietary).	srm database (Synopsys proprietary).

Opening the Views

The procedure for opening an RTL or Technology view is similar; the main difference is the design stage at which these views are available.

1. Start at the appropriate design stage:

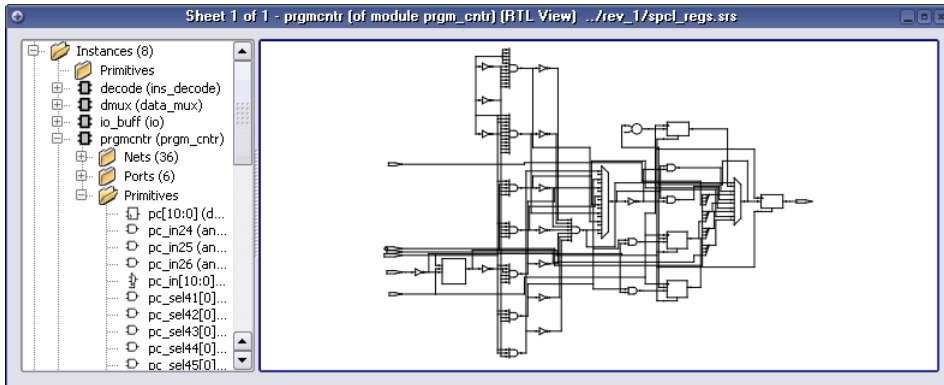
RTL view	Start with a compiled design.
Technology view	Start with a mapped (synthesized) design.

2. Open the view as described in this table:

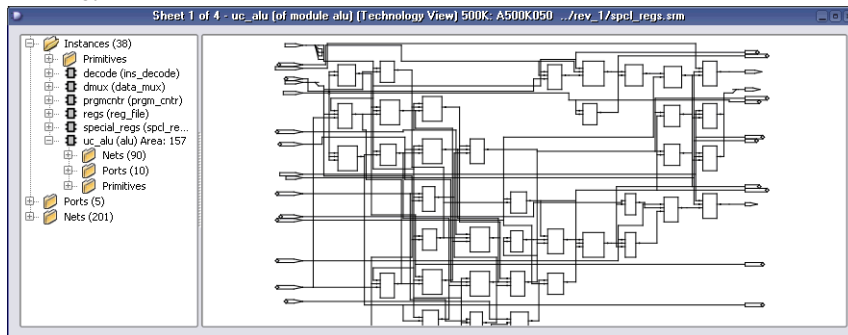
Hierarchical RTL or Technology view	Use one of these methods: • Select HDL Analyst->RTL->Hierarchical View. • Click the RTL View icon ( (a plus sign inside a circle)). • Double-click the <code>vm</code> file in the Implementation Results view. To open a flattened RTL view, select HDL Analyst->RTL->Flattened View.
Hierarchical Technology view	Use one of these methods: • Select HDL Analyst ->Technology->Hierarchical View. • Click the Technology View icon (NAND gate icon ). • Double-click the <code>srm</code> file in the Implementation Results view.
Flattened RTL or Technology view	Select HDL Analyst->RTL->Flattened View or HDL Analyst->Technology->Flattened View

All RTL and Technology views have the schematic on the right and a pane on the left that contains a hierarchical list of the objects in the design. This pane is called the Hierarchy Browser. The bar at the top of contains additional information. See [Hierarchy Browser, on page 46](#) in the *Reference Manual* for a description of the Hierarchy Browser.

RTL View



Technology View



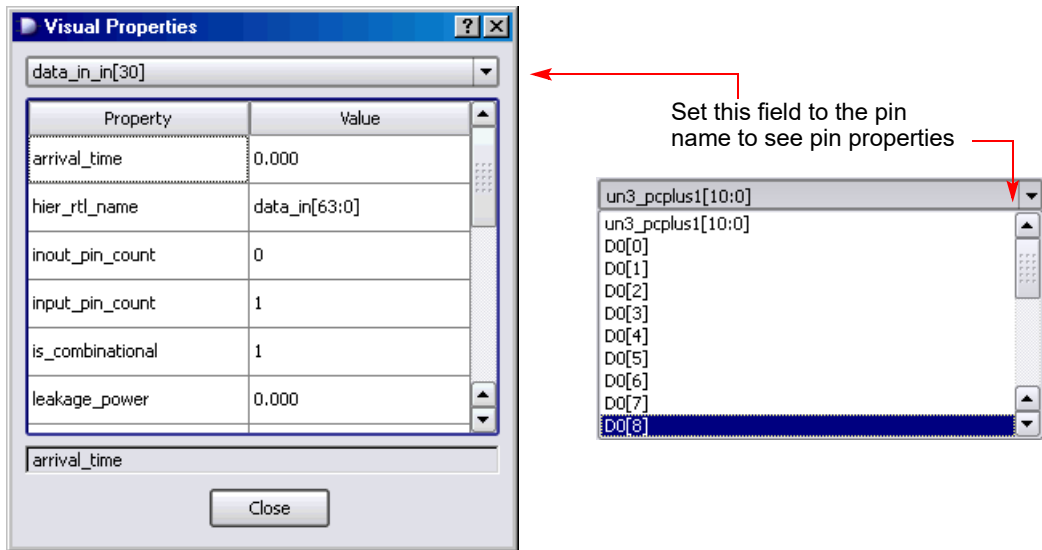
Viewing Object Properties

There are a few ways in which you can view the properties of objects.

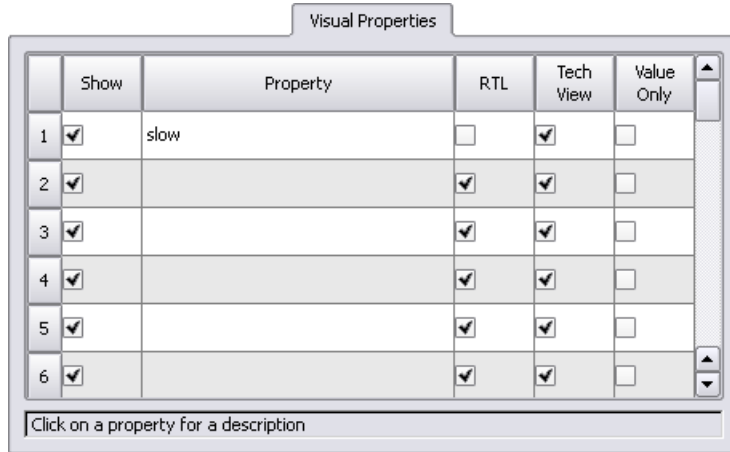
1. To temporarily display the properties of a particular object, hold the cursor over the object. A tooltip temporarily displays the information. at the cursor and in the status bar at the bottom of the tool window.

2. Select the object, right-click, and select Properties. The properties and their values are displayed in a table.

If you select an instance, you can view the properties of the associated pins by selecting the pin from the list. Similarly, if you select a port, you can view the properties on individual bits.



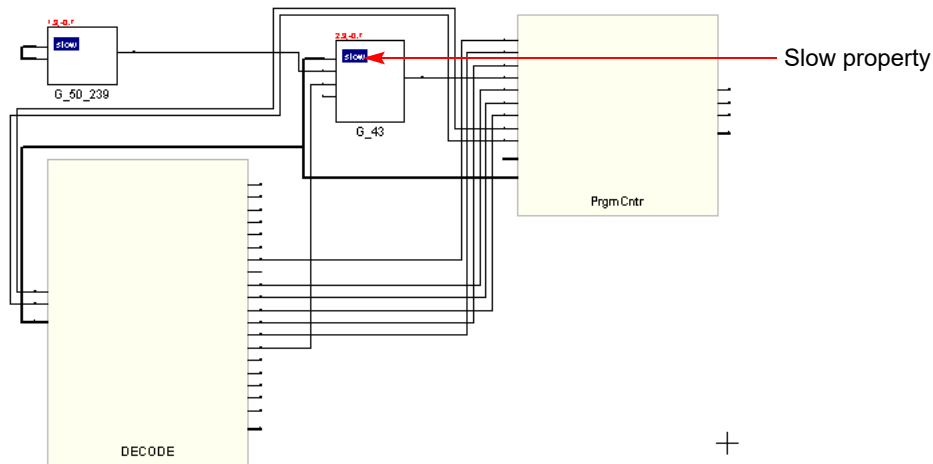
3. To flag objects by property, follow these steps:
 - Open an RTL or Technology view.
 - Select Options->HDL Analyst Options->Visual Properties, and select the properties you want to view from the pull-down list. Some properties are only available in certain views.



- Close the HDL Analyst Options dialog box.
- Enable View->Visual Properties. If you do not enable this, the software does not display the property flags in the schematics. The tool uses a rectangular flag with the property name and value to annotate all objects in the current view that have the specified property. Different properties use different colors, so you can enable and view many properties at the same time.

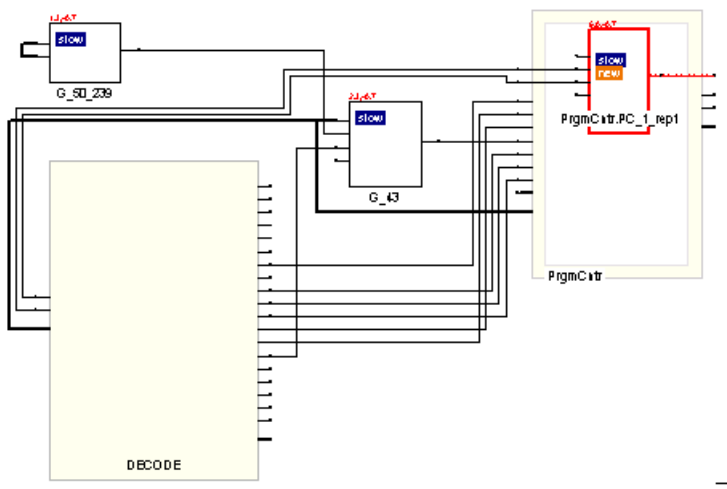
Example: Slow and New Properties

The slow property is useful for analyzing your critical path, because it denotes objects that do not meet the timing criteria. The following figure shows a filtered view of a critical path, with slow instances flagged in blue.



The New property helps with debugging because it quickly identifies objects that have been added to the current schematic with commands like Expand. You can step through successive filtered views to determine what was added at each step.

The next figure expands one of the pins from the previous filtered view. The new instance added to the view has two flags: new and slow.



Using the `orig_inst_of` Property for Parameterized Modules

The compiler automatically uniquifies parameterized modules or instances. Properties are available to identify the RTL names of both uniquified and original modules or instances.

- `inst_of` property – identifies module or instance by uniquified name
- `orig_inst_of` property – identifies module or instance by its original name before it was uniquified

In the following example, the top-level module (`top`) instantiates the module `sub` multiple times using different parameter values. The compiler uniquifies the module `sub` as `sub_3s`, `sub_1s`, and `sub_4s`.

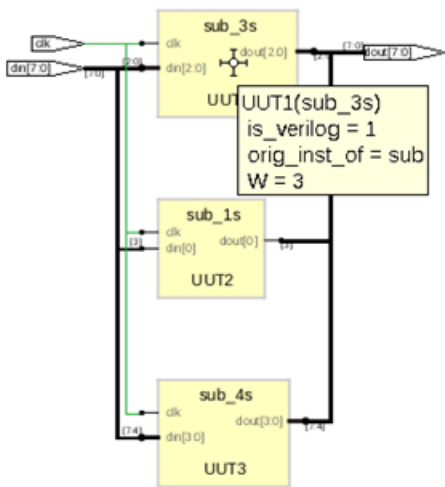
Top.v

```
module top (input clk, [7:0] din, output [7:0] dout);
    sub #(.W(3)) UUT1 (.clk, .din(din[2:0]), .dout(dout[2:0]));
    sub #(.W(1)) UUT2 (.clk, .din(din[3]), .dout(dout[3]));
    sub #(.W(4)) UUT3 (.clk, .din(din[7:4]), .dout(dout[7:4]));
endmodule

module sub #(parameter W = 0) (
    input clk,
    input [W-1:0] din,
    output logic [W-1:0] dout);

    always@(posedge clk)
        begin
            dout <= din;
        end
endmodule
```

RTL View



TCL Command Example

Use the `get_prop` command with the `orig_inst_of` property to identify the original RTL name for the module:

```
% get_prop -prop orig_inst_of {v:sub_3s}
sub

% get_prop -prop orig_inst_of {i:UUT3}
sub
```

Selecting Objects in the RTL/Technology Views

For mouse selection, standard object selection rules apply: In selection mode, the pointer is shaped like a crosshair.

To select...	Do this...
Single objects	Click on the object in the RTL or Technology schematic, or click the object name in the Hierarchy Browser.

To select...	Do this...
Multiple objects	Use one of these methods: • Draw a rectangle around the objects. • Select an object, press Ctrl, and click other objects you want to select. • Select multiple objects in the Hierarchy Browser. See Browsing With the Hierarchy Browser , on page 315. • Use Find to select the objects you want. See Using Find for Hierarchical and Restricted Searches , on page 317.
Objects by type (instances, ports, nets)	Use Edit->Find to select the objects (see Browsing With the Find Command , on page 316), or use the Hierarchy Browser, which lists objects by type.
All objects of a certain type (instances, ports, nets)	To select all objects of a certain type, do either of the following: • Right-click and choose the appropriate command from the Select All Schematic/Current Sheet popup menus. • Select the objects in the Hierarchy Browser.
No objects (deselect all currently selected objects)	Click the left mouse button in a blank area of the schematic or click the right mouse button to bring up the pop-up menu and choose Unselect All. Deselected objects are no longer highlighted.

The HDL Analyst view highlights selected objects in red. If the object you select is on another sheet of the schematic, the schematic tracks to the appropriate sheet. If you have other windows open, the selected object is highlighted in the other windows as well (crossprobing), but the other windows do not track to the correct sheet. Selected nets that span different hierarchical levels are highlighted on all the levels. See [Crossprobing \(Standard\)](#), on page 328 for more information about crossprobing.

Some commands affect selection by adding to the selected set of objects: the Expand commands, the Select All commands, and the Select Net Driver and Select Net Instances commands.

Working with Multisheet Schematics

The title bar of the RTL or Technology view indicates the number of sheets in that schematic. In a multisheet schematic, nets that span multiple sheets are indicated by sheet connector symbols, which you can use for navigation.

1. To reduce the number of sheets in a schematic, select Options->HDL Analyst Options and increase the values set for Sheet Size Options - Instances and Sheet Size Options - Filtered Instances. To display fewer objects per sheet (increase the number of sheets), increase the values.

These options set a limit on the number of objects displayed on an unfiltered and filtered schematic sheet, respectively. A low Filtered Instances value can cause lower-level logic inside a transparent instance to be displayed on a separate sheet. The sheet numbers are indicated inside the empty transparent instance.

2. To navigate through a multisheet schematic, refer to this table. It summarizes common operations and ways to navigate.

To view...	Use one of these methods...
Next sheet or previous sheet	Select View->Next/Previous Sheet. Press the right mouse button and draw a horizontal mouse stroke (left to right for next sheet, right to left for previous sheet). Click the icons: Next Sheet () or Previous Sheet () Press Shift-right arrow (Next Sheet) or Shift-left arrow (Previous sheet). Navigate with View->Back and View ->Forward if the next/previous sheets are part of the display history.
A specific sheet number	Select View->View Sheets and select the sheet. Click the right mouse button, select View Sheets from the popup menu, and then select the sheet you want. Press Ctrl-g and select the sheet you want.
Lower-level logic of a transparent instance on separate sheets	Check the sheet numbers indicated inside the empty transparent instance. Use the sheet navigation commands like Next Sheet or View Sheets to move to the sheet you need.
All objects of a certain type	To highlight all the objects of the same type in the schematic, right-click and select the appropriate command from the Select All Schematic popup menu. To highlight all the objects of the same type on the current sheet, right-click and select the appropriate command from the Select All Sheet popup menu.
Selected items only	Filter the schematic as described in Filtering Schematics , on page 340.

Setting Schematic Preferences

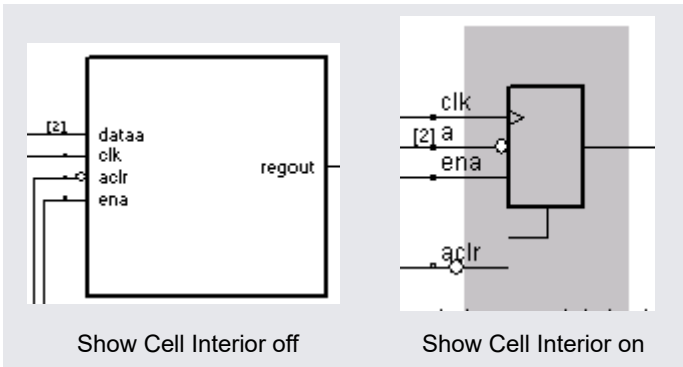
You can set various preferences for the RTL and Technology views from the user interface.

1. Select Options->HDL Analyst Options. For a description of all the options on this form, see [HDL Analyst Options Command](#), on page 475 in the *Reference Manual*.
2. The following table details some common operations:

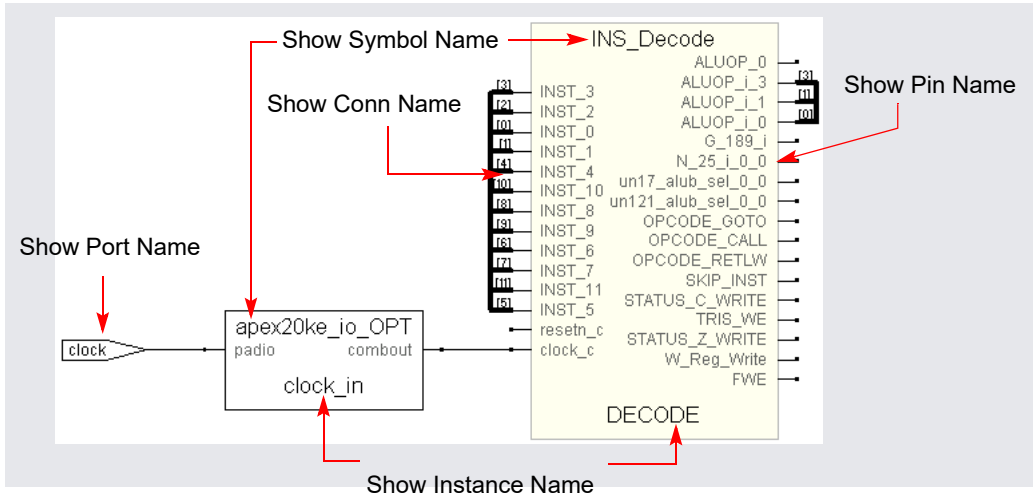
To...	Do this...
Display the Hierarchy Browser	Enable Show Hierarchy Browser (General tab).
Control crossprobing from an object to a P&R text file	Enable Enhanced Text Crossprobing. (General tab)
Determine the number of objects displayed on a sheet.	Set the value with Maximum Instances on the Sheet Size tab. Increase the value to display more objects per sheet.
Determine the number of objects displayed on a sheet in a filtered view.	Set the value with Maximum Filtered Instances on the Sheet Size tab. Increase the number to display more objects per sheet. You cannot set this option to a value less than the Maximum Instances value.

Some of these options do not take effect in the current view, but are visible in the next schematic you open.

3. To view hierarchy within a cell, enable the General->Show Cell Interiors option.



- To control the display of labels, first enable the Text->Show Text option, and then enable the Label Options you want. The following figure illustrates the label that each option controls.



For a more detailed information about some of these options, see [Schematic Display, on page 93](#) in the *Reference Manual*.

- Click OK on the HDL Analyst Options form.

The software writes the preferences you set to the `ini` file, and they remain in effect until you change them.

Managing Windows

As you work on a project, you open different windows. For example, you might have two Technology views, an RTL view, and a source code window open. The following guidelines help you manage the different windows you have open. For information about cycling through the display history in a single schematic, see [Moving Between Views in a Schematic Window, on page 303](#).

- Toggle on View->Workbook Mode.

Below the Project view, you see tabs like the following for each open view. The tab for the current view is on top. The symbols in front of the view name on the tab help identify the kind of view.



2. To bring an open view to the front, if the window is not visible, click its tab. If part of the window is visible, click in any part of the window.

If you previously minimized the view, it will be in minimized form.
Double-click the minimized view to open it.

3. To bring the next view to the front, click Ctrl-F6 in that window.
4. Order the display of open views with the commands from the Window menu. You can cascade the views (stack them, slightly offset), or tile them horizontally or vertically.
5. To close a view, press Ctrl-F4 in that window or select File->Close.

Exploring Design Hierarchy

Schematics generally have a certain amount of design hierarchy. You can move between hierarchical levels using the Hierarchy Browser or Push/Pop mode. For additional information, see [Analyzing With the Standard HDL Analyst Tool, on page 336](#). The topics include:

- [Traversing Design Hierarchy with the Hierarchy Browser](#), on page 307
- [Exploring Object Hierarchy by Pushing/Popping](#), on page 308
- [Exploring Object Hierarchy of Transparent Instances](#), on page 314

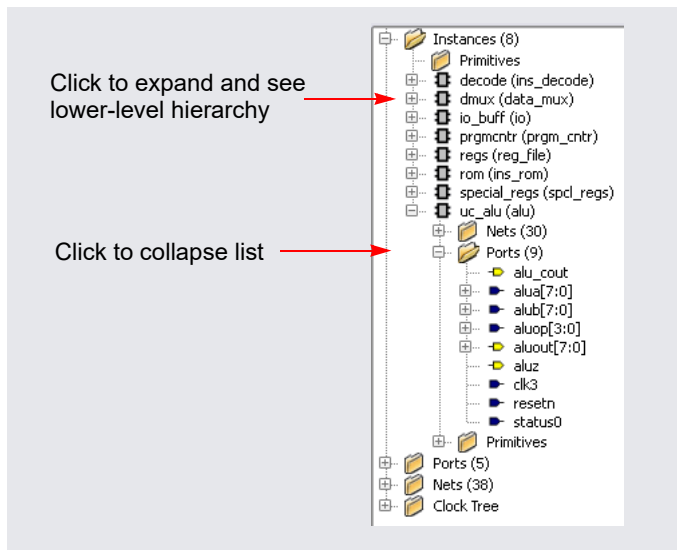
Traversing Design Hierarchy with the Hierarchy Browser

The Hierarchy Browser is the list of objects on the left side of the RTL and Technology views. It is best used to get an overview, or when you need to browse and find an object. If you want to move between design levels of a particular object, Push/Pop mode is more direct. Refer to [Exploring Object Hierarchy by Pushing/Popping, on page 308](#) for details.

The hierarchy browser allows you to traverse and select the following:

- Instances and submodules
- Ports
- Internal nets
- Clock trees (in an RTL view)

The browser lists the objects by type. A plus sign in a square icon indicates that there is hierarchy under that object and a minus sign indicates that the design hierarchy has been expanded. To see lower-level hierarchy, click on the plus sign for the object. To ascend the hierarchy, click on the minus sign.



Refer to [Hierarchy Browser Symbols](#), on page 47 in the *Reference Manual* for an explanation of the symbols.

Exploring Object Hierarchy by Pushing/Popping

To view the internal hierarchy of a specific object, it is best to use Push/Pop mode or examine transparent instances, instead of using the Hierarchy Browser described in [Traversing Design Hierarchy with the Hierarchy Browser](#), on page 307. You can access Push/Pop mode with the Push/Pop Hierarchy icon, the Push/Pop Hierarchy command, or mouse strokes.

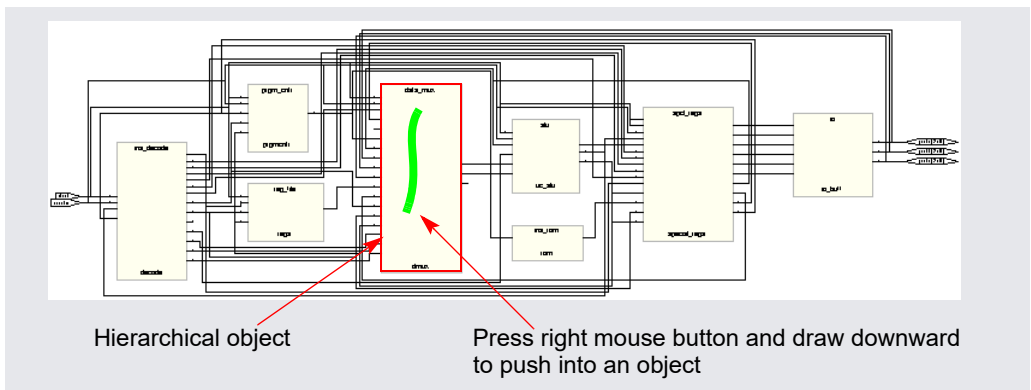
When combined with other commands like filtering and expansion commands, Push/Pop mode can be a very powerful tool for isolating and analyzing logic. See [Filtering Schematics](#), on page 340, [Expanding Pin and Net Logic](#), on page 342, and [Expanding and Viewing Connections](#), on page 346 for details about filtering and expansion. See the following sections for information about pushing down and popping up in hierarchical design objects:

- [Pushing into Objects](#), on page 309, next
- [Popping up a Hierarchical Level](#), on page 312

Pushing into Objects

In the schematic, you can push into objects and view the lower-level hierarchy. You can use a mouse stroke, the command, or the icon to push into objects:

1. To move down a level (push into an object) with a mouse stroke, put your cursor near the top of the object, hold down the right mouse button, and draw a vertical stroke from top to bottom. You can push into the following objects; see step 3 for examples of pushing into different types of objects.
 - Hierarchical instances. They can be displayed as pale yellow boxes (opaque instances) or hollow boxes with internal logic displayed (transparent instances). You cannot push into a hierarchical instance that is hidden with the Hide Instance command (internal logic is hidden).



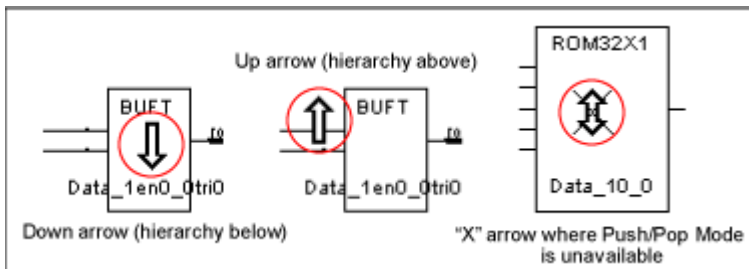
- Technology-specific primitives. The primitives are listed in the Hierarchy Browser in the Technology view, under Instances - Primitives.
- Inferred ROMs and state machines.

The remaining steps show you how to use the icon or command to push into an object.

2. Enable Push/Pop mode by doing one of the following:

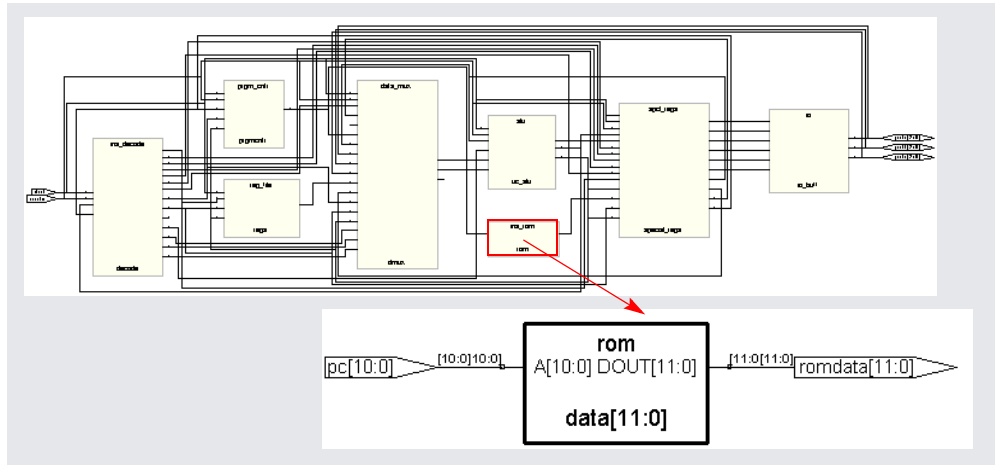
- Select View->Push/Pop Hierarchy.
- Right-click in the Technology view and select Push/Pop Hierarchy from the pop-up menu.
- Click the Push/Pop Hierarchy icon () in the toolbar (two arrows pointing up and down).
- Press F2.

The cursor changes to an arrow. The direction of the arrow indicates the underlying hierarchy, as shown in the following figure. The status bar at the bottom of the window reports information about the objects over which you move your cursor.

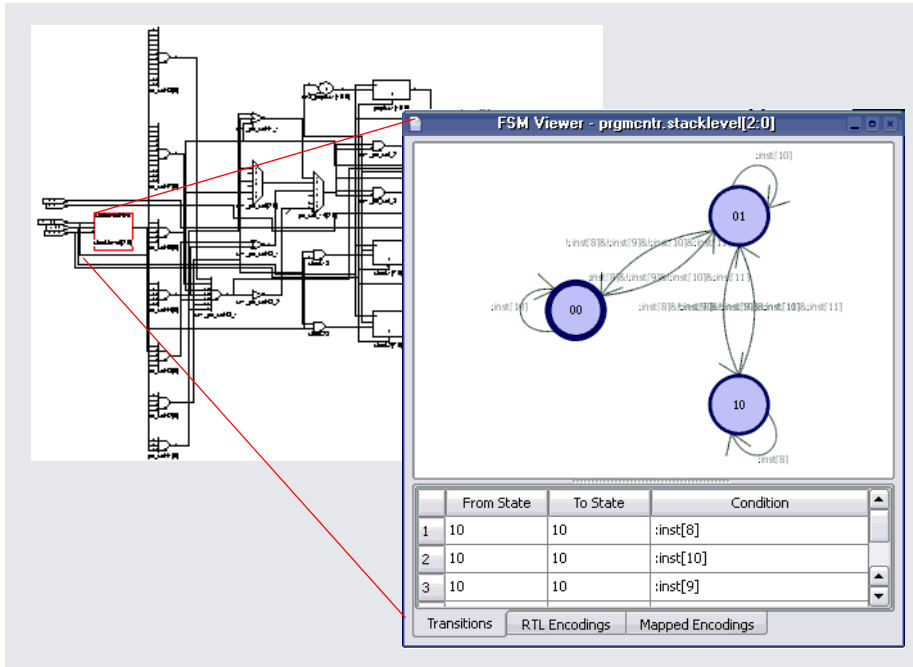


3. To push (descend) into an object, click on the hierarchical object. For a transparent instance, you must click on the pale yellow border. The following figure shows the result of pushing into a ROM.

When you descend into a ROM, you can push into it one more time to see the ROM data table. The information is in a view-only text file called rom.info.

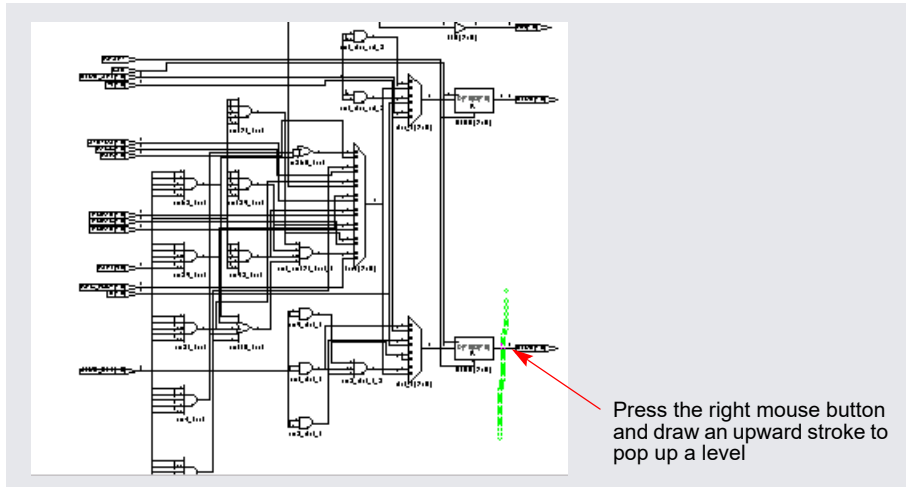


Similarly, you can push into a state machine. When you push into an FSM from the RTL view, you open the FSM viewer where you can graphically view the transitions. For more information, see [Using the FSM Viewer \(Standard\), on page 353](#). If you push into a state machine from the Technology view, you see the underlying logic.



Popping up a Hierarchical Level

1. To move up a level (pop up a level), put your cursor anywhere in the design, hold down the right mouse button, and draw a vertical mouse stroke, moving from the bottom upwards.



The software moves up a level, and displays the next level of hierarchy.

2. To pop (ascend) a level using the commands or icon, do the following:
 - Select the command or icon if you are not already in Push/Pop mode. See [Pushing into Objects, on page 309](#) for details.
 - Move your cursor to a blank area and click.
3. To exit Push/Pop mode, do one of the following:
 - Click the right mouse button in a blank area of the view.
 - Deselect View->Push/Pop Hierarchy.
 - Deselect the Push/Pop Hierarchy icon.
 - Press F2.

Exploring Object Hierarchy of Transparent Instances

Examining a transparent instance is one way of exploring the design hierarchy of an object. The following table compares this method with pushing (described in [Exploring Object Hierarchy by Pushing/Popping, on page 308](#)).

	Pushing	Transparent Instance
User control	You initiate the operation through the command or icon.	You have no direct control; the transparent instance is automatically generated by some commands that result in a filtered view.
Design context	Context lost; the lower-level logic is shown in a separate view	Context maintained; lower-level logic is displayed inside a hollow yellow box at the hierarchical level of the parent.

Finding Objects (Standard)

In the schematic, you can use the Hierarchy Browser or the Find command to find objects, as explained in these sections:

- [Browsing to Find Objects in HDL Analyst Views](#), on page 315
- [Using Find for Hierarchical and Restricted Searches](#), on page 317
- [Using Wildcards with the Find Command](#), on page 320
- [Using Find to Search the Output Netlist](#), on page 325

For information about the Tcl Find command, which you use to locate objects, and create collections, see [find](#), on page 151 in the *Command Reference Manual*.

Browsing to Find Objects in HDL Analyst Views

You can always zoom in to find an object in the RTL and Technology schematics. The following procedure shows you how to browse through design objects and find an object at any level of the design hierarchy. You can use the Hierarchy Browser or the Find command to do this. If you are familiar with the design hierarchy, the Hierarchy Browser can be the quickest method to locate an object. The Find command is best used to graphically browse and locate the object you want.

Browsing With the Hierarchy Browser

1. In the Hierarchy Browser, click the name of the net, port, or instance you want to select.

The object is highlighted in the schematic.

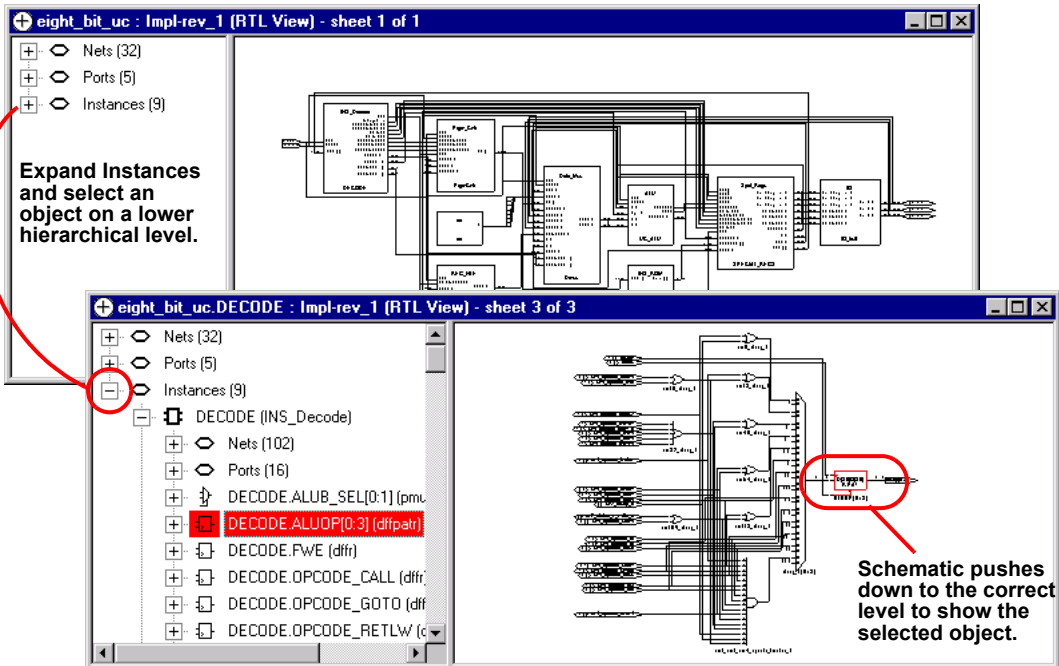
2. To select a range of objects, select the first object in the range. Then, scroll to display the last object in the range. Press and hold the Shift key while clicking the last object in the range.

The software selects and highlights all the objects in the range.

3. If the object is on a lower hierarchical level, do either of the following:
 - Expand the appropriate higher-level object by clicking the plus symbol next to it, and then select the object you want.

- Push down into the higher-level object, and then select the object from the Hierarchy Browser.

The selected object is highlighted in the schematic. The following example shows how moving down the object hierarchy and selecting an object causes the schematic to move to the sheet and level that contains the selected object.



4. To select all objects of the same type, select them from the Hierarchy Browser. For example, you can find all the nets in your design.

Browsing With the Find Command

1. In a schematic, select HDL Analyst->Find or press Ctrl-f to open the Object Query dialog box.
2. Do the following in the dialog box:
 - Select objects in the selection box on the left. You can select all the objects or a smaller set of objects to browse. If length makes it hard to read a name, click the name in the list to cause the software to display the entire name in the field at the bottom of the dialog box.

- Click the arrow to move the selected objects over to the box on the right.

The software highlights the selected objects.

3. In the Object Query dialog box, click on an object in the box on the right.

The software tracks to the schematic page with that object.

Using Find for Hierarchical and Restricted Searches

You can always zoom in to find an object in the RTL and Technology schematics or use the Hierarchy Browser (see [Browsing to Find Objects in HDL Analyst Views, on page 315](#)). This procedure shows you how to use the Find command to do hierarchical object searches or restrict the search to the current level or the current level and its underlying hierarchy.

Note that Find only adds to the current selection; it does not deselect anything that is already selected. you can use successive searches to build up exactly the selection you need, before filtering.

1. If needed, restrict the range of the search by filtering the view.

See [Viewing Design Hierarchy and Context, on page 337](#) and [Filtering Schematics, on page 340](#) for details. With a filtered view, the software only searches the filtered instances, unless you set the scope of the search to Entire Design, as described below, in which case Find searches the entire design.

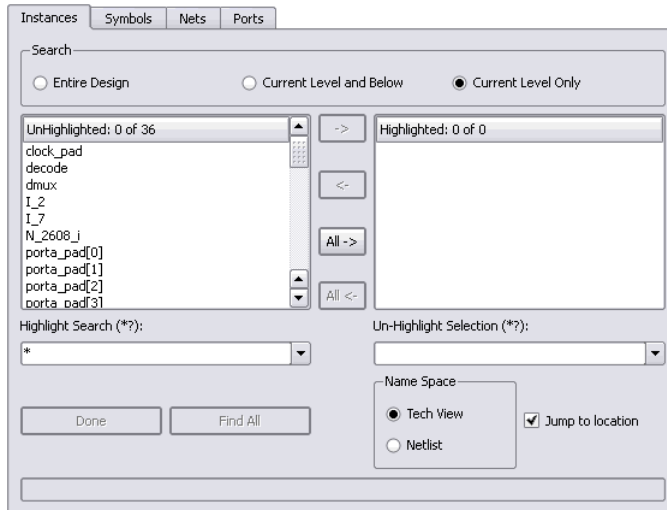
You can use the filtering technique to restrict your search to just one schematic sheet. Select all the objects on one sheet and filter the view. Continue with the procedure.

2. To further restrict the range of the search, hide instances you do not need.

You can do this in addition to filtering the view, or instead of filtering the view. Hidden instances and their hierarchy are excluded from the search. When you have finished the search, use the Unhide Instances command to make the hierarchy visible again.

3. Open the Object Query dialog box.

- Do one of the following: right click in the RTL or Technology view and select Find from the popup menu, press Ctrl-f, or click the Find icon ().
- Reposition the dialog box so you can see both your schematic and the dialog box.



4. Select the tab for the type of object. The Unhighlighted box on the left lists all objects of that type (instances, symbols, nets, or ports).

For fastest results, search by Instances rather than Nets. When you select Nets, the software loads the whole design, which could take some time.

5. Click one of these buttons to set the hierarchical range for the search: Entire Design, Current Level & Below, or Current Level Only, depending on the hierarchical level of the design to which you want to restrict your search.

The range setting is especially important when you use wildcards. See [Effect of Hierarchy and Range on Wildcard Searches, on page 320](#) for details. Current Level Only or Current Level & Below are useful for searching filtered schematics or critical path schematics.

The lower-level details of a transparent instance appear at the current level and are included in the search when you set it to Current Level Only. To exclude them, temporarily hide the transparent instances, as described in step 2.

Use Entire Design to hierarchically search the whole design. For large hierarchical designs, reduce the scope of the search by using the techniques described in the first step.

The Unhighlighted box shows available objects within the scope you set. Objects are listed in alphabetical order, not hierarchical order.

6. To search for objects in the mapped database or the output netlist, set the Name Space option.

The name of an object might be changed because of synthesis optimizations or to match the place-and-route tool conventions, so that the object name may no longer match the name in the original netlist. Setting the Name Space option ensures that the Find command searches the correct database for the object. For example, if you set this option to Tech View, the tool searches the mapped database (`srn`) for the object name you specify. For information about using this feature to find objects from an output netlist, see [Using Find to Search the Output Netlist, on page 325](#).

7. Do the following to select objects from the list. To use wildcards in the selection, see the next step.
 - Click on the objects you want from the list. If length makes it hard to read a name, click the name in the list to cause the software to display the entire name in the field at the bottom of the dialog box.
 - Click Find 200 or Find All. The former finds the first 200 matches, and then you can click the button again to find the next 200.
 - Click the right arrow to move the objects into the box on the right, or double-click individual names.

The schematic displays highlighted objects in red.

8. Do the following to select objects using patterns or wildcards.
 - Type a pattern in the Highlight Wildcard field. See [Using Wildcards with the Find Command, on page 320](#) for a detailed discussion of wildcards.

The Unhighlighted list shows the objects that match the wildcard criteria. If length makes it hard to read a name, click the name in the list to cause the software to display the entire name in the field at the bottom of the form.

- Click the right arrow to move the selections to the box on the right, or double-click individual names. The schematic displays highlighted objects in red.

You can use wildcards to avoid typing long pathnames. Start with a general pattern, and then make it more specific. The following example browses and uses wildcards successively to narrow the search.

Find all instances three levels down	*.*.*
Narrow search to find instances that begin with i_	i_*.*.*
Narrow search to find instances that begin with un2 after the second hierarchy separator	i_*.*.un2*

Note that there are some differences when you specify the find command in the RTL view, Technology view, or the constraint file.

9. You can leave the dialog box open to do successive Find operations. Click OK or Cancel to close the dialog box when you are done.

For detailed information about the Find command and the Object Query dialog box, see [Find Command \(HDL Analyst\)](#), on page 335 of the *Reference Manual*.

Using Wildcards with the Find Command

Use the following wildcards when you search the schematics:

*	The asterisk matches any sequence of characters.
?	The question mark matches any single character.
.	The dot explicitly matches a hierarchy separator, so type one dot for each level of hierarchy. To use the dot as a pattern and not a hierarchy separator, type a backslash before the dot: \.

Effect of Hierarchy and Range on Wildcard Searches

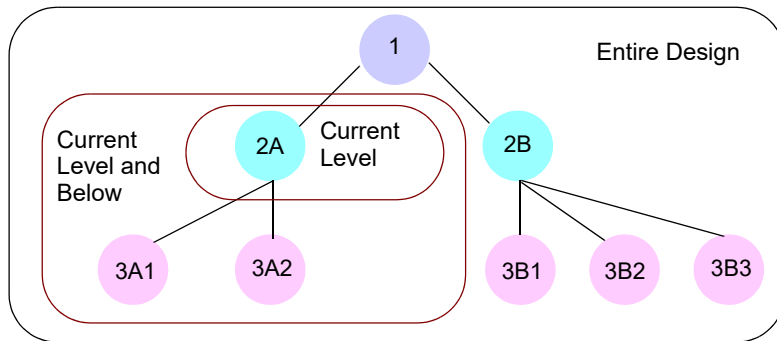
The asterisk and question mark wildcards do not cross hierarchical boundaries, but search each level of hierarchy individually with the search pattern. This default is affected by the following:

- Hierarchical separators

Dots match hierarchy separators, unless you use the backslash escape character in front of the dot (\.). Hierarchical search patterns with a dot (!*.*) are repeated at each level included in the scope. If you use the *.* pattern with Current Level, the software matches non-hierarchical names at the current level that include a dot.

- Search range

The scope of the search determines the starting point for the searches. Some times the starting point might make it appear as if the wildcards cross hierarchical boundaries. If you are at 2A in the following figure and the scope of the search is set to Current Level and Below, separate searches start at 2A, 3A1, and 3A2. Each search does not cross hierarchical boundaries. If the scope of the search is Entire Design, the wildcard searches run from each hierarchical point (1, 2A, 2B, 3A1, 3A2, 3B1, 3B2, and 3B3). The result of an asterisk search (*) with Entire Design is a list of all matches in the design, regardless of the current level.



See [Wildcard Search Examples](#), on page 322 for examples.

How a Wildcard Search Works

1. The starting point of a wildcard search depends on the range set for the search.

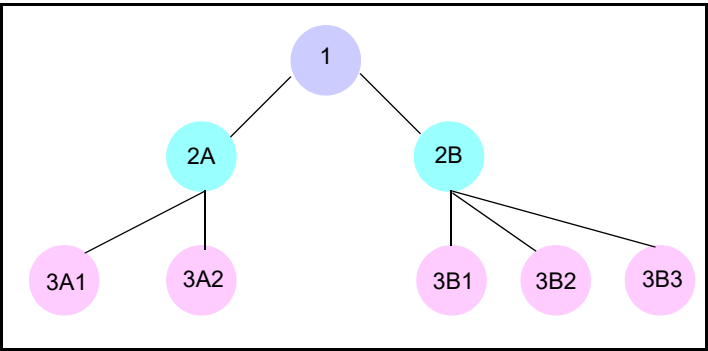
Entire Design	Starts at top level and uses the pattern to search from that level. It then moves to any child levels below the top level and searches them. The software repeats the search pattern at each hierarchical point in the design until it searches the entire design.
Current Level	Starts at the current hierarchical level and searches that level only. A search started at 2A only covers 2A.
Current Level and Below	Starts at the current hierarchical level and searches that level. It then moves to any child levels below the starting point and conducts separate searches from each of these starting points.

2. The software applies the wildcard pattern to all applicable objects within the range. For Current Level and Current Level and Below, the current level determines the starting point.

Dots match hierarchy separators, unless you use the backslash escape character in front of the dot (\.). Hierarchical search patterns with a dot (l*.*) are repeated at each level included in the scope. See [Effect of Hierarchy and Range on Wildcard Searches, on page 320](#) and [Wildcard Search Examples, on page 322](#) for details and examples, respectively. If you use the *.* pattern with Current Level, the software matches non-hierarchical names at the current level that include a dot.

Wildcard Search Examples

The figure shows a design with three hierarchical levels, and the table shows the results of some searches on this design.



Scope	Pattern	Starting Point	Finds Matches in...
Entire Design	*	3A1	1, 2A, 2B, 3A1, 3A2, 3B1, 3B2, and 3B3 (* at all levels)
	.	2B	2A and 2B (*. from 1) 3A1, 3A2, 3B1, 3B2, and 3B3 (*. from 2A and 2B) No matches in 1 (because of the hierarchical dot), unless a name includes a non-hierarchical dot.
Current Level	*	1	1 only (no hierarchical boundary crossing)
	.	2B	2B only. No search of lower levels even though the dot is specified, because the scope is Current Level. No matches, unless a 2B name includes a non-hierarchical dot.
Current Level and Below	*	2A	2A only (no hierarchical boundary crossing)
	.	1	2A and 2B (*. from 1) 3A1, 3A2, 3B1, 3B2, and 3B3 (*. from 2A and 2B) No matches from 1, because the dot is specified.
	.	2B	3B1, 3B2, and 3B3 (*. from 2B)
	.	3A2	No matches (no hierarchy below 3A2)
	..*	1	3A1, 3A2, 3B1, 3B2, and 3B3 (*. from 1) Search ends because there is no hierarchy two levels below 2A and 2B.

Difference from Tcl Search

In a simple Tcl search, no character (except the backslash, \) has special meaning. This means that the asterisk matches everything in a string. The FPGA synthesis tool confines the simple search to within one level of hierarchy. The following command searches each level of hierarchy individually for the specified pattern:

```
find -hier *abc*addr_reg[*]
```

If you want to go through the hierarchy, you must add the hierarchy separators to the search pattern:

```
find {*.*.abc.*.*.addr_reg[*]}
```

Find Command Differences in HDL Analyst Views and Constraint File

There are some slight differences when you use the Find command in the RTL view, Technology view, and the constraint files:

- You cannot use find to search for bit registers of a bit array in the RTL or Technology views, but you can specify it in a constraint file, where the following command will work:

```
find -seq {i:modulex_inst.qb[7]}
```

In a HDL Analyst view, you cannot find {i:modulex_inst.qb[7]}, but you can specify and find {i:modulex_inst.qb[7:0]}.

- By default, the following Tcl command does not find objects in the RTL view, although it does find objects in the Technology view:

```
-hier -seq * -filter @clock == clk75
```

To make this work in an RTL view, you must turn on Annotated Properties for Analyst in the Device tab of the Implementation Options dialog box, recompile the design, and then open a new RTL view.

Combining Find with Filtering to Refine Searches

You can combine the Find command with the filtering commands to better effect. Depending on what you want to do, use the Find command first, or a filtering command.

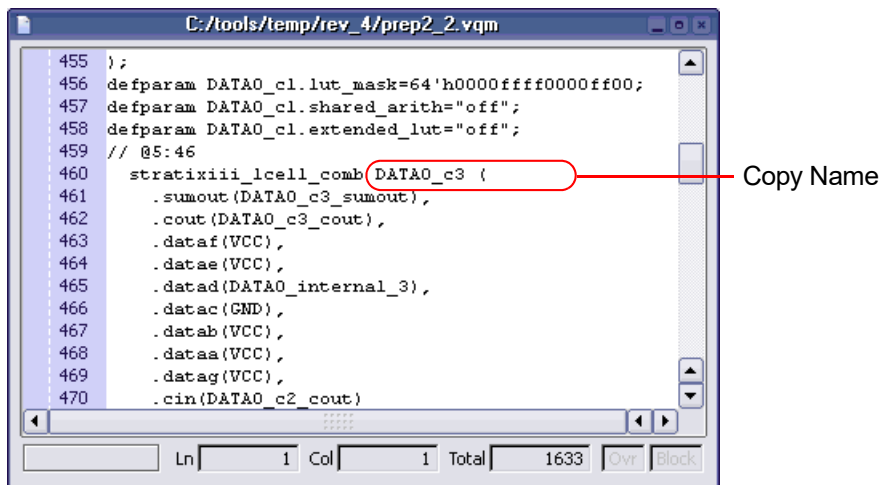
1. To limit the range of a search, do the following:

- Filter the design.
 - Use the Find command on the filtered view, but set the search range to Current Level Only.
2. Select objects for a filtered view.
 - Use the Find command to browse and select objects.
 - Filter the objects to display them.

Using Find to Search the Output Netlist

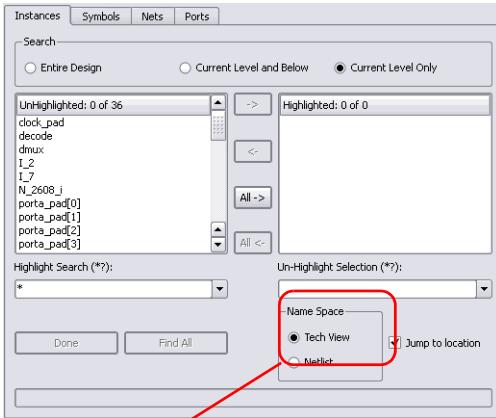
When the synthesis tool creates an output netlist file, some names are optimized for use in the P&R tool. When you debug your design for place and route looking for a particular object, use the Name Space option in the Object Query dialog box to locate the optimized names in the output netlist. The following procedure shows you how to locate an object, highlight and filter it in the Technology view, and crossprobe to the source code for editing.

1. Select the output netlist file option in the Implementations Results tab of the Implementation Options dialog box.
2. After you synthesize your design, open your output netlist file and select the name of the object you want to find.

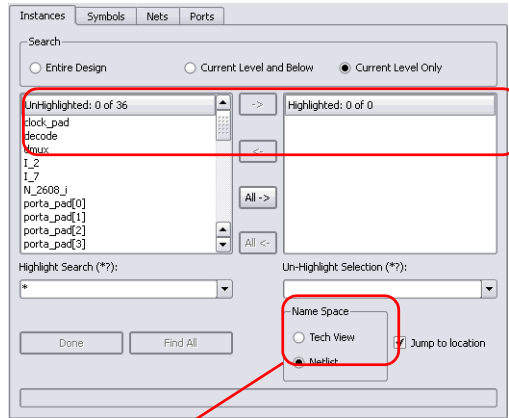


3. Copy the name and open a Technology view.

4. In the Technology view, press Ctrl-f or select Edit->Find to open the Object Query dialog box and do the following:
 - Paste the object name you copied into the Highlight Search field.
 - Set the Name Space option to Netlist and click Find All.



Search by Tech View



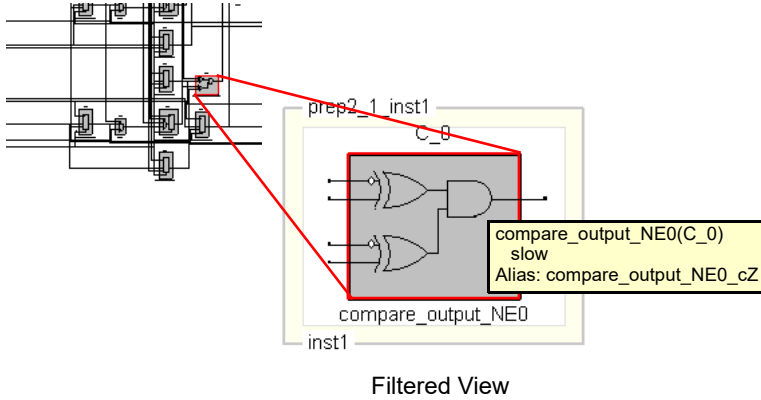
Search by Netlist

If you leave the Name Space option set to the default of Tech View, the tool does not find the name because it is searching the mapped database instead of the output netlist.

- Double click the name to move it into the Highlighted field and close the dialog box.

In the Technology view, the name is highlighted in the schematic.

5. Select HDL Analyst->Filter Schematic to view only the highlighted portion of the schematic.



The tooltip shows the equivalent name in the Technology view.

6. Double click on the filtered schematic to crossprobe to the corresponding code in the HDL file.

Crossprobing (Standard)

Crossprobing is the process of selecting an object in one view and having the object or the corresponding logic automatically highlighted in other views. Highlighting a line of text, for example, highlights the corresponding logic in the schematic. Crossprobing helps you visualize where coding changes or timing constraints might help to reduce area or improve performance.

You can crossprobe between the RTL view, Technology view, the FSM Viewer, the log file, the source files, and some external text files from place-and-route tools. However, not all objects or source code crossprobe to other views, because some source code and RTL view logic is optimized away during the compilation or mapping processes.

This section describes how to crossprobe from different views. It includes the following:

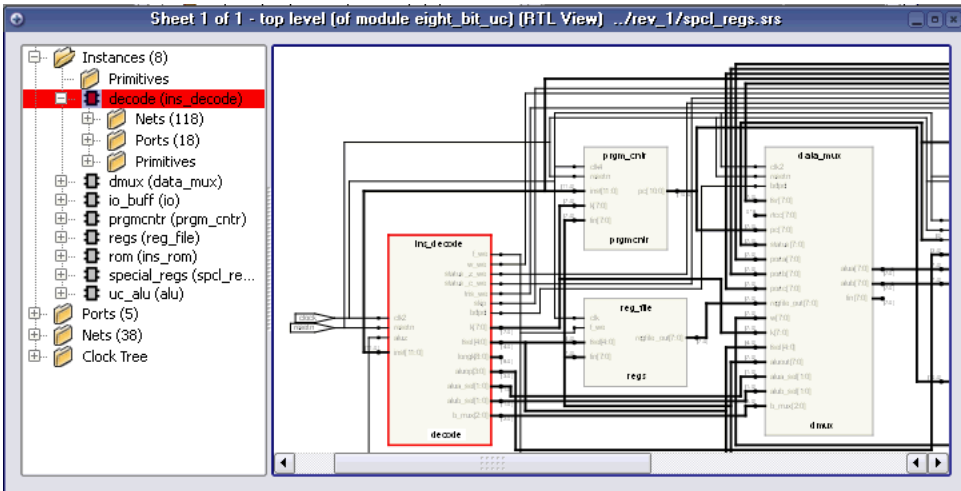
- [Crossprobing within an RTL/Technology View](#), on page 328
- [Crossprobing from the RTL/Technology View](#), on page 329
- [Crossprobing from the Text Editor Window](#), on page 331
- [Crossprobing from the Tcl Script Window](#), on page 334
- [Crossprobing from the FSM Viewer](#), on page 334

Crossprobing within an RTL/Technology View

Selecting an object name in the Hierarchy Browser highlights the object in the schematic, and vice versa.

Selected Object	Highlighted Object
Instance in schematic (single-click)	Module icon in Hierarchy Browser
Net in schematic	Net icon in Hierarchy Browser
Port in schematic	Port icon in Hierarchy Browser
Logic icon in Hierarchy Browser	Instance in schematic
Net icon in Hierarchy Browser	Net in schematic
Port icon in Hierarchy Browser	Port in schematic

In this example, when you select the DECODE module in the Hierarchy Browser, the DECODE module is automatically selected in the RTL view.

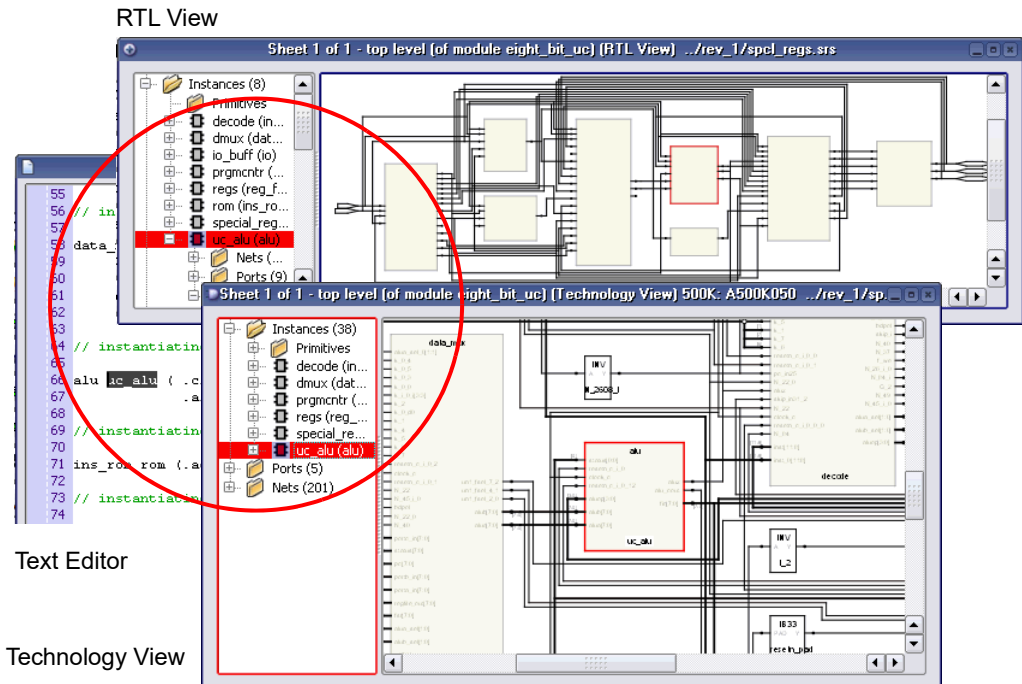


Crossprobing from the RTL/Technology View

1. To crossprobe from an RTL or Technology views to other open views, select the object by clicking on it.

The software automatically highlights the object in all open views. If the open view is a schematic, the software highlights the object in the Hierarchy Browser on the left as well as in the schematic. If the highlighted object is on another sheet of a multi-sheet schematic, the view does not automatically track to the page. If the crossprobed object is inside a hidden instance, the hidden instance is highlighted in the schematic.

If the open view is a source file, the software tracks to the appropriate code and highlights it. The following figure shows crossprobing between the RTL, Technology, and Text Editor (source code) views.



2. To crossprobe from the RTL or Technology view to the source file when the source file is not open, double-click on the object in the RTL or Technology view.

Double-clicking automatically opens the appropriate source code file and highlights the appropriate code. For example, if you double-click an object in a Technology view, the HDL Analyst tool automatically opens an editor window with the source code and highlights the code that contains the selected register.

The following table summarizes the crossprobing capability from the RTL or Technology view.

From	To	Procedure
RTL	Source code	Double-click an object. If the source code file is not open, the software opens the Text Editor window to the appropriate section of code. If the source file is already open, the software scrolls to the correct section of the code and highlights it.
RTL	Technology	The Technology view must be open. Click the object to highlight and crossprobe.
RTL	FSM Viewer	The FSM view must be open. The state machine must be coded with a onehot encoding style. Click the FSM to highlight and crossprobe.
Technology	Source code	If the source code file is already open, the software scrolls to the correct section of the code and highlights it. If the source code file is not open, double-click an object in the Technology view to open the source code file.
Technology	RTL	The RTL view must be open. Click the object to highlight and crossprobe.

Crossprobing from the Text Editor Window

To crossprobe from a source code window or from the log file to an RTL, Technology, or FSM view, use this procedure. You can use this method to crossprobe from any text file with objects that have the same instance names as in the synthesis software. For example, you can crossprobe from place-and-route files. See [Example of Crossprobing a Path from a Text File, on page 332](#) for a practical example of how to use crossprobing.

1. Open the RTL, FSM, or Technology view to which you want to crossprobe.
2. To crossprobe from an error, warning, or note in the html log file, click on the file name to open the corresponding source code in another Text Editor window; to crossprobe from a text log file, double-click on the text of the error, warning, or note.
3. To crossprobe from a third-party text file (not source code or a log file), select Options->HDL Analyst Options->General, and enable Enhanced text crossprobing.

4. Select the appropriate portion of text in the Text Editor window. In some cases, it may be necessary to select an entire block of text to crossprobe.

The software highlights the objects corresponding to the selected code in all the open windows. For example, if you select a state name in the code, it highlights the state in the FSM viewer. If an object is on another schematic sheet or on another hierarchical level, the highlighting might not be obvious. If you filter the RTL or schematic view (right-click in the source code window with the selected text and select **Filter Schematic** from the popup menu), you can isolate the highlighted objects for easy viewing.

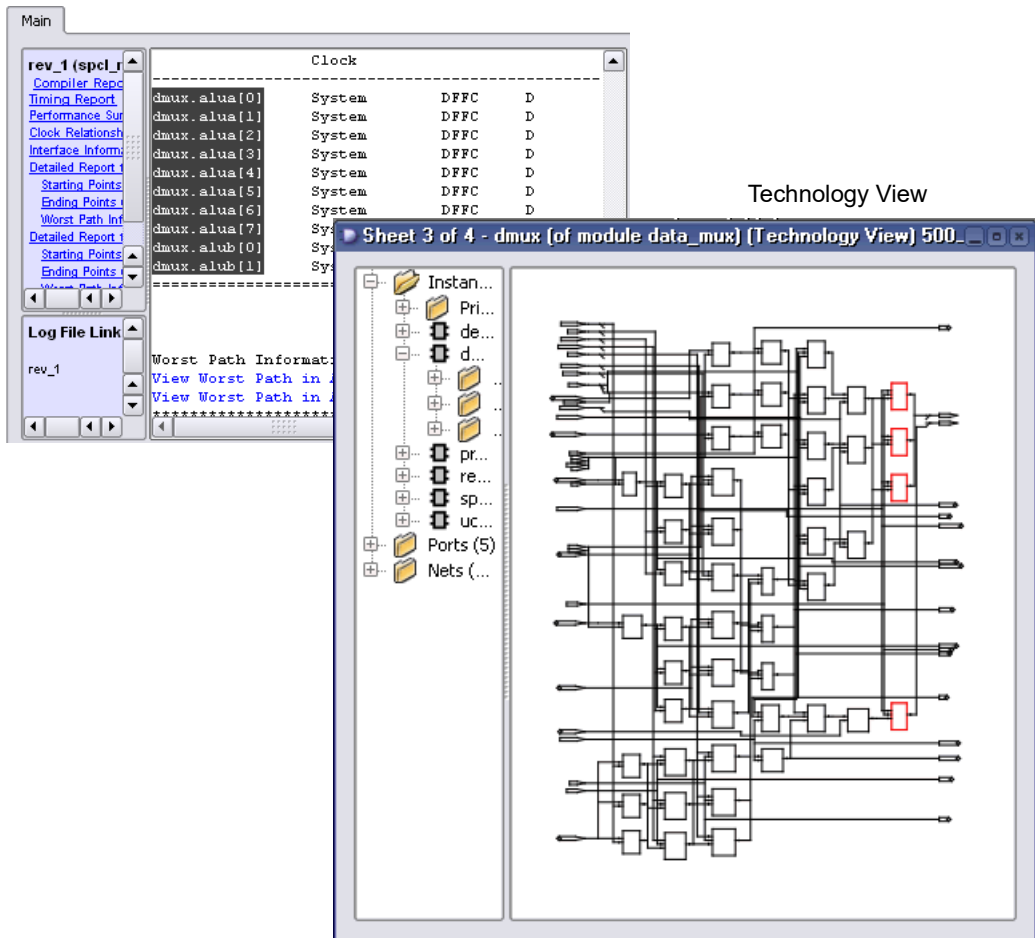
Example of Crossprobing a Path from a Text File

This example selects a path in a log file and crossprobes it in the Technology view. You can use the same technique to crossprobe from other text files like place-and-route files, as long as the instance names in the text file match the instance names in the synthesis tool.

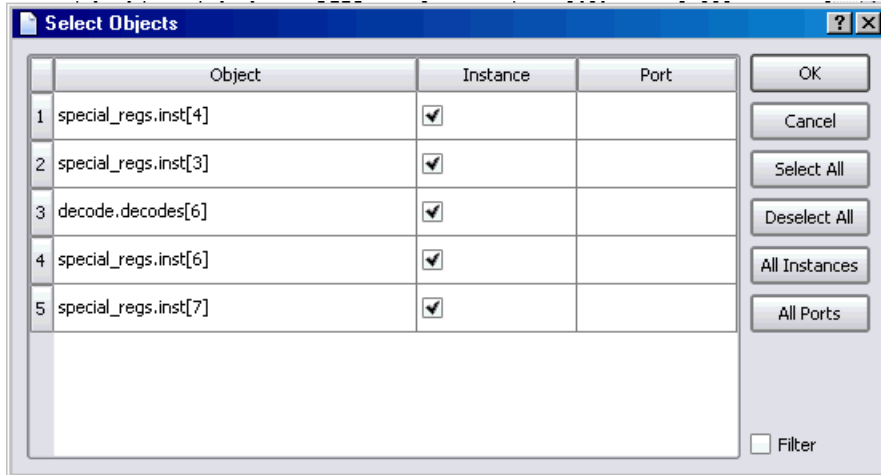
1. Open the log file, the RTL, and Technology views.
2. Select the path objects in the log file.
 - Select the column by pressing Alt and dragging the cursor to the end of the column. On the Linux platform, use the key to which the Alt function is mapped; this is usually the Ctrl-Alt key combination.
 - To select all the objects in the path, right-click and choose **Select in Analyst** from the popup menu. Alternatively, you can select certain objects only, as described next.

The software selects the objects in the column, and highlights the path in the open RTL and Technology views.

Text Editor



- To further filter the objects in the path, right-click and choose **Select From** from the popup menu. On the form, check the objects you want, and click **OK**. Only the corresponding objects are highlighted.



3. To isolate and view only the selected objects, do this in the Technology view: press F12, or right-click and select the Filter Schematic command from the popup menu.

You see just the selected objects.

Crossprobing from the Tcl Script Window

Crossprobing from the Tcl script window is useful for debugging error messages.

To crossprobe from the Tcl Script window to the source code, double-click a line in the Tcl window. To crossprobe a warning or error, first click the Messages tab and then double-click the warning or error. The software opens the relevant source code file and highlights the corresponding code.

Crossprobing from the FSM Viewer

You can crossprobe to the FSM Viewer if you have the FSM view open. You can crossprobe from an RTL, Technology, or source code window.

To crossprobe from the FSM Viewer, do the following:

1. Open the view to which you want to crossprobe: RTL/Technology view, or the source code file.

2. Do the following in the open FSM view:

- For FSMs with a onehot encoding style, click the state bubbles in the bubble diagram or the states in the FSM transition table.
- For all other FSMs, click the states in the bubble diagram. You cannot use the transition table because with these encoding styles, the number of registers in the RTL or Technology views do not match the number of registers in the FSM Viewer.

The software highlights the corresponding code or object in the open views. You can only crossprobe from a state in the FSM table if you used a onehot encoding style.

Analyzing With the Standard HDL Analyst Tool

The HDL Analyst tool is a graphical productivity tool that helps you visualize your synthesis results. It consists of RTL-level and technology-primitive level schematics that let you graphically view and analyze your design.

- **RTL View**
Using BEST® (Behavior Extracting Synthesis Technology) in the RTL view, the software keeps a high-level of abstraction and makes the RTL view easy to view and debug. High-level structures like RAMs, ROMs, operators, and FSMs are kept as abstractions in this view instead of being converted to gates. You can examine the high-level structure, or push into a component and view the gate-level structure.
- **Technology View**
The software uses module generators to implement the high-level structures from the RTL view, and maps them to technology-specific resources.

To analyze information, compare the current view with the information in the RTL/Technology view, the log file, the FSM view, and the source code, you can use techniques like crossprobing, flattening, and filtering. See the following for more information about analysis techniques.

- [Viewing Design Hierarchy and Context](#), on page 337
- [Filtering Schematics](#), on page 340
- [Expanding Pin and Net Logic](#), on page 342
- [Expanding and Viewing Connections](#), on page 346
- [Flattening Schematic Hierarchy](#), on page 347
- [Minimizing Memory Usage While Analyzing Designs](#), on page 352

For additional information about navigating the HDL Analyst views or using other techniques like crossprobing, see the following:

- [Working in the Schematic](#), on page 232
- [Exploring Design Hierarchy](#), on page 307
- [Finding Objects \(Standard\)](#), on page 315
- [Crossprobing \(Standard\)](#), on page 328

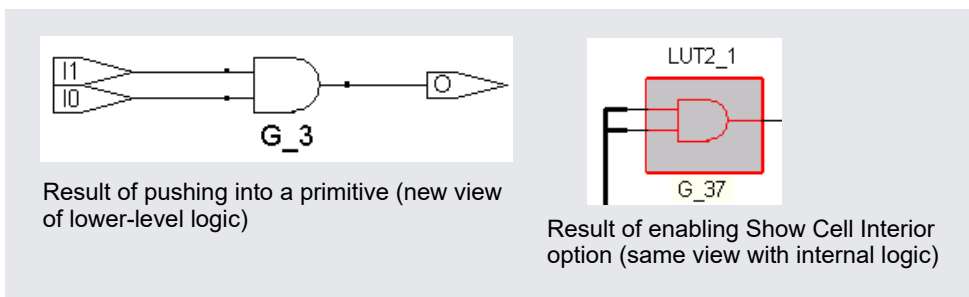
Viewing Design Hierarchy and Context

Most large designs are hierarchical, so the synthesis software provides tools that help you view hierarchy details or put the details in context. Alternatively, you can browse and navigate hierarchy with Push/Pop mode, or flatten the design to view internal hierarchy.

This section describes how to use interactive hierarchical viewing operations to better analyze your design. Automatic hierarchy viewing operations that are built into other commands are described in the context in which they appear. For example, [Viewing Critical Paths, on page 363](#) describes how the software automatically traces a critical path through different hierarchical levels using hollow boxes with nested internal logic (transparent instances) to indicate levels in hierarchical instances.

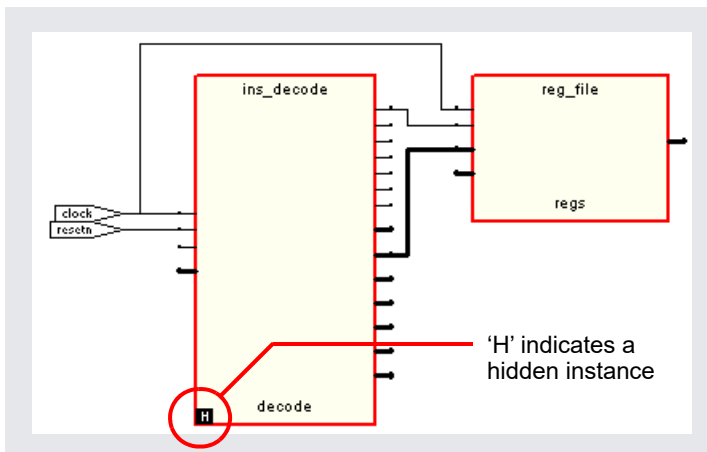
1. To view the internal logic of primitives in your design, do either of the following:
 - To view the logic of an individual primitive, push into it. This generates a new schematic with the internal details. Click the Back icon to return to the previous view.
 - To view the logic of all primitives in the design, select Options->HDL Analyst Options->General, and enable Show Cell Interior. This command lets you see internal logic in context, by adding the internal details to the current schematic and all subsequent views. If the view is too cluttered with this option on, filter the view (see [Filtering Schematics, on page 340](#)) or push into the primitive. Click the Back icon to return to the previous view after filtering or pushing into the object.

The following figure compares these two methods:



2. To hide selected hierarchy, select the instance whose hierarchy you want to exclude, and then select Hide Instances from the HDL Analyst menu or the right-click popup menu in the schematic.

You can hide opaque (solid yellow) or transparent (hollow) instances. The software marks hidden instances with an H in the lower left. Hidden instances are like black boxes; their hierarchy is excluded from filtering, expanding, dissolving, or searching in the current window, although they can be crossprobed. An instance is only hidden in the current view window; other view windows are not affected. Temporarily hiding unnecessary hierarchy focuses analysis and saves time in large designs.



Before you save a design with hidden instances, select Unhide Instances from the HDL Analyst menu or the right-click popup menu and make the hidden internal hierarchy accessible again. Otherwise, the hidden instances are saved as black boxes, without their internal logic. Conversely, you can use this feature to reduce the scope of analysis in a large design by hiding instances you do not need, saving the reduced design to a new name, and then analyzing it.

3. To view the internal logic of a hierarchical instance, you can push into the instance, dissolve the selected instance with the Dissolve Instances command, or flatten the design. You cannot use these methods to view the internal logic of a hidden instance.

Pushing into an instance	Generates a view that shows only the internal logic. You do not see the internal hierarchy in context. To return to the previous view, click Back. See Exploring Object Hierarchy by Pushing/Popping , on page 308 for details.
Flattening the entire design	Opens a new view where the entire design is flattened, except for hidden hierarchy. Large flattened designs can be overwhelming. See Flattening Schematic Hierarchy , on page 347 for details about flattening designs. Because this is a new view, you cannot use Back to return to the previous view. To return to the top-level unflattened schematic, right-click in the view and select Unflatten Schematic.
Flattening an instance by dissolving	Generates a view where the hierarchy of the selected instances is flattened, but the rest of the design is unaffected. This provides context. See Flattening Schematic Hierarchy , on page 347 for details about dissolving instances.

4. If the result of filtering or dissolving is a hollow box with no internal logic, try either of the following, as appropriate, to view the internal hierarchy:
 - Select Options->HDL Analyst Options->Sheet Size and increase the value of Maximum Filtered Instances. Use this option if the view is not too cluttered.
 - Use the sheet navigation commands to go to the sheets indicated in the hollow box.

If there is too much internal logic to display in the current view, the software puts the internal hierarchy on separate schematic sheets. It displays a hollow box with no internal logic and indicates the schematic sheets that contain the internal logic.

5. To view the design context of an instance in a filtered view, select the instance, right-click, and select Show Context from the popup menu.

The software displays an unfiltered view of the hierarchical level that contains the selected object, with the instance highlighted. This is useful when you have to go back and forth between different views during analysis. The context differs from the Expand commands, which show connections. To return to the original filtered view, click Back.

Filtering Schematics

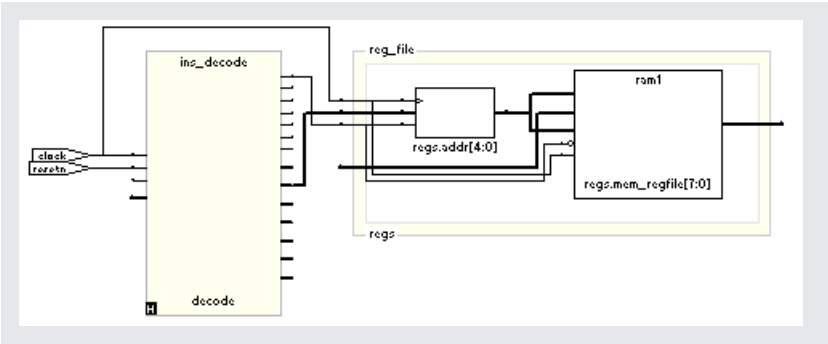
Filtering is a useful first step in analysis, because it focuses analysis on the relevant parts of the design. Some commands, like the Expand commands, automatically generate filtered views; this procedure only discusses manual filtering, where you use the Filter Schematic command to isolate selected objects. See Chapter 3 of the *Reference Manual* for details about these commands.

This table lists the advantages of using filtering over flattening:

Filter Schematic Command	Flatten Commands
Loads part of the design; better memory usage	Loads entire design
Combine filtering with Push/Pop mode, and history buttons (Back and Forward) to move freely between hierarchical levels	Must use Unflatten Schematic to return to top level, and flatten the design again to see lower levels. Cannot return to previous view if the previous view is not the top-level view.

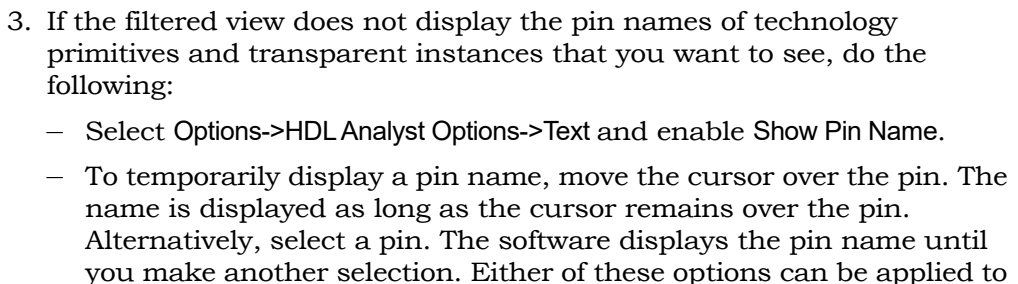
1. Select the objects that you want to isolate. For example, you can select two connected objects.

If you filter a hidden instance, the software does not display its internal hierarchy when you filter the design. The following example illustrates this.



2. Select the Filter Schematic command, using one of these methods:
 - Select Filter Schematic from the HDL Analyst menu or the right-click popup menu.
 - Click the Filter Schematic icon (buffer gate) ().

- The software filters the design and displays the selected objects in a filtered view. The title bar indicates that it is a filtered view. Hidden instances have an H in the lower left. The view displays other hierarchical instances as hollow boxes with nested internal logic (transparent instances). For descriptions of filtered views and transparent instances, see *Filtered and Unfiltered Schematic Views, on page 86* and *Transparent and Opaque Display of Hierarchical Instances, on page 91* in the *Reference Manual*. If the transparent instance does not display internal logic, use one of the alternatives described in *Viewing Design Hierarchy and Context, on page 337*, step 4.



individual pins. Use them to view just the pin names you need and keep design clutter to a minimum.

- To see all the hierarchical pins, select the instance, right-click, and select Show All Hier Pins.

You can now analyze the problem, and do operations like the following:

Trace paths, build up logic	See Expanding Pin and Net Logic , on page 342 and Expanding and Viewing Connections , on page 346
Filter further	Select objects and filter again
Find objects	See Finding Objects (Standard) , on page 315
Flatten, or hide and flatten	See Flattening Schematic Hierarchy , on page 347. You can hide transparent or opaque instances.
Crossprobe from filtered view	See Crossprobing from the RTL/Technology View , on page 329

4. To return to the previous schematic, click the Back icon. If you flattened the hierarchy, right-click and select Unflatten Schematic to return to the top-level unflattened view.

For additional information about filtering schematics, see [Filtering Schematics](#), on page 340 and [Flattening Schematic Hierarchy](#), on page 347.

Expanding Pin and Net Logic

When you are working in a filtered view, you might need to include more logic in your selected set to debug your design. This section describes commands that expand logic fanning out from pins or nets; to expand paths, see [Expanding and Viewing Connections](#), on page 346.

Use the Expand commands with the Filter Schematic, Hide Instances, and Flatten commands to isolate just the logic that you want to examine. Filtering isolates logic, flattening removes hierarchy, and hiding instances prevents their internal hierarchy from being expanded. See [Filtering Schematics](#), on page 340 and [Flattening Schematic Hierarchy](#), on page 347 for details.

1. To expand logic from a pin hierarchically across boundaries, use the following commands.

To...	Do this (HDL Analyst->Hierarchical/Popup menu)...
See all cells connected to a pin	Select a pin and select Expand. See Expanding Filtered Logic Example , on page 344.
See all cells that are connected to a pin, up to the next register	Select a pin and select Expand to Register/Port. See Expanding Filtered Logic to Register/Port Example , on page 345.
See internal cells connected to a pin	Select a pin and select Expand Inwards. The software filters the schematic and displays the internal cells closest to the port. See Expanding Inwards Example , on page 345.

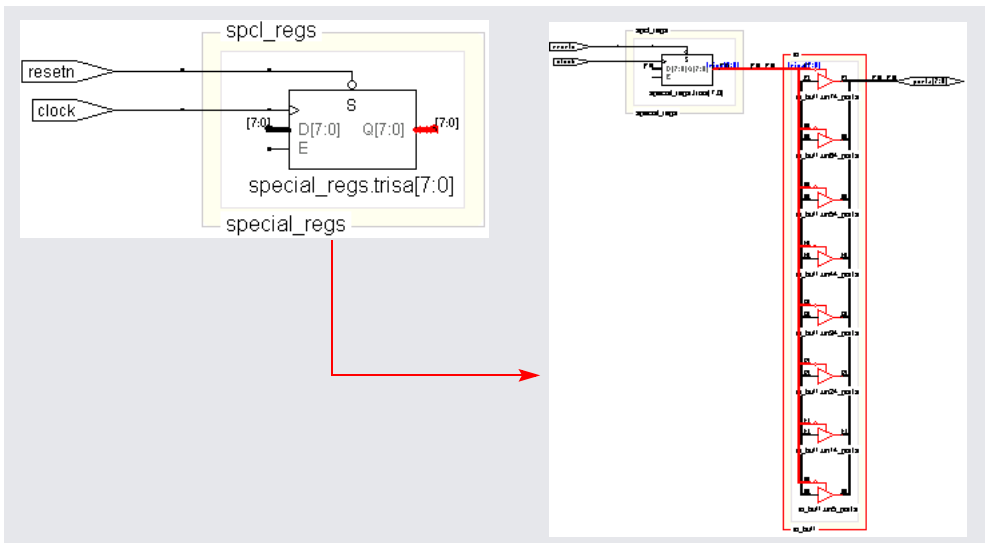
The software expands the logic as specified, working on the current level and below or working up the hierarchy, crossing hierarchical boundaries as needed. Hierarchical levels are shown nested in hollow bounding boxes. The internal hierarchy of hidden instances is not displayed.

For descriptions of the Expand commands, see [HDL Analyst Menu](#), on page 451 of the *Reference Manual*.

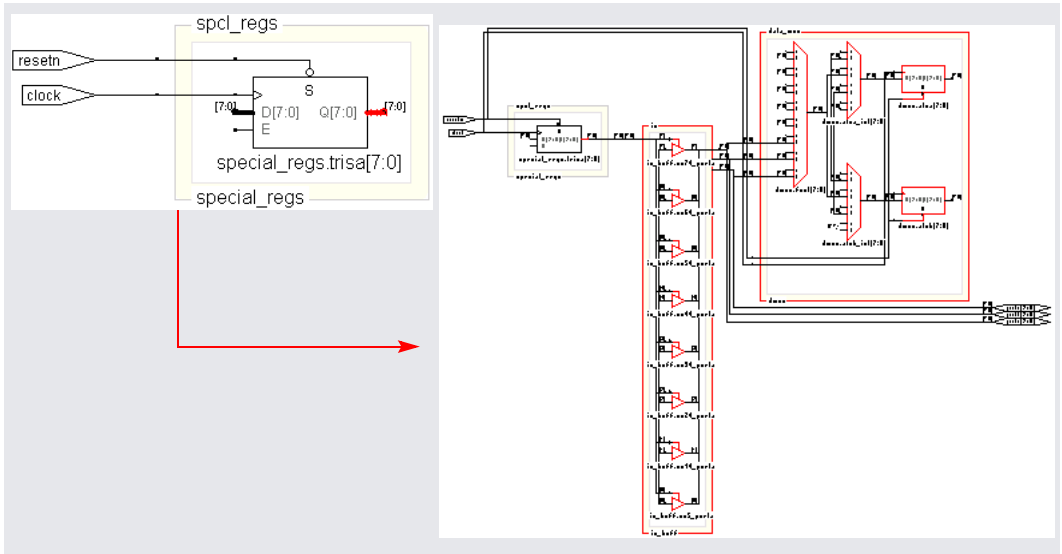
2. To expand logic from a pin at the current level only, do the following:
 - Select a pin, and go to the HDL Analyst->Current Level menu or the right-click popup menu->Current Level.
 - Select Expand or Expand to Register/Ports. The commands work as described in the previous step, but they do not cross hierarchical boundaries.
3. To expand logic from a net, use the commands shown in the following table.
 - To expand at the current level and below, select the commands from the HDL Analyst->Hierarchical menu or the right-click popup menu.
 - To expand at the current level only, select the commands from the HDL Analyst->Current Level menu or the right-click popup menu->Current Level.

To...	Do this...
Select the driver of a net	Select a net and select Select Net Driver. The result is a filtered view with the net driver selected (Selecting the Net Driver Example , on page 346).
Trace the driver, across sheets if needed	Select a net and select Go to Net Driver. The software shows a view that includes the net driver.
Select all instances on a net	Select a net and select Select Net Instances. You see a filtered view of all instances connected to the selected net.

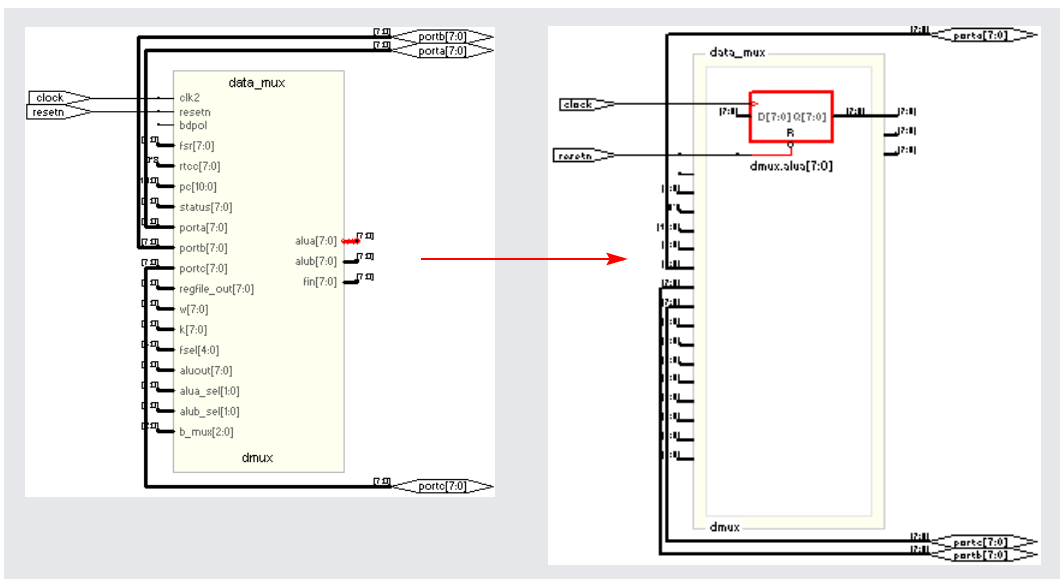
Expanding Filtered Logic Example



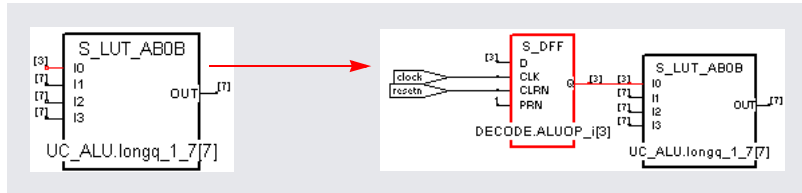
Expanding Filtered Logic to Register/Port Example



Expanding Inwards Example



Selecting the Net Driver Example



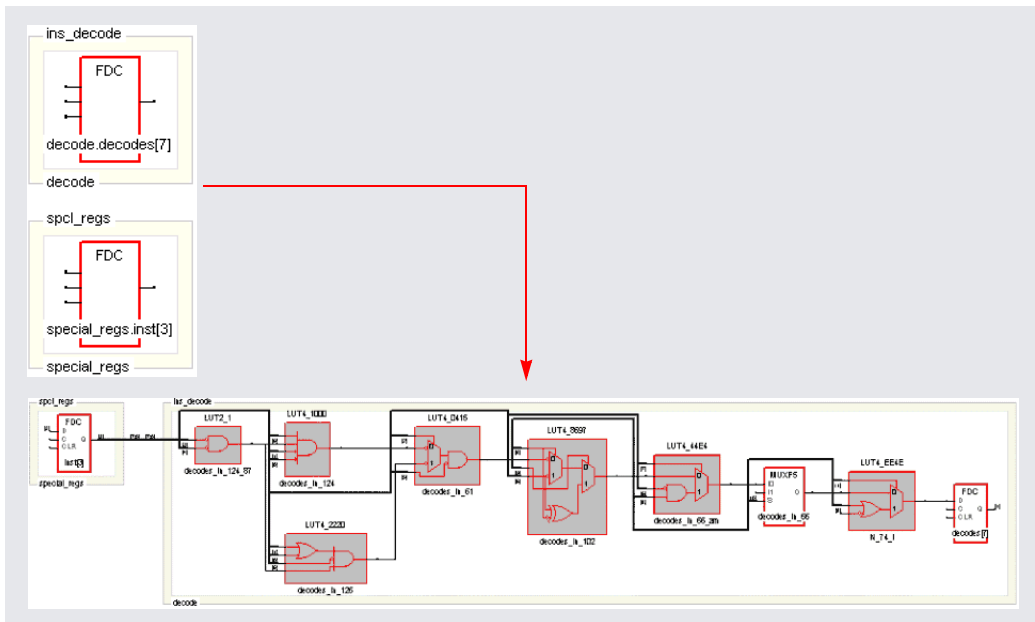
Expanding and Viewing Connections

This section describes commands that expand logic between two or more objects; to expand logic out from a net or pin, see [Expanding Pin and Net Logic, on page 342](#). You can also isolate the critical path or use the Timing Analyst to generate a schematic for a path between objects, as described in [Analyzing Timing in Schematic Views, on page 360](#).

Use the following path commands with the Filter Schematic and Hide Instances commands to isolate just the logic that you want to examine. The two techniques described here differ: Expand Paths expands connections between selected objects, while Isolate Paths pares down the current view to only display connections to and from the selected instance.

For detailed descriptions of the commands mentioned here, see [Commands That Result in Filtered Schematics, on page 112](#) in the *Reference Manual*.

1. To expand and view connections between selected objects, do the following:
 - Select two or more points.
 - To expand the logic at the current level only, select HDL Analyst->Current Level->Expand Paths or popup menu->Current Level Expand Paths.
 - To expand the logic at the current level and below, select HDL Analyst->Hierarchical->Expand Paths or popup menu->Expand Paths.



2. To view connections from all pins of a selected instance, right-click and select **Isolate Paths** from the popup menu.

Starting Point	The Filtered View Traces Paths (Forward and Back) From All Pins of the Selected Instance...

Filtered view	Traces through all sheets of the filtered view, up to the next port, register, hierarchical instance, or black box.

Unfiltered view	Traces paths on the current schematic sheet only, up to the next port, register, hierarchical instance, or black box.

Unlike the **Expand Paths** command, the connections are based on the schematic used as the starting point; the software does not add any objects that were not in the starting schematic.

Flattening Schematic Hierarchy

Flattening removes hierarchy so you can view the logic without hierarchical levels. In most cases, you do not have to flatten your hierarchical schematic to debug and analyze your design, because you can use a combination of

filtering, Push/Pop mode, and expanding to view logic at different levels. However, if you must flatten the design, use the following techniques, which include flattening, dissolving, and hiding instances.

1. To flatten an entire design down to logic cells, use one of the following commands:
 - For an RTL view, select HDL Analyst->RTL->Flattened View. This flattens the design to generic logic cells.
 - For a Technology view, select Flattened View or Flattened to Gates View from the HDL Analyst->Technology menu. Use the former command to flatten the design to the technology primitive level, and the latter command to flatten it further to the equivalent Boolean logic.

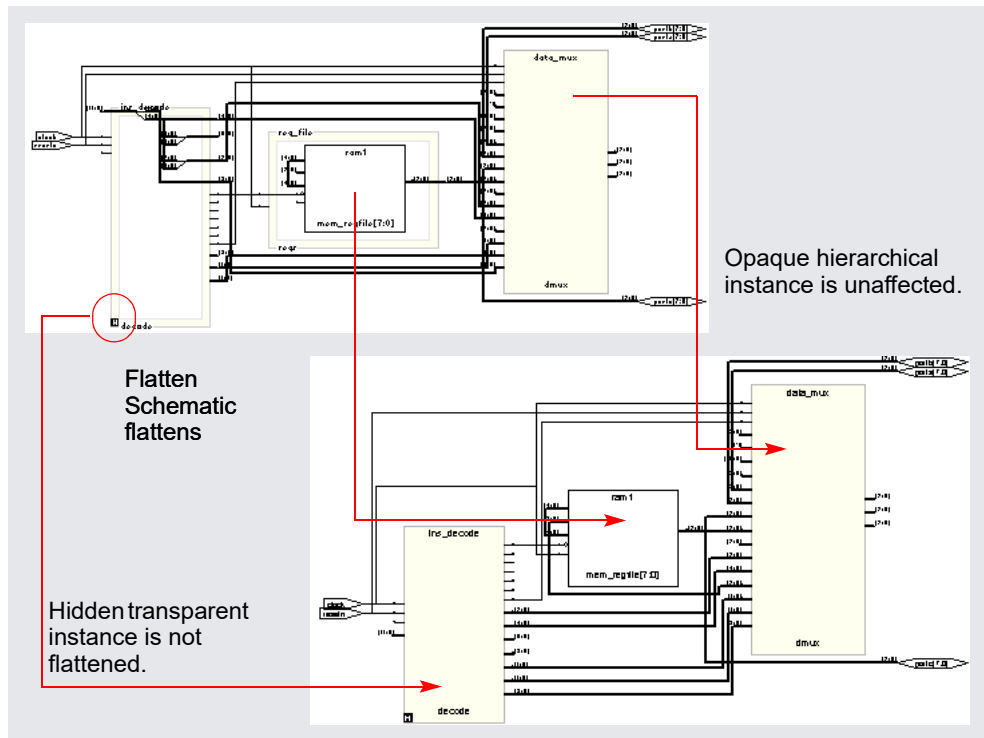
The software flattens the top-level design and displays it in a new window. To return to the top-level design, right-click and select Unflatten Schematic.

Unless you really require the entire design to be flattened, use Push/Pop mode and the filtering commands ([Filtering Schematics, on page 340](#)) to view the hierarchy. Alternatively, you can use one of the selective flattening techniques described in subsequent steps.

2. To selectively flatten transparent instances when you analyze critical paths or use the Expand commands, select Flatten Current Schematic from the HDL Analyst menu, or select Flatten Schematic from the right-click popup menu.

The software generates a new view of the current schematic in the same window, with all transparent instances at the current level and below flattened. RTL schematics are flattened down to generic logic cells and Technology views down to technology primitives. To control the number of hierarchical levels that are flattened, use the Dissolve Instances command described in step 4.

If your view only contains hidden hierarchical instances or pale yellow (opaque) hierarchical instances, nothing is flattened. If you flatten an unfiltered (usually the top-level design) view, the software flattens all hierarchical instances (transparent and opaque) at the current level and below. The following figure shows flattened transparent instances.



Because the flattened view is a new view, you cannot use Back to return to the unflattened view or the views before it. Use Unflatten Schematic to return to the unflattened top-level view.

3. To selectively flatten the design by hiding instances, select hierarchical instances whose hierarchy you do not want to flatten, right-click, and select Hide Instances. Then flatten the hierarchy using one of the Flatten commands described above.

Use this technique if you want to flatten most of your design. If you want to flatten only part of your design, use the approach described in the next step.

When you hide instances, the software generates a new view where the hidden instances are not flattened, but marked with an H in the lower left corner. The rest of the design is flattened. If unhidden hierarchical instances are not flattened by this procedure, use the Flattened View or Flattened to Gates View commands described in step 1 instead of the Flatten

Current Schematic command described in step 2, which only flattens transparent instances in filtered views.

You can select the hidden instances, right-click, and select Unhide Instances to make their hierarchy accessible again. To return to the unflattened top-level view, right-click in the schematic and select Unflatten Schematic.

4. To selectively flatten some hierarchical instances in your design by dissolving them, do the following:
- If you want to flatten more than one level, select Options->HDL Analyst Options and change the value of Dissolve Levels. If you want to flatten just one level, leave the default setting.
 - Select the instances to be flattened.
 - Right-click and select Dissolve Instances.

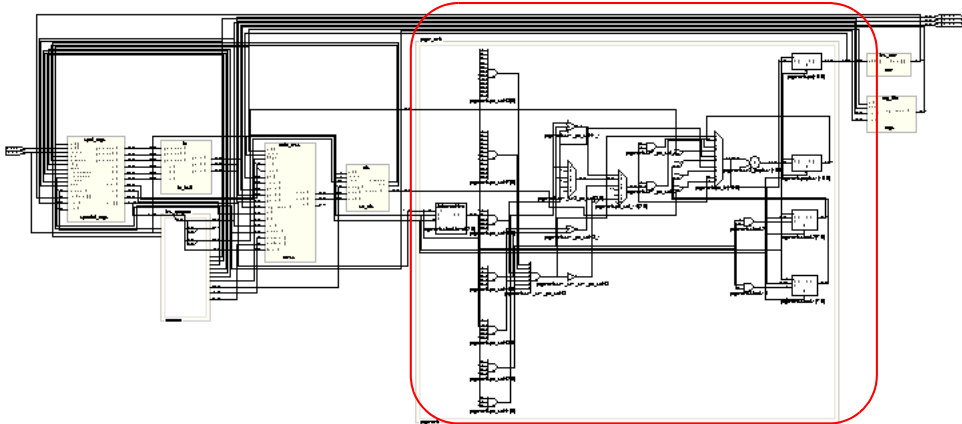
The results differ slightly, depending on the kind of view from which you dissolve instances.

Starting View Software Generates a...

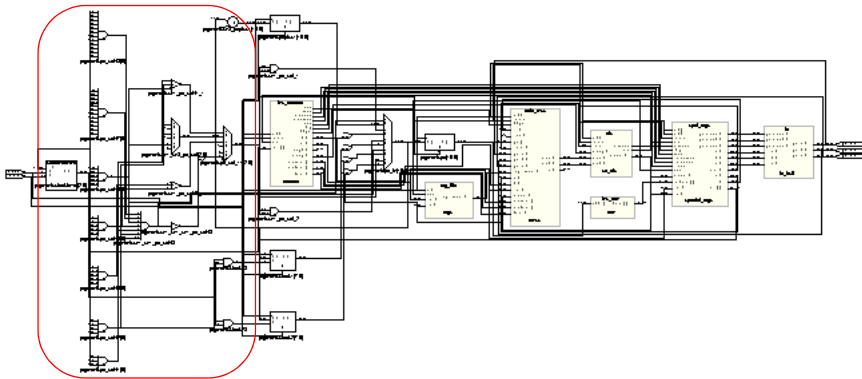
Filtered	Filtered view with the internal logic of dissolved instances displayed within hollow bounding boxes (transparent instances), and the hierarchy of the rest of the design unchanged. If the transparent instance does not display internal logic, use one of the alternatives described in step 4 of Viewing Design Hierarchy and Context , on page 337. Use the Back button to return to the undissolved view.
Unfiltered	New, flattened view with the dissolved instances flattened in place (no nesting) to Boolean logic, and the hierarchy of the rest of the design unchanged. Select Unflatten Schematic to return to the top-level unflattened view. You cannot use the Back button to return to previous views because this is a new view.

The following figure illustrates this.

Dissolved logic for prgmctr shown nested when started from filtered view



Dissolve



Use this technique if you only want to flatten part of your design while retaining the hierarchical context. If you want to flatten most of the design, use the technique described in the previous step. Instead of dissolving instances, you can use a combination of the filtering commands and Push/Pop mode.

Minimizing Memory Usage While Analyzing Designs

When working with large hierarchical designs, use the following techniques to use memory resources efficiently.

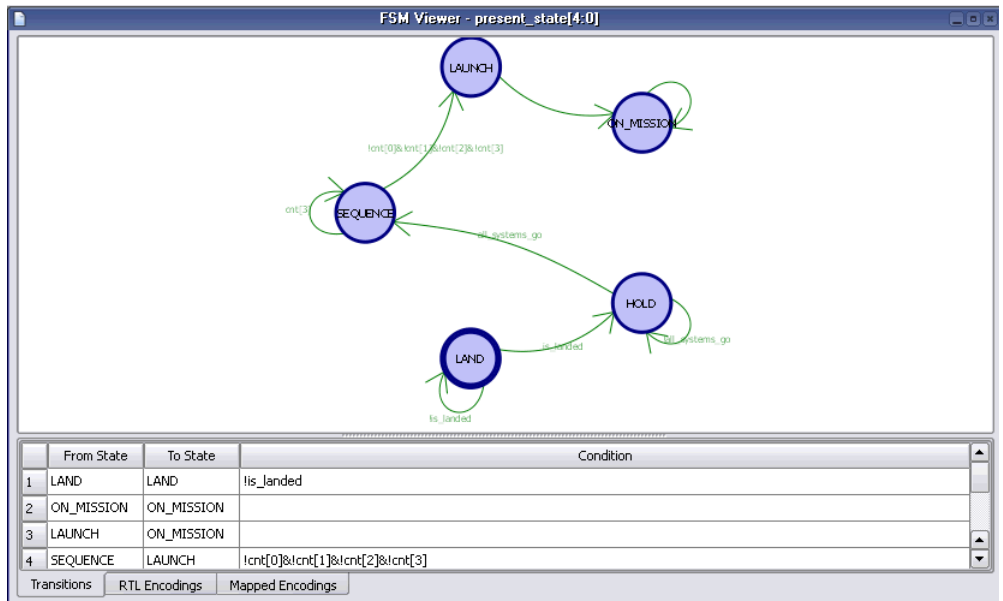
- Before you do any analysis operations such as searching, flattening, expanding, or pushing/popping, hide (HDL Analyst->Hide Instances) the hierarchical instances you do not need. This saves memory resources, because the software does not load the hierarchy of the hidden instances.
- Temporarily divide your design into smaller working files. Before you do any analysis, hide the instances you do not need. Save the design. The `srs` and `srm` files generated are smaller because the software does not save the hidden hierarchy. Close any open HDL Analyst windows to free all memory from the large design. In the Implementation Results view, double-click one of the smaller files to open the RTL or Technology schematic. Analyze the design using the smaller, working schematics.
- Filter your design instead of flattening it. If you must flatten your design, hide the instances whose hierarchy you do not need before flattening, or use the Dissolve Instances command. See [Flattening Schematic Hierarchy, on page 347](#) for details. For more information on the Expand Paths and Isolate Paths commands, see [RTL and Technology Views Popup Menus, on page 526](#) of the *Reference Manual*.
- When searching your design, search by instance rather than by net. Searching by net loads the entire design, which uses memory.
- Limit the scope of a search by hiding instances you do not need to analyze. You can limit the scope further by filtering the schematic in addition to hiding the instances you do not want to search.

Using the FSM Viewer (Standard)

The FSM viewer displays state transition bubble diagrams for FSMs in the design, along with additional information about the FSM. You can use this viewer to view state machines implemented by the FSM Compiler.

1. To start the FSM viewer, open the RTL view and either
 - Select the FSM instance, click the right mouse button and select View FSM from the popup menu.
 - Push down into the FSM instance (Push/Pop icon).

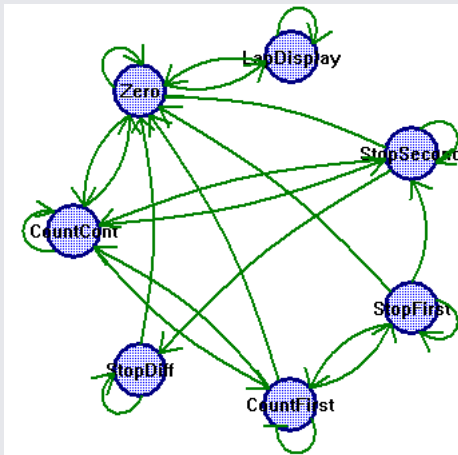
The FSM viewer opens. The viewer consists of a transition bubble diagram and a table for the encodings and transitions. If you used Verilog to define the FSMs, the viewer displays binary values for the state machines if you defined them with the 'define keyword, and actual names if you used the parameter keyword.



2. The following table summarizes basic viewing operations.

To view...	Do...
from and to states, and conditions for each transition	Click the Transitions tab at the bottom of the table.
the correspondence between the states and the FSM registers in the RTL view	Click the RTL Encoding tab.
the correspondence between the states and the registers in the Technology View	Click the Mapped Encodings tab (available after synthesis).
only the transition diagram without the table	Select View->FSM table or click the FSM Table icon. You might have to scroll to the right to see it.

This figure shows you the mapping information for a state machine. The Transitions tab shows you simple equations for conditions for each state. The RTL Encodings tab has a State column that shows the state names in the source code, and a Registers column for the corresponding RTL encoding. The Mapped Encoding tab shows the state names in the code mapped to actual values.



	From State	To State	Condition
1	LapDisplay	LapDisplay	Laplevel&Reset
2	Zero	LapDisplay	Lap&Start&Reset
3	StopSecond	StopSecond	!Lap&Start&Reset
4	StopFirst	StopSecond	Lap&Start&Reset
5	CountCont	StopSecond	Start&Reset
6	StopSecond	StopDiff	Lap&Start&Reset
7	StopDiff	StopDiff	!Lap&Start&Reset
8	StopFirst	StopFirst	!Lap&Start&Reset
9	CountFirst	StopFirst	Start&Reset
10	StopSecond	CountCont	Start&Reset

States and Conditions

State	cur_state[6]	cur_state[5]	cur_state[4]	cur_state[3]	cur_state[2]	cur_state[1]	cur_state[0]
Zero	0	0	0	0	0	0	1
CountFirst	0	0	0	0	0	1	0
CountCont	0	0	0	0	1	0	0
StopFirst	0	0	0	1	0	0	0
StopDiff	0	0	1	0	0	0	0
StopSecond	0	1	0	0	0	0	0
LapDisplay	1	0	0	0	0	0	0

Mapped Encoding

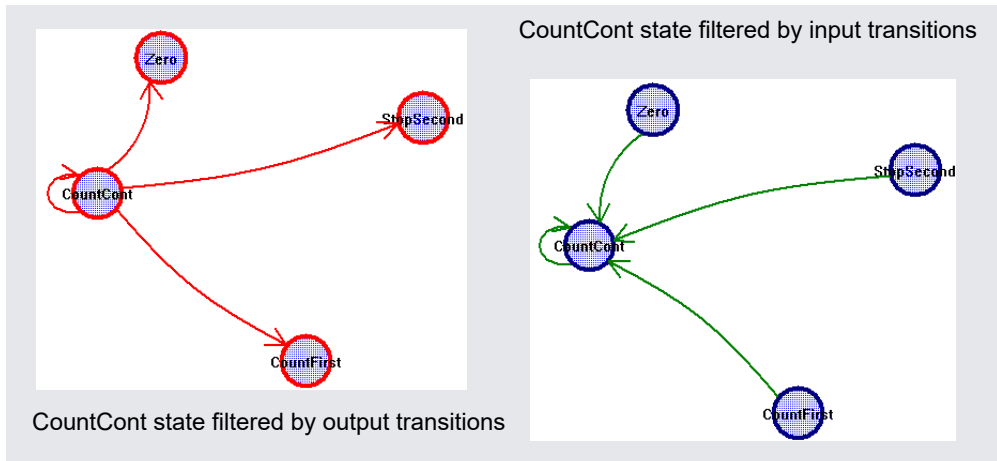
State	Register
Zero	cur_state[0]
CountFirst	cur_state[1]
CountCont	cur_state[2]
StopFirst	cur_state[3]
StopDiff	cur_state[4]
StopSecond	cur_state[5]
LapDisplay	cur_state[6]

RTL Encoding

3. To view just one selected state,

- Select the state by clicking on its bubble. The state is highlighted.
- Click the right mouse button and select the filtering criteria from the popup menu: output, input, or any transition.

The transition diagram now shows only the filtered states you set. The following figure shows filtered views for output and input transitions for one state.



Similarly, you can check the relationship between two or more states by selecting the states, filtering them, and checking their properties.

4. To view the properties for a state,
 - Select the state.
 - Click the right mouse button and select Properties from the popup menu. A form shows you the properties for that state.

To view the properties for the entire state machine like encoding style, number of states, and total number of transitions between states, deselect any selected states, click the right mouse button outside the diagram area, and select Properties from the popup menu.

5. To view the FSM description in text format, select the state machine in the RTL view and View FSM Info File from the right mouse popup. This is an example of the FSM Info File, *statemachine.info*.

```
State Machine: work.Control(verilog)-cur_state[6:0]
No selected encoding - Synplify will choose
Number of states: 7
Number of inputs: 4
Inputs:
  0: Laplevel
  1: Lap
  2: Start
  3: Reset
Clock: Clk
```

Transitions: (input, start state, destination state)

```
-100 S0 S6
--10 S0 S2
---1 S0 S0
-00- S0 S0
--10 S1 S3
-100 S1 S2
-000 S1 S1
---1 S1 S0
--10 S2 S5
-000 S2 S2
-100 S2 S1
---1 S2 S0
-100 S3 S5
-000 S3 S3
--10 S3 S1
---1 S3 S0
-000 S4 S4
--1- S4 S0
-1-- S4 S0
---1 S4 S0
-000 S5 S5
-100 S5 S4
--10 S5 S2
---1 S5 S0
1--0 S6 S6
---1 S6 S0
0--- S6 S0
```


CHAPTER 8

Analyzing Timing

This chapter describes typical analysis tasks. It describes graphical analysis with the HDL Analyst tool as well as interpretation of the text log file. It covers the following:

- [Analyzing Timing in Schematic Views](#), on page 360
- [Generating Custom Timing Reports with STA](#), on page 367
- [Using Analysis Design Constraints](#), on page 370
- [Using Auto Constraints](#), on page 377

Analyzing Timing in Schematic Views

You can use the HDL Analyst and Timing Analyst functionality to analyze timing. This section describes the following:

- [Viewing Timing Information](#), on page 360
- [Annotating Timing Information in the Schematic Views](#), on page 361
- [Analyzing Clock Trees in the RTL View](#), on page 363
- [Viewing Critical Paths](#), on page 363
- [Handling Negative Slack](#), on page 366
- [Generating Custom Timing Reports with STA](#), on page 367

Viewing Timing Information

Some commands, like Show Critical Path, Hierarchical Critical Path, Flattened Critical Path, automatically enable Show Timing Information and display the timing information. The following procedure shows you how to do so manually.

1. To analyze timing, enable HDL Analyst->Show Timing Information.

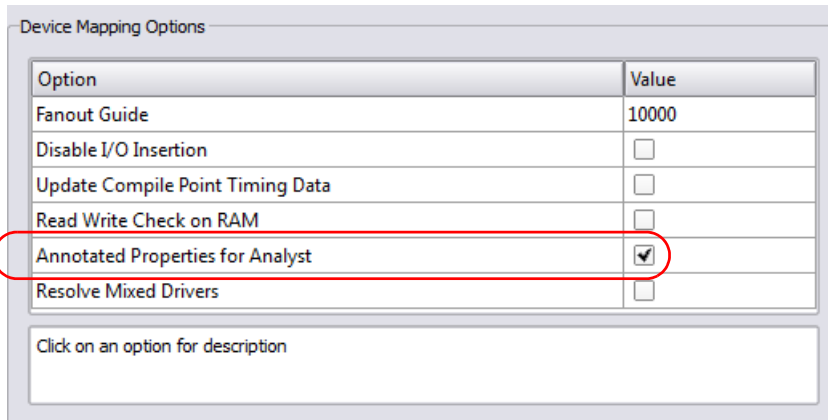
This displays the timing numbers for all instances in a Technology view. It shows the following:

Delay	This is the first number displayed. • Combinational logic This first number is the cumulative path delay to the output of the instance, which includes the net delay of the output. • Flip-flops This first number is the path delay attributed to the flip-flop. The delay can be associated with either the input or output path, whichever is worse, because the flip-flop is the end of one path and the start of another.
Slack Time	This is the second number, and it is the slack time of the worst path that goes through the instance. A negative value indicates that timing constraints can not be met.

Annotating Timing Information in the Schematic Views

You can annotate the schematic views with timing information for the components in the design. Once the design is annotated, you can search for these properties and their associated instances.

1. On the Device tab of the Implementation Options dialog box, enable Annotated Properties for Analyst. This feature is supported on Lattice iCE devices.



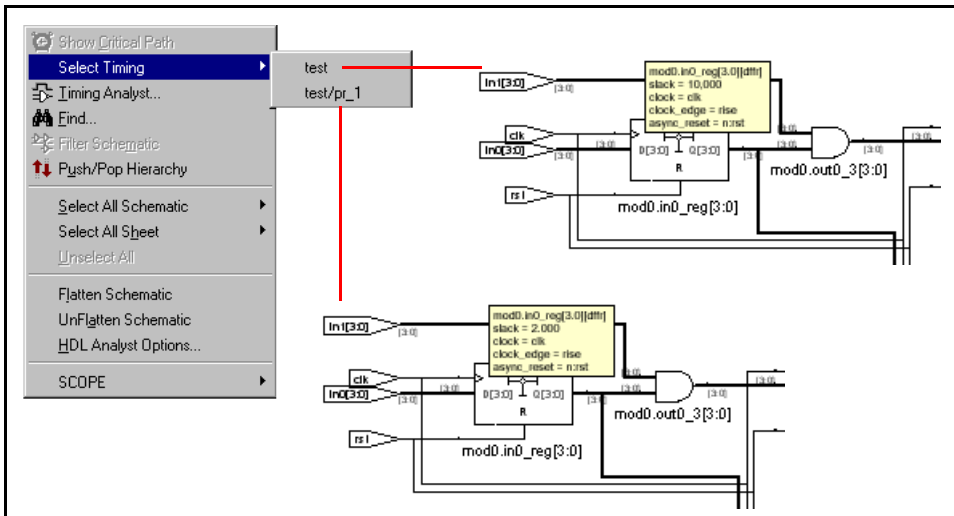
For each synthesis implementation and each place-and-route implementation, the tool generates properties and stores them in two files located in the project folder:

.sap	Synplify Annotated Properties Contains the annotated design properties generated after compilation, like clock pins.
.tap	Timing Annotated Properties Contains the annotated timing properties generated after compilation.

2. To view the annotated timing, open an RTL or Technology view.
3. To view the timing information from another associated implementation, do the following:
 - Open an RTL or Technology view. It displays the timing information for that implementation.
 - Select HDL Analyst->Select Timing, and select another implementation from the list. The list contains the main implementation and all

associated place-and-route implementations. The timing numbers in the current Analyst view change to reflect the numbers from the selected implementation.

In the following example, an RTL View shows timing data from the test implementation and the test/pr_1 (place and route) implementation.



4. Once you have annotated your design, you can filter searches using these properties with the find command.
 - Use the find -filter `{@propName>=propValue}` command for the searches. See [Find Filter Properties, on page 166](#) in the *Command Reference Manual* for a list of properties. For information about the find command, see [find, on page 151](#) in the *Command Reference Manual*.
 - Precede the property name with the @ symbol.

For example to find fanouts larger than 60, specify find -filter `{@fanout>=60}`.

Analyzing Clock Trees in the RTL View

To analyze clock trees in the RTL view, do the following:

1. In the Hierarchy Browser, expand Clock Tree, select all the clocks, and filter the design.

The Hierarchy Browser lists all clocks and the instances that drive them under Clock Tree. The filtered view shows the selected objects.

2. If necessary, use the filter and expand commands to trace clock connections back to the ports and check them.

For details about the commands for filtering and expanding paths, see [Filtering Schematics, on page 340](#), [Expanding Pin and Net Logic, on page 342](#) and [Expanding and Viewing Connections, on page 346](#).

3. Check that your defined clock constraints cover the objects in the design.

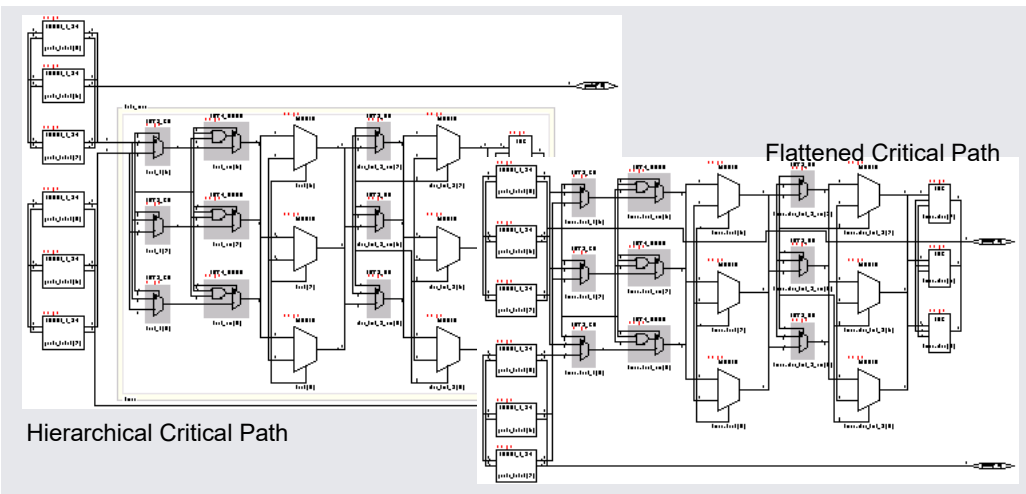
If you do not define your clock constraints accurately, you might not get the best possible synthesis optimizations.

Viewing Critical Paths

The HDL Analyst tool makes it simple to find and examine critical paths and the relevant source code. The following procedure shows you how to filter and analyze a critical path. You can also use the procedure described in [Generating Custom Timing Reports with STA, on page 367](#) to view this and other paths.

1. If needed, set the slack time for your design.
 - Select HDL Analyst->Set Slack Margin.
 - To view only instances with the worst-case slack time, enter a zero.
 - To set a slack margin range, type a value for the slack margin, and click OK. The software gets a range by subtracting this number from the slack time, and the Technology view displays instances within this range. For example, if your slack time is -10 ns, and you set a slack margin of 4 ns, the command displays all instances with slack times between -6 ns and -10 ns. If your slack margin is 6 ns, you see all instances with slack times between -4 ns and -10 ns.

2. Display the critical path using one of the following methods. The Technology view displays a hierarchical view that highlights the instances and nets in the most critical path of your design.
 - To generate a hierarchical view of the critical path, click the Show Critical Path icon (stopwatch icon (🕒)), select HDL Analyst->Technology->Hierarchical Critical Path, or select the command from the popup menu. This is a filtered view in the same window, with hierarchical logic shown in transparent instances. History commands apply, so you can return to the previous view by clicking Back.
 - To flatten the hierarchical critical path described above, right-click and select Flatten Schematic. The software generates a new view in the current window, and flattens only the transparent instances needed to show the critical path; the rest of the design remains hierarchical. Click Back to go the top-level design.
 - To generate a flattened critical path in a new window, select HDL Analyst->Technology->Flattened Critical Path. This command uses more memory because it flattens the entire design and generates a new view for the flattened critical path in a new window. Click Back in this window to go to the flattened top-level design or to return to the previous window.



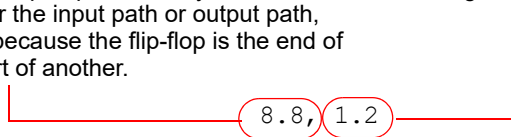
3. Use the timing numbers displayed above each instance to analyze the path. If no numbers are displayed, enable HDL Analyst->Show Timing Information. Interpret the numbers as follows:

Delay

For combinational logic, it is the cumulative delay to the output of the instance, including the net delay of the output. For flip-flops, it is the portion of the path delay attributed to the flip-flop. The delay can be associated with either the input path or output path, whichever is worse, because the flip-flop is the end of one path and the start of another.

Slack time

Slack of the worst path that goes through the instance. A negative value indicates that timing has not been met.



4. View instances in the critical path that have less than the worst-case slack time. For additional information on handling slack times, see [Handling Negative Slack, on page 366](#).

If necessary change the slack margin and regenerate the critical path.

5. Crossprobe and check the RTL view and source code. Analyze the code and the schematic to determine how to address the problem. You can add more constraints or make code changes.
6. Click the Back icon to return to the previous view. If you flattened your design during analysis, select Unflatten Schematic to return to the top-level design.

There is no need to regenerate the critical path, unless you flattened your design during analysis or changed the slack margin. When you flatten your design, the view is regenerated so the history commands do not apply and you must click the Critical Path icon again to see the critical path view.

7. Rerun synthesis, and check your results.

If you have fixed the path, the window displays the next most critical path when you click the icon.

Repeat this procedure and fix the design for the remaining critical paths. When you are within 5-10 percent of your desired results, place and route your design to see if you meet your goal. If so, you are done. If your vendor provides timing-driven place and route, you might improve your results further by adding timing constraints to place and route.

Handling Negative Slack

Positive slack time values (greater than or equal to 0 ns) are good, while negative slack time values (less than 0 ns) indicate the design has not met timing requirements. The negative slack value indicates the amount by which the timing is off because of delays in the critical paths of your design.

The following procedure shows you how to add constraints to correct negative slack values. Timing constraints can improve your design by 10 to 20 percent.

1. Display the critical path in a filtered Technology view.
 - For a hierarchical critical path, either click the Critical Path icon, select HDL Analyst->Show Critical Path, or select HDL Analyst->Technology->Hierarchical Critical Path.
 - For a flat path, select HDL Analyst->Technology->Flattened Critical Path.
2. Analyze the critical path.
 - Check the end points of the path. The start point can be a primary input or a flip-flop. The end point can be a primary output or a flip-flop.
 - Examine the instances. Use the commands described in [Expanding Pin and Net Logic](#), on page 342 and [Expanding and Viewing Connections](#), on page 346. For more information on filtering schematics, see [Filtering Schematics](#), on page 340.
3. Determine whether there is a timing exception, like a false or multicycle path. If this is the cause of the negative slack, set the appropriate timing constraint.

If there are fewer start points, pick a start point to add the constraint. If there are fewer end points, add the constraint to an end point.
4. If your design does not meet timing by 20 percent or more, you may need to make structural changes. You could do this by doing either of the following:
 - Enabling options like pipelining ([Retiming](#), on page 418), retiming ([Retiming](#), on page 418), or resource sharing ([Sharing Resources](#), on page 434).
 - Modifying the source code.
5. Rerun synthesis and check your results.

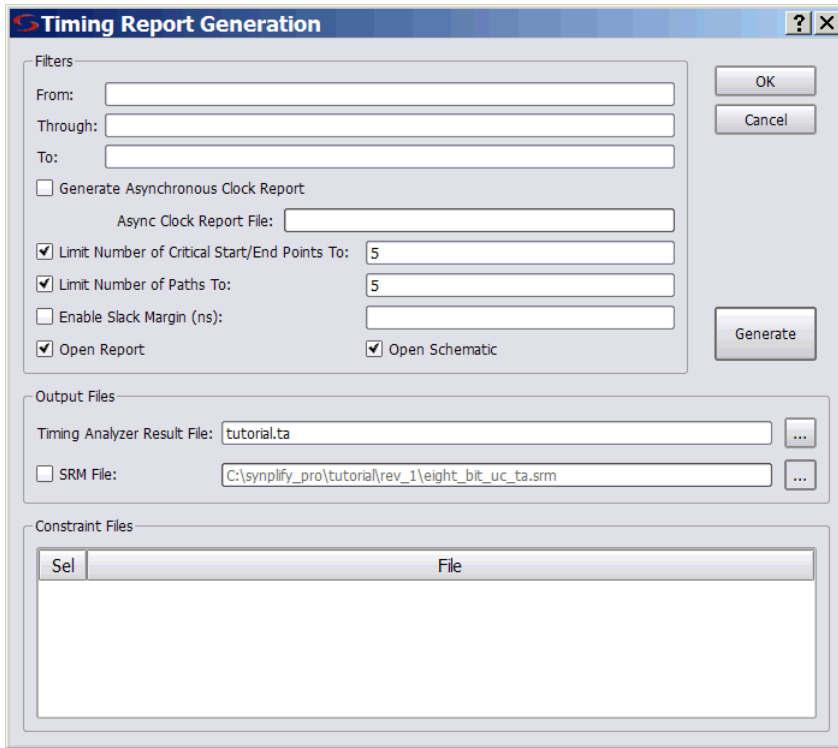
Generating Custom Timing Reports with STA

The log file generated after synthesis includes a timing report and default timing information. Use the stand-alone timing analyst (STA) when you need to generate a customized timing report (ta) for the following situations:

- You need more details about a specific path
- You want results for paths other than the top five timing paths (log file default)
- You want to modify constraints and analyze, without resynthesizing. See [Using Analysis Design Constraints, on page 370](#) for details.

The following procedure shows you how to generate a custom report:

1. Select Analysis->Timing Analyst or click on the Timing Analyst icon() .
2. Fill in the parameters.
 - You can type in the from/to or through points, or you can cut and paste or drag and drop valid objects from the Technology view (not the RTL view) into the fields. See [Timing Report Generation Parameters, on page 440](#) in the *Command Reference Manual* for details on timing analysis parameters and how they can be filtered.
 - Set options for clock reports as needed.
 - Specify a name for the output timing report (ta).



The image shows a 'Timing Report Generation' dialog box. It has a title bar with a question mark and a close button. The dialog is divided into several sections: 'Filters', 'Output Files', and 'Constraint Files'. In the 'Filters' section, there are input fields for 'From:', 'Through:', and 'To:'. Below these are checkboxes for 'Generate Asynchronous Clock Report', 'Limit Number of Critical Start/End Points To:' (set to 5), 'Limit Number of Paths To:' (set to 5), 'Enable Slack Margin (ns):', 'Open Report', and 'Open Schematic'. There is also an 'Async Clock Report File:' input field. In the 'Output Files' section, there is a 'Timing Analyzer Result File:' input field with the value 'tutorial.ta' and a file selection button. Below it is an 'SRM File:' input field with the value 'C:\synplify_pro\tutorial\rev_1\eight_bit_uc_ta.srm' and a file selection button. In the 'Constraint Files' section, there is a table with two columns: 'Sel' and 'File'. The table is currently empty. On the right side of the dialog, there are buttons for 'OK', 'Cancel', and 'Generate'.

3. Click **Generate** to run the report.

The software generates a custom report file called *projectName.ta*, located in the implementation directory (the directory you specified for synthesis results). The software also generates a corresponding output netlist file, with an `srm` extension.

4. Analyze results.

- View the report (Open Report) in the Text Editor. The following figure is a sample report showing analysis results based on maximum delay for the worst paths.

```
#### START OF TIMING REPORT ####[
# Timing Report written on Mon Jun 05 11:05:33 2006
#
```

```
Top view:          sw
Requested Frequency: 10.0 MHz
Wire load mode:    top
Paths requested:   8
from:              i:swmult2o[11]
Constraint File(s):
```

```
Worst From-To Path Information
*****
```

```
Path information for path number 1:
  Requested Period:          100.000
    - Setup time:            0.245
    = Required time:         99.755

  - Propagation time:        2.215
  = Slack :                  97.540

Number of logic level(s):    12
Starting point:              smult2o[11] / regout
Ending point:                o_o[12] / datain
The start point is clocked by clk_i [rising] on pin clk
The end point is clocked by  clk_i [rising] on pin clk
```

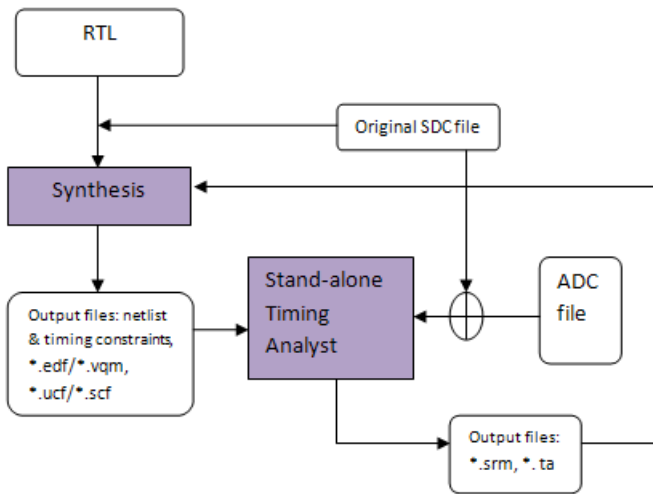
Instance / Net Name	Type	Pin Name	Pin Dir	Delay	Arrival Time	No. of Fan Out(s)
smult2o[11]	stratixii_lcell_ff	regout	Out	0.094	0.094	-
smult2o[11]	Net	-	-	0.311	-	2
smultl_carry_9	stratixii_lcell_comb	dataf	In	-	0.405	-

- View the netlist (View Critical Path) in a Technology view. This Technology view, labeled Timing View in the title bar, shows only the paths you specified in the Timing Analyst dialog box. Note that the Timing Analyst and Show Critical Path commands (and equivalent icons and shortcuts) are disabled whenever the Timing View is active.

Using Analysis Design Constraints

Besides generating custom timing reports (see [Generating Custom Timing Reports with STA, on page 367](#)), you can also use the Stand-alone Timing Analyst to create constraints in an `adc` file. You can use these constraints to experiment with different timing values, or to add or modify timing constraints.

The advantage to using analysis design constraints (ADC) is that you do not have to resynthesize the whole design. This reduces debugging time because you can get a quick estimate, or try out different values. The Standalone Timing Analyst (STA) puts these constraints in an Analysis Design Constraints file (`adc`). The process for using this file is summarized in the following flow diagram:



See the following for details:

- [Scenarios for Using Analysis Design Constraints](#), on page 371
- [Creating an ADC File](#), on page 372
- [Using Object Names Correctly in the adc File](#), on page 376

Scenarios for Using Analysis Design Constraints

The following describe situations where you can effectively use `adc` constraints to debug, explore options or modify constraints. For details about creating these constraints, see [Creating an ADC File, on page 372](#).

- **What-if analysis of design performance**
If your design meets the target frequency, you can use `adc` constraints to analyze higher target frequencies, or analyze performance of a module in a different design/technology/target device.
- **Constraints on enable registers**
Similarly, you can apply `syn_reference_clock` on enable registers to analyze if the enables have a regular pattern like clock, or if they operate on a frequency other than clock. For example:

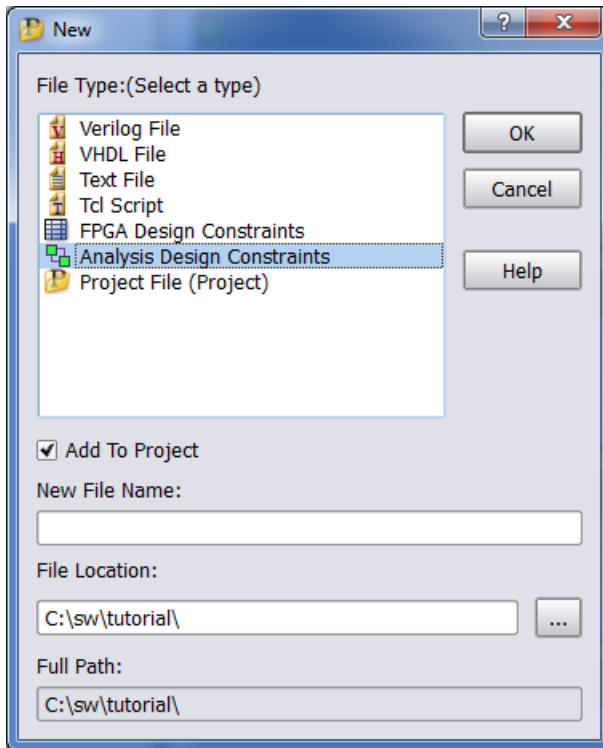
FDC	<code>create_clock {clk} -name {clk} -freq 100 -clockgroup clk_grp_0</code>
ADC	<code>define_attribute {n:en} syn_reference_clock {clk2} create_clock {clk2} -name {clk2} -freq 50 -clockgroup clk_grp_1</code>

- **Adding additional timing exceptions**
When you analyze the results of the first synthesis run, you often find functional or clock-to-clock timing exceptions, and you can handle these with `adc` constraints. For example:
 - Applying false paths on synchronization circuitry
 - Adding false paths between clocks belonging to different clock groupsYou must add these constraints to see more critical paths in the design. The `adc` constraints let you add these constraints on the fly, and helps you debug designs faster.
- **Modifying timing exceptions that were previously applied**
For example you might want to set a multicycle path constraint for a path that was defined as a false path in the constraint file or vice versa. To modify the timing exception, you must first ignore or reset the timing exception that was set in the constraint file, as described in [Using Analysis Design Constraints, on page 370](#), step 3.

Creating an ADC File

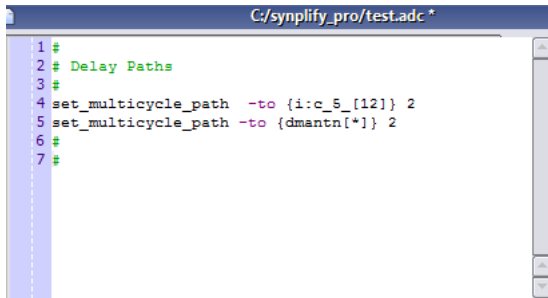
The following procedure explains how to create an `adc` file.

1. Select File->New.
2. Do the following in the dialog box that opens:
 - Select Analysis Constraint File.



- Type a name and location for the file. The tool automatically assigns the `adc` extension to the filename.
 - Enable Add to Project, and click OK. This opens the text editor where you can specify the new constraints.
3. Type in the constraints you want and save the file. Remember the following when you enter the constraints:
 - Keep in mind that the original `fdc` file has already been applied to the design. Any timing exception constraints in this file must not conflict

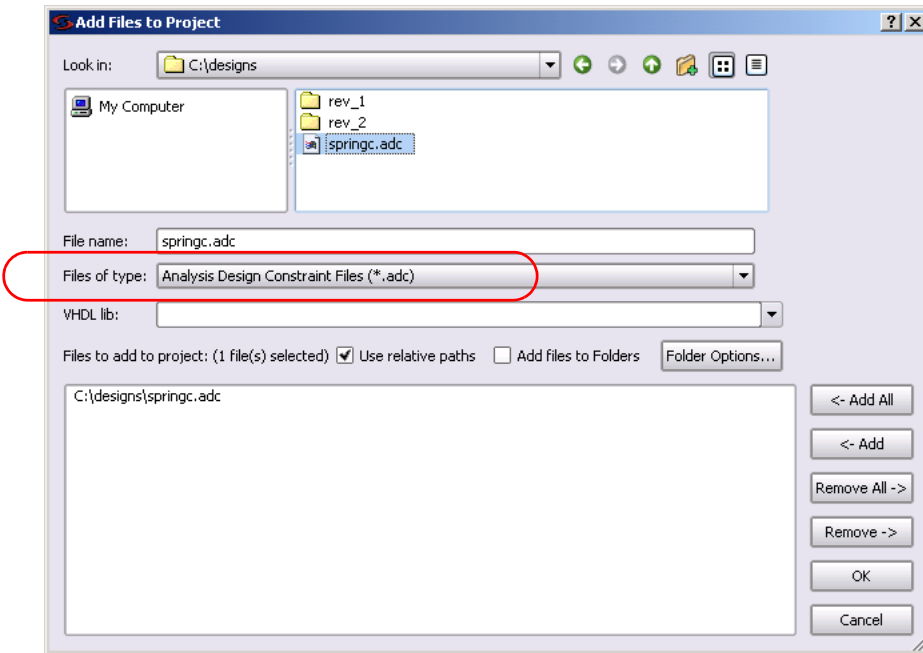
with constraints that are already in effect. For example, if there is a conflict when multiple timing exceptions (false path, path delay, and multicycle timing constraints) are applied to the same path, the tool uses this order to resolve conflicts: false path, multicycle path, max delay. See [Conflict Resolution for Timing Exceptions, on page 197](#) for details about how the tool prioritizes timing exceptions.



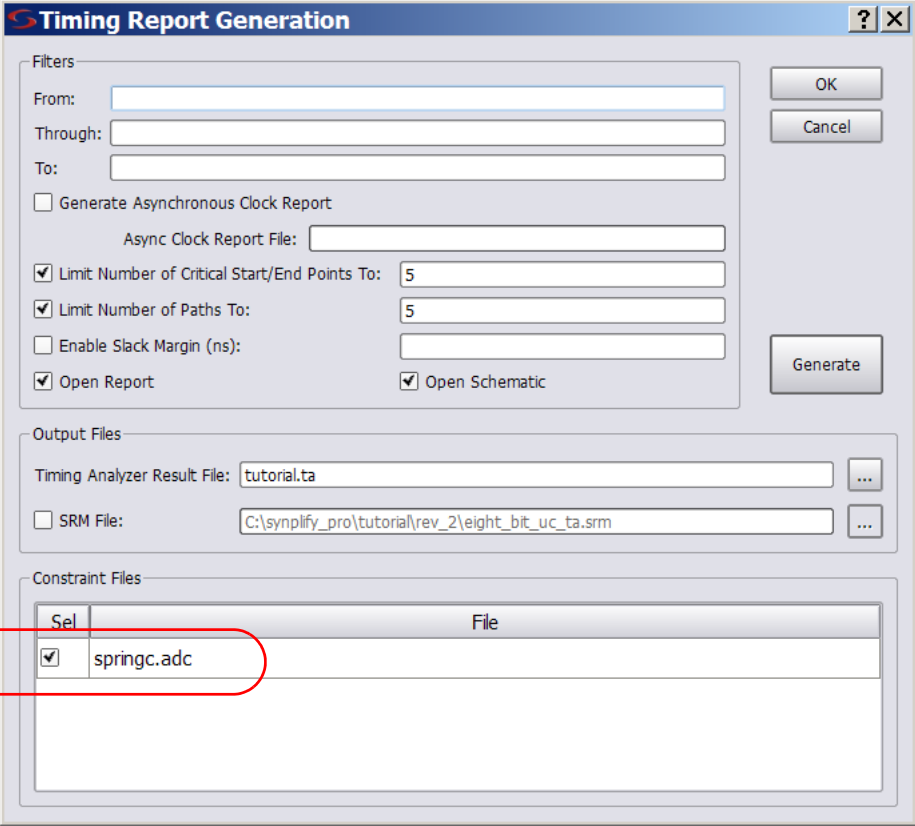
- The object names must be mapped object names, so use names from the Technology view, not names from the RTL view. Unlike the constraint file (RTL view), the `adc` constraints apply to the mapped database because the database is not remapped with this flow. For more information, see [Using Object Names Correctly in the `adc` File, on page 376](#).
- If you want to modify an existing constraint for a timing exception, you must first reset the original `fdc` constraint, and then apply the new constraint. In the following example the multicycle path constraint was changed to 3:

Original FDC	<code>set_multicycle_path -to [get_cells{a_reg*}] 2</code>
ADC	<code>reset_path -to {get_cells{a_reg*}}</code> <code>set_multicycle_path -to [get_cells{a_reg*}] 3</code>

- When you are done, save and close the file. This adds the file to your project.



- You can create multiple `adc` files for different purposes. For example, you might want to keep timing exception constraints, I/O constraints, and clock constraints in separate files. If you have an existing `adc` file, use the Add File command to add this file to your project. Select Analysis Design Constraint Files (*.adc) as the file type.
4. Run timing analysis.
- Select Analysis->Timing Analyst or click the Timing Analyst icon (). The Timing Analyst window will look like the example below, with pointers to the `srm` file, the original `fdc` and the new `adc` files you created.



The image shows a 'Timing Report Generation' dialog box. It has several sections: 'Filters', 'Output Files', and 'Constraint Files'. In the 'Filters' section, there are input fields for 'From:', 'Through:', and 'To:'. Below these are checkboxes for 'Generate Asynchronous Clock Report', 'Limit Number of Critical Start/End Points To:' (set to 5), 'Limit Number of Paths To:' (set to 5), 'Enable Slack Margin (ns):', 'Open Report', and 'Open Schematic'. In the 'Output Files' section, there is a 'Timing Analyzer Result File:' field with the value 'tutorial.ta' and an 'SRM File:' field with the value 'C:\synplify_pro\tutorial\rev_2\eight_bit_uc_ta.srm'. In the 'Constraint Files' section, there is a table with two columns: 'Sel' and 'File'. The first row has a checked checkbox in the 'Sel' column and the file name 'springc.adc' in the 'File' column. This row is highlighted with a red oval. There are 'OK', 'Cancel', and 'Generate' buttons on the right side of the dialog.

Sel	File
☒	springc.adc

- If you have multiple `adc` files, enable the ones you want.
- If you have a previous run and want to save that report, type a new name for the output `ta` file. If you do not specify a name, the tool overwrites the previous report.
- Fill in other parameters as appropriate, and click **Generate**.

The tool runs static timing analysis in the same implementation directory as the original implementation. The tool applies the `adc` constraints on top of the `fdc` constraints. Therefore, `adc` constraints affect timing results only if there are no conflicts with `fdc` constraints.

The tool generates a timing report called `*_adc.ta` and an `*_adc.srm` file by default. It does not change any synthesis outputs, like the output netlist or timing constraints for place and route.

5. Analyze the results in the timing report and *_adc.srm file.
6. If you need to resynthesize after analysis, add the adc constraints as an fdc file to the project and rerun synthesis.

Using Object Names Correctly in the adc File

Constraints and collections applied in the constraint file reference the RTL-level database. Synthesis optimizations such as retiming and replication can change object names during mapping because objects may be merged.

The standalone timing analyst does not map objects. It just reads the gate-level object names from the post-mapping database; this is reflected in the Technology view. Therefore, you must define objects either explicitly or with collections from the Technology view when you enter constraints into the adc file. Do not use RTL names when you create these constraints (see [Creating an ADC File](#), on page 372 for details of that process).

Example

Assume that register en_reg is replicated during mapping to reduce fanout. Further, registers en_reg and en_reg_rep2 connect to register dataout[31:0]. In this case, if you define the following false path constraint in the adc file, then the standalone timing analyzer does not automatically treat paths from the replicated register en_reg_rep2 as false paths.

```
set_false_path -from {{i:en_reg}} -to {{i:dataout[31:0]}}
```

Unlike constraints in the fdc file, you must specify this replicated register explicitly or as a collection. Only then are all paths properly treated as false paths. So in this example, you must define the following constraints in the adc file:

```
set_false_path -from {{i:en_reg}} -to {{i:dataout[31:0]}}
set_false_path -from {{i:en_reg_rep2}}
                -to {{i:dataout[31:0]}}
```

or

```
define_scope_collection en_regs {find -seq {i:en_reg*}
    -filter (@name == en_reg || @name == en_reg_rep2)}
set_false_path -from {{i:$en_regs}} -to {{i:dataout[31:0]}}
```

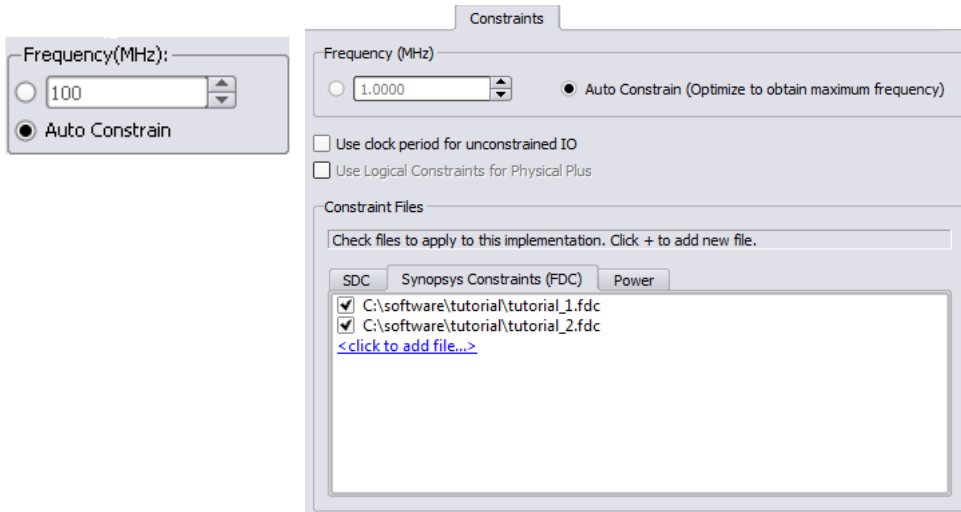
Using Auto Constraints

Auto constraining lets you synthesize with automatic constraints as a first step to get an idea of what you can achieve. Automatic constraints generate the fastest design implementation, so they force the timing engine to work harder. Based on the results from auto-constraining, you can refine the constraints manually later. For an explanation of how auto constraints work, see [Results of Auto Constraints, on page 379](#)

1. To automatically constrain your design, first do the following:
 - Set your device to a technology that supports auto-constraining. With supported technologies, the Auto Constrain button under Frequency in the Project view is available.



- Do not define any clocks. If you define clocks using the SCOPE window or a constraint file, or set the frequency in the Project view, the software uses the user-defined create_clock constraints instead of auto constraints.
 - Make sure any multi-cycle or false path constraints are specified on registers.
2. Enable the Auto Constrain button on the left side of the Project view. Alternatively, select Project->Implementation Options->Constraints, and enable the Auto Constrain option there.



3. If you want to auto constrain I/O paths, select Project->Implementation Options->Constraints and enable Use Clock Period for Unconstrained IO.

If you do not enable this option, the software only auto constrains flop-to-flop paths. Even when the software auto constrains the I/O paths, it does not generate these constraints for forward-annotation.

4. Synthesize the design.

The software puts each clock in a separate clock group and adjusts the timing of each clock individually. At different points during synthesis it adjusts the clock period of each clock to be a target percentage of the current clock period, usually 15% - 25%.

After the clocks, the timing engine constrains I/O paths by setting the default combinational path delay for each I/O path to be one clock period.

The software writes out the generated constraints in a file called AutoConstraint_*designName*.sdc in the run directory. It also forward-annotates these constraints to the place-and-route tools.

5. Check the results in AutoConstraint_*designName*.sdc and the log file. To open the constraint file as a text file, right-click on the file in the Implementation Results view and select Open as Text.

The flop-to-flop constraints use syntax like the following:

```
create_clock -name {c:leon|clk} -period 13.327 -clockgroup  
Autoconstr_clkgroup_0 -rise 0.000 -fall 6.664 -route 0.000
```

6. You can now add this generated constraint file to the project and rerun synthesis with these constraints.

Results of Auto Constraints

This section contains information about the following:

- [Stages of the Auto Constrain Algorithm](#), on page 379
- [I/O Constraints and Timing Exceptions](#), on page 379
- [Reports and Forward-annotation](#), on page 380
- [Repeatability of Results](#), on page 381

Stages of the Auto Constrain Algorithm

To auto constrain, do not define any clocks. When you enable the Auto Constrain option, the synthesis software goes through these stages:

1. It infers every clock in the design.
2. It puts each clock in its own clock group.
3. It invokes mapper optimizations in stages and generates the best possible synthesis results. You should only use Auto Constrain early in the synthesis process to get a general idea of how fast your design runs. This option is not meant to be a substitute for declaring clocks.
4. For each clock, including the system clock, the software maintains a negative slack of between 15 and 25 percent of the requested frequency.

I/O Constraints and Timing Exceptions

The auto constrain algorithm infers all the clocks, because none are defined. It handles the following timing situations as described below:

- I/O constraints

You can auto constrain I/O paths as well as flop-to-flop paths by selecting Project->Implementation Options->Constraints and enabling Use Clock

Period for Unconstrained IO. The software does not write out these I/O constraints.

- Timing exceptions like multicycle and false paths

The auto constraint algorithm honors SCOPE multicycle and false path constraints that are specified as constraints on registers.

Auto Constrain Limitations

The Auto Constrain feature has the following limitations:

- Does not respect the vendor-provided maximum frequency constraints for clock generators.
- Over constrains designs with output critical paths.

Reports and Forward-annotation

In the log file, the software reports the Requested and Estimated Frequency or Requested and Estimated Period and the negative slack for each clock it infers. The log file contains all the details.

The software also generates a constraint file in the run directory called `AutoConstraint_designName.sdc`, which contains the auto constraints generated. The following is an example of an auto constraint file:

```
#Begin clock constraint

create_clock -name {c:leon|clk} -period 13.327 -rise 0.000 -fall
6.664

#End clock constraint
```

The software forward-annotates the `create_clock` constraints, writing out the appropriate file for the place-and-route tool.

Repeatability of Results

If you use the requested frequency resulting from the Auto constrain option as the requested frequency for a regular synthesis run, you might not get the same results as you did with auto constraints. This is because the software invokes the mapper optimizations in stages when it auto constrains. The results from a previous stage are used to drive the next stage. As the interim optimization results vary, there is no guarantee that the final results will stay the same.

CHAPTER 9

Inferring High-Level Objects

This chapter contains guidelines on how to structure your code or attach attributes so that the synthesis tools can automatically infer high-level objects like RAMs. See the following for more information:

- [Defining Black Boxes for Synthesis](#), on page 384
- [Defining State Machines for Synthesis](#), on page 393
- [Automatic RAM Inference](#), on page 398
- [Initializing RAMs](#), on page 408

Defining Black Boxes for Synthesis

Black boxes are predefined components for which the interface is specified, but whose internal architectural statements are ignored. They are used as place holders for IP blocks, legacy designs, or a design under development.

This section discusses the following topics:

- [Instantiating Black Boxes and I/Os in Verilog](#), on page 384
- [Instantiating Black Boxes and I/Os in VHDL](#), on page 386
- [Adding Black Box Timing Constraints](#), on page 388
- [Adding Other Black Box Attributes](#), on page 392

For information about using black boxes with the Clock Conversion option, see [Working with Gated Clocks](#), on page 582.

Instantiating Black Boxes and I/Os in Verilog

Verilog black boxes for macros and I/Os come from two sources: commonly-used or vendor-specific components that are predefined in Verilog macro libraries, or black boxes that are defined in another input source like a schematic. For information about instantiating black boxes in VHDL, see [Instantiating Black Boxes and I/Os in VHDL](#), on page 386. Additional information about black boxes can be found in [Working with Gated Clocks](#), on page 582.

The following process shows you how to instantiate both types as black boxes. Refer to the *installDirectory/examples* directory for examples of instantiations of low-level resources.

1. To instantiate a predefined Verilog module as a black box:
 - Select the library file with the macro you need from the *installDirectory/lib/technology* directory. Files are named *technology.v*. Most vendor architectures provide macro libraries that predefine the black boxes for primitives and macros.
 - Make sure the library macro file is the first file in the source file list for your project.

2. To instantiate a module that has been defined in another input source as a black box:
 - Create an empty macro that only contains ports and port directions.
 - Put the `syn_black_box` synthesis directive just before the semicolon in the module declaration.

```
module myram (out, in, addr, we) /* synthesis syn_black_box */;
    output [15:0] out;
    input [15:0] in;
    input [4:0] addr;
    input we;
endmodule
```

- Make an instance of the stub in your design.
- Compile the stub along with the module containing the instantiation of the stub.
- To simulate with a Verilog simulator, you must have a functional description of the black box. To make sure the synthesis software ignores the functional description and treats it as a black box, use the `translate_off` and `translate_on` constructs. For example:

```
module adder8(cout, sum, a, b, cin);
    // Code that you want to synthesize
    /* synthesis translate_off */
    // Functional description.
    /* synthesis translate_on */
    // Other code that you want to synthesize.
endmodule
```

3. To instantiate a vendor-specific (black box) I/O that has been defined in another input source:
 - Create an empty macro that only contains ports and port directions.
 - Put the `syn_black_box` synthesis directive just before the semicolon in the module declaration.
 - Specify the external pad pin with the `black_box_pad_pin` directive, as in this example:

```
module BBDLHS (D,E,GIN,GOUT,PAD,Q)
    /* synthesis syn_black_box black_box_pad_pin="PAD" */
endmodule
```

- Make an instance of the stub in your design.

- Compile the stub along with the module containing the instantiation of the stub.
- 4. Add timing constraints and attributes as needed. See [Adding Black Box Timing Constraints, on page 388](#) and [Adding Other Black Box Attributes, on page 392](#).
- 5. After synthesis, merge the black box netlist and the synthesis results file using the method specified by your vendor.

Instantiating Black Boxes and I/Os in VHDL

VHDL black boxes for macros and I/Os come from two sources: commonly-used or vendor-specific components that are predefined in VHDL macro libraries, or black boxes that are defined in another input source like a schematic. For information about instantiating black boxes in VHDL, see [Instantiating Black Boxes and I/Os in Verilog, on page 384](#).

The following process shows you how to instantiate both types as black boxes. Refer to the *installDirectory/examples* directory for examples of instantiations of low-level resources.

1. To instantiate a predefined VHDL macro (for a component or an I/O),
 - Select the library file with the macro you need from the *installDirectory/lib/vendor* directory. Files are named *family.vhd*. Most vendor architectures provide macro libraries that predefine the black boxes for primitives and macros.
 - Add the appropriate library and use clauses to the beginning of your design units that instantiate the macros.

```
library family;  
use family.components.all;
```

2. To create a black box for a component from another input source:
 - Create a component declaration for the black box.
 - Declare the `syn_black_box` attribute as a boolean attribute.
 - Set the attribute to true.

```

library synplify;
use synplify.attributes.all;
entity top is
    port (clk, rst, en, data: in bit; q: out bit);
end top;

architecture structural of top is
    component bbox
        port(Q: out bit; D, C, CLR: in bit);
    end component;

    attribute syn_black_box of bbox: component is true;
    ...

```

- Instantiate the black box and connect the ports.

```

begin
my_bbox: bbox port map (
    Q => q,
    D => data,
    C => clk,
    CLR => rst);

```

- To simulate with a VHDL simulator, you must have the functional description of a black box. To make sure the synthesis software ignores the functional description and treats it as a black box, use the `translate_off` and `translate_on` constructs. For example:

```

architecture behave of ram4 is
begin
    -- synthesis translate_off
    stimulus: process (clk, a, b)
    -- Functional description
    end process;
    -- synthesis translate_on

    -- Other source code you WANT synthesized

```

3. To create a vendor-specific (black box) I/O for an I/O defined in another input source:
 - Create a component declaration for the I/O.
 - Declare the `black_box_pad_pin` attribute as a string attribute.
 - Set the attribute value on the component to be the external pin name for the pad.

```

library synplify;
use synplify.attributes.all;
...

component mybuf
  port(O: out bit; I: in bit);
end component;
attribute black_box_pad_pin of mybuf: component is "I";

```

- Instantiate the pad and connect the signals.

```

begin
data_pad: mybuf port map (
  O => data_core,
  I => data);

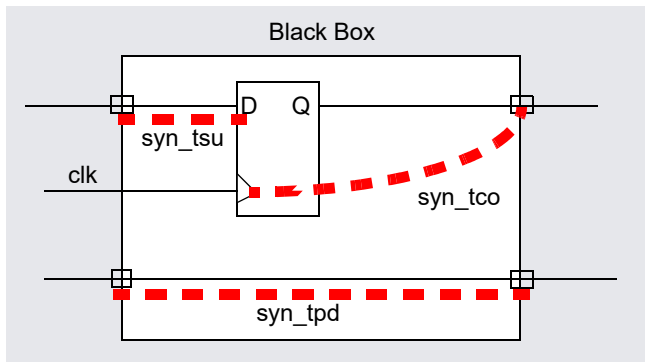
```

4. Add timing constraints and attributes. See [Adding Black Box Timing Constraints](#), on page 388, [Using Gated Clocks for Black Boxes](#), on page 605, and [Adding Other Black Box Attributes](#), on page 392.

Adding Black Box Timing Constraints

A black box does not provide the software with any information about internal timing characteristics. You must characterize black box timing accurately, because it can critically affect the overall timing of the design. To do this, you add constraints in the source code or in the SCOPE interface.

You attach black box timing constraints to instances that have been defined as black boxes. There are three black box timing constraints, `syn_tpd`, `syn_tsu`, and `syn_tco`. There are additional attributes for black box pins and black boxes with gated clocks; see [Adding Other Black Box Attributes](#), on page 392 and [Using Gated Clocks for Black Boxes](#), on page 605.



1. Define the instance as a black box, as described in [Instantiating Black Boxes and I/Os in Verilog, on page 384](#) or [Instantiating Black Boxes and I/Os in VHDL, on page 386](#).
2. Determine the kind of constraint for the information you want to specify:

To define...	Use...
Propagation delay through the black box	syn_tpd
Setup delay (relative to the clock) for input pins	syn_tsu
Clock-to-output delay through the black box	syn_tco

3. In VHDL, use the following syntax for the constraints.
 - Use the predefined attributes package by adding this syntax

```
library synplify;
use synplify.attributes.all;
```

In VHDL, you must use the predefined attributes package. For each directive, there are ten predeclared constraints in the attributes package, from *directive_name1* to *directive_name10*. If you need more constraints, declare the additional constraints using integers greater than 10. For example:

```
attribute syn_tcoll : string;
attribute syn_tcol2 : string;
```

- Define the constraints in either of these ways:

VHDL syntax	<code>attribute attributeName<n> : "att_value"</code>

Verilog-style notation	<code>attribute attributeName<n> of bbox_name : component is "att_value"</code>

The following table shows the appropriate syntax for `att_value`. See the *Attribute Reference Manual* for complete syntax information.

Attribute	Value Syntax
<code>syn_tsu<n></code>	<code>bundle -> [!]clock = value</code>
<code>syn_tco<n></code>	<code>[!]clock -> bundle = value</code>
<code>syn_tpd<n></code>	<code>bundle -> bundle = value</code>

- `<n>` is a numerical suffix.
- `bundle` is a comma-separated list of buses and scalar signals, with no intervening spaces. For example, A,B,C.
- `!` indicates (optionally) a negative edge for a clock.
- `value` is in ns.

The following is an example of black box attributes, using VHDL signal notation:

```
architecture top of top is
  component rcf16x4z port (
    ad0, ad1, ad2, ad3 : in std_logic;
    di0, di1, di2, di3 : in std_logic;
    wren, wpe : in std_logic;
    tri : in std_logic;
    do0, do1, do2, do3 : out std_logic;
  ) end component

  attribute syn_tpd1 of rcf16x4z : component is
    "ad0,ad1,ad2,ad3 -> do0,do1,do2,do3 = 2.1";
  attribute syn_tpd2 of rcf16x4z : component is
    "tri -> do0,do1,do2,do3 = 2.0";
  attribute syn_tsu1 of rcf16x4z : component is
    "ad0,ad1,ad2,ad3 -> ck = 1.2";
  attribute syn_tsu2 of rcf16x4z : component is
    "wren,wpe,do0,do1,do2,do3 -> ck = 0.0";
```

4. In Verilog, add the directives as comments, as shown in the following example. For explanations about the syntax, see the table in the previous step or the *Attribute Reference Manual*.

```
module ram32x4 (z, d, addr, we, clk)
  /* synthesis syn_black_box
    syn_tpd1="addr[3:0]->z[3:0]=8.0"
    syn_tsu1="addr[3:0]->clk=2.0"
    syn_tsu2="we->clk=3.0" */;
  output [3:0] z;
```



```
input [3:0] d;  
input [3:0] addr;  
input we;  
input clk;  
endmodule
```

5. To add black box attributes through the SCOPE interface, do the following:
 - Open the SCOPE spreadsheet and select the Attributes panel.
 - In the Object column, select the name of the black-box module or component declaration from the pull-down list. Manually prefix the black box name with **v:** to apply the constraint to the view.
 - In the Attribute column, type the name of the timing attribute, followed by the numerical suffix, as shown in the following table. You cannot select timing attributes from the pull-down list.
 - In the Value column, type the appropriate value syntax, as shown in the table in step 3.
 - Save the constraint file, and add it to the project.

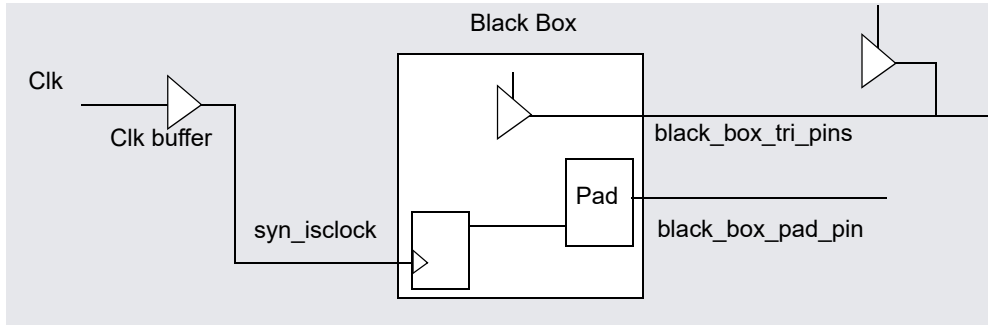
The resulting constraint file contains syntax like this:

```
define_attribute v:{blackboxModule} attribute<n> {attributeValue}
```

6. Synthesize the design and check black box timing.

Adding Other Black Box Attributes

Besides black box timing constraints, you can also add other attributes to define pin types on the black box or define gated clocks. You cannot use the attributes for all technologies. Check the *Attribute Reference Manual* for details about which technologies are supported. For information about black boxes with gated clocks, see [Using Gated Clocks for Black Boxes](#), on page 605.



1. To specify that a clock pin on the black box has access to global clock routing resources, use `syn_isclock`.
2. To specify that the software need not insert a pad for a black box pin, use `black_box_pad_pin`. Use this for technologies that automatically insert pad buffers for the I/Os.
3. To define a tristate pin so that you do not get a mixed driver error when there is another tristate buffer driving the same net, use `black_box_tri_pins`.

Defining State Machines for Synthesis

A finite state machine (FSM) is a piece of hardware that advances from state to state at a clock edge. The synthesis software recognizes and extracts the state machines from the HDL source code. For guidelines on setting up the source code, see the following:

- [Defining State Machines in Verilog](#), on page 393
- [Defining State Machines in VHDL](#), on page 394
- [Specifying FSMs with Attributes and Directives](#), on page 395

For information about the attributes used to define state machines, see [Running the FSM Compiler](#), on page 438.

Defining State Machines in Verilog

The synthesis software recognizes and automatically extracts state machines from the Verilog source code if you follow the coding guidelines listed below. The software attaches the `syn_state_machine` attribute to each extracted FSM.

For alternative ways to define state machines, see [Defining State Machines for Synthesis](#), on page 393.

Follow these Verilog coding guidelines:

- In Verilog, model the state machine with `case`, `casex`, or `casez` statements in `always` blocks. Check the current state to advance to the next state and then set output values. Do not use `if` statements.
- Always use a default assignment as the last assignment in the `case` statement, and set the state variable to 'bx. This is a “don't care” statement and ensures that the software can remove unnecessary decoding and gates.
- Make sure the state machines have a synchronous or asynchronous reset to set the hardware to a valid state after power-up, or to reset the hardware when you are operating.

- Specify explicit state values for states with parameter or ``define` statements. This is an example of a parameter statement that sets the current state to `2'h2`:

```
parameter state1 = 2'h1, state2 = 2'h2;  
...  
current_state = state2;
```

This example shows how to set the current state value with ``define` statements:

```
`define state1 2'h1  
`define state2 2'h2  
...  
current_state = `state2;
```

- Make state assignments using parameter with symbolic state names. Use parameter over ``define`, because ``define` is applied globally while parameter definitions are local. Local definitions make it easier to reuse common state names in multiple FSM designs, like RESET, IDLE, READY, READ, WRITE, ERROR, and DONE.

If you use ``define` to assign the names, you cannot reuse a state name because it has already been used in the global name space. To reuse the same names in this scenario, you have to use ``undef` and ``define` statements between modules to redefine the names. This method makes it difficult to probe the internal values of FSM state buses from a testbench and compare them to the state names.

Defining State Machines in VHDL

The synthesis software recognizes and automatically extracts state machines from the VHDL source code if you follow the coding guidelines below. For alternative ways to define state machines, see [Defining State Machines for Synthesis, on page 393](#).

The following are VHDL guidelines for coding. The software attaches the `syn_state_machine` attribute to each extracted FSM.

- Use case statements to check the current state at the clock edge, advance to the next state, and set output values. You can also use if-then-else statements, but case statements are preferable.

- If you do not cover all possible cases explicitly, include a `when others` assignment as the last assignment of the `case` statement, and set the state vector to some valid state.
- If you create implicit state machines with multiple `WAIT` statements, the software does not recognize them as state machines.
- Make sure the state machines have a synchronous or asynchronous reset to set the hardware to a valid state after power-up, or to reset the hardware when you are operating.
- To choose an encoding style, attach the `syn_encoding` attribute to the enumerated type. The software automatically encodes your state machine with the style you specified.

Specifying FSMs with Attributes and Directives

If your design has state machines, the software can extract them automatically with the FSM Compiler, or you can manually attach attributes to state registers to define them as state machines. See [Optimizing State Machines, on page 437](#) for information about automatic FSM extraction, and [Defining State Machines for Synthesis, on page 393](#) for other ways to specify FSMs.

The following steps show you how to manually attach attributes to define FSMs for extraction.

1. To determine how state machines are extracted, set attributes in the source code as shown in the following table:

To ...	Attribute
Specify a state machine for extraction and optimization	<code>syn_state_machine=1</code>
Prevent state machines from being extracted and optimized	<code>syn_state_machine=0</code>
Prevent the state machine from being optimized away	<code>syn_preserve=1</code>

For information about how to add attributes, see [Specifying Attributes and Directives, on page 86](#).

2. To determine the encoding style for the state machine, set the `syn_encoding` attribute in the source code or in the SCOPE window. For VHDL users there are alternative methods, described in the next step.

The FSM Compiler honors the `syn_encoding` setting. The different values for this attribute are briefly described here; refer to the *Attributes Reference* manual for complete details.

Situation: If ...	<code>syn_encoding</code> Value	Explanation
Area is important	<code>sequential</code>	One of the smallest encoding styles.
Speed is important	<code>onehot</code>	Usually the fastest style and suited to most FPGA styles.
Recovery from an invalid state is important	<code>safe</code> , with another style. For example: <pre>/* synthesis syn_encoding = "safe, onehot" */</pre>	Forces the state machine to reset in certain situations. For example, if an alpha particle hit in a hostile operating environment causes a spontaneous register change, you can use <code>safe</code> to reset the state machine.
There are <5 states	<code>sequential</code>	Default encoding.
A large output decoder follows the FSM	<code>sequential gray</code>	Could be faster than <code>onehot</code> , even though the value must be decoded to determine the state. For <code>sequential</code> , more than one bit can change at a time; for <code>gray</code> , only one bit changes at a time, but more than one bit can be hot.
There are a large number of flip-flops	<code>onehot</code>	Fastest style, because each state variable has one bit set, and only one bit of the state register changes at a time.

3. If you are using VHDL, you have two choices for defining encoding:
 - Use `syn_encoding` as described above and enable the FSM compiler.
 - Use `syn_enum_encoding` to define the states (`sequential`, `onehot`, `gray`, and `safe`) and disable the FSM compiler. If you do not disable the FSM

compiler, the `syn_enum_encoding` values are not implemented. This is because the FSM compiler, a mapper operation, overrides `syn_enum_encoding`, which is a compiler directive. Use the `syn_enum_encoding` method for user-defined FSM encoding. For example:

```
attribute syn_enum_encoding of state_type : type is "001 010 101";
```

Automatic RAM Inference

Instead of instantiating synchronous RAMs, you can let the synthesis tools automatically infer them directly from the HDL source code and map them to the appropriate technology-specific RAM resources on the FPGA. This approach lets you maintain portability.

Here are some of the advantages offered by the inference approach:

- The tool automatically infers the RAMs from the HDL code, which is technology-independent. This means that the design is portable from one technology to another without rework.
- RAM inference is the best method for prototyping.
- The tool automatically adds the extra glue logic needed to ensure that the logic is correct.
- The software automatically runs timing-driven synthesis for inferred RAMs.

For further details about RAM inference, see [Inferring Block RAM, on page 401](#).

Block RAM

The synthesis software can implement the block RAM it infers using different types of block RAM and different block RAM modes.

Types of Block RAM

The synthesis software can infer different kinds of block RAM, according to how the code is set up. For details about block RAM inference, see [Inferring Block RAM, on page 401](#) and [RAM Attributes, on page 399](#). For inference examples, and see [Block RAM Examples, on page 191](#).

The synthesis tool can infer the following kinds of block RAM:

- Single-port RAM
- Dual-port RAM

Based on how the read and write ports are used, dual-port RAM can be further classified as follows:

- Simple dual-port
- Dual-port
- True dual-port

Supported Block RAM Modes

Block RAM supports three operating modes, which determine the output of the RAM when write enable is active. The synthesis tools infer the mode from the RTL you provide. It is best to explicitly describe the RAM behavior in the code, so as to correctly infer the operating mode you want. Refer to the examples for recommended coding styles.

The block RAM operating modes are described in the following table:

Mode	When write enable (WE) is active ...
WRITE_FIRST	This is a transparent mode, and the input data is simultaneously written into memory and stored in the RAM data output (DO). DO uses the value of the RAM data input (DI). See WRITE_FIRST Mode Example , on page 192 for an example.
READ_FIRST	This mode is read before write. The data previously stored at the write address appears at the RAM data output (DO) first, and then the RAM input data is stored in memory. DO uses the value of the memory content. See READ_FIRST Mode Example , on page 193 for an example.
NO_CHANGE	RAM data output (DO) remains the same during a write operation, with DO containing the last read data. See NO_CHANGE Mode Example , on page 194 for an example.

RAM Attributes

In addition to the automatic inference by the tool, you can specify RAM inference with the `syn_ramstyle` and `syn_rw_conflict_logic` attributes. The `syn_ramstyle` attribute explicitly specifies the kind of RAM you want, while the `syn_rw_conflict_logic` attribute specifies that you want to infer a RAM, but leave it to the synthesis tools to select the kind of RAM, as appropriate.

Attribute-Based Inference of Block RAM

For block RAM, the `syn_ramstyle` attribute has a number of valid values, all of which are extensively described in the documentation. This section confines itself to the following values, which are most relevant to the discussion:

<code>syn_ramstyle</code> Value	Description
<code>block_ram</code>	Enforces the inference and implementation of a technology-specific RAM.
<code>registers</code>	Prevents inference of a RAM, and maps the RAM to flip-flops and logic.
<code>no_rw_check</code>	Does not create overhead logic to account for read-write conflicts.

If you specify the `syn_rw_conflict_logic` attribute, the synthesis tools can infer block RAM, depending on the design. If the tool does infer block RAM, it does not insert bypass logic around the block RAM to account for read-write conflicts and prevent simulation mismatches. In this way its functionality is the same as `syn_ramstyle` with `no_rw_check`, which does not insert bypass logic either.

Specifying the Attributes

You set the attribute in the HDL source code, through the SCOPE interface or in an FPGA constraint file.

HDL Source Code

Set the attribute on the Verilog register or VHDL signal that holds the output values of the RAM. The following syntax shows how to specify the attribute in Verilog and VHDL code:

Verilog	<pre>reg [7:0] ram_dout [127:0] /*synthesis syn_ramstyle = "block_ram"*/; reg [d_width-1:0] mem [mem_depth-1:0] /*synthesis syn_rw_conflict_logic = 0*/;</pre>
VHDL	<pre>attribute syn_ramstyle of ram_dout : signal is "block_ram";</pre>

SCOPE

For the `syn_ramstyle` attribute, set the attribute on the RAM register memory signal, `mem`, as shown below. For the `syn_rw_conflict_logic` attribute, set it on the instance or set it globally. The attributes are written out to a constraints file using the syntax described in the next section.

	Enabled	Object Type	Object	Attribute	Value	Val Type
1	☒	<any>	i:mem[7:0]	syn_ramstyle	block_ram	string

Constraints File

In the `fdc Tcl` constraints file written out from the SCOPE interface, the `syn_ramstyle` attribute is attached to the register `mem` signal of the RAM, and the `syn_rw_conflict_logic` attribute is attached to the view, as shown below:

```
define_attribute {i:mem[7:0]} {syn_ramstyle} {block_ram}
define_attribute {v:mem[0:7]} syn_rw_conflict_logic {0}
```

For the `syn_rw_conflict_logic` attribute, you can also specify it globally, as well as on individual modules and instances:

```
define_global_attribute syn_rw_conflict_logic {0}
```

Inferring Block RAM

Based on the design and how you code it, the tool can infer the following kinds of block RAM: single-port, simple dual-port, dual-port, and true dual-port. The details about RAM inference and setup guidelines are described here:

- [Setting up the RTL and Inferring Block RAM](#), on page 402
- [Simple Dual-Port Block RAM Inference](#), on page 403
- [Dual-Port RAM Inference](#), on page 405
- [True Dual-Port RAM Inference](#), on page 405
- [True Dual-Port Byte-Enabled RAM Inference](#), on page 407

Setting up the RTL and Inferring Block RAM

To ensure that the tool infers the kind of block RAM you want, do the following:

1. Set up the RAM HDL code in accordance with the following guidelines:
 - The RAM must be synchronous. It must not have any asynchronous control signals connected. The synthesis tools do not infer asynchronous block RAM.
 - You must register either the read address or the output.
 - The RAMs must not be too small, as the tool does not infer block RAM for small-sized RAMs. The size threshold varies with the target technology.
2. Set up the clocks and read and write ports to infer the kind of RAM you want. The following table summarizes how to set up the RAM in the RTL:

RAM	Clock	Read Ports	Write Ports
Single-port	Single clock	One; same as write	One; same as read
Simple dual-port	Single or dual clock	One dedicated read	One dedicated write
Dual-port	Single or dual clock	Two independent reads	One dedicated write
True dual-port	Single or dual clock	Two independent reads	Two independent writes

See [Dual-Port RAM Inference](#) , on page 405 and [True Dual-Port RAM Inference](#) , on page 405 for additional information.

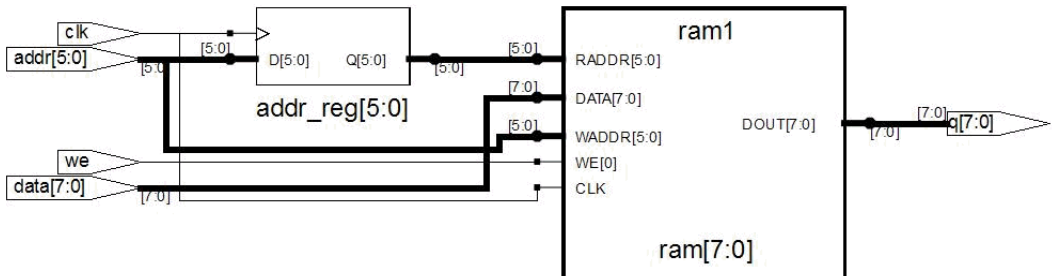
For illustrative code examples, see the single-port and dual-port examples listed in [Block RAM Examples](#), on page 191.

3. If needed, guide automatic inference with the `syn_ramstyle` attribute:
 - To force the inference of block RAM, specify `syn_ramstyle=blockram`.
 - To prevent a block RAM from being inferred or if your resources are limited, use `syn_ramstyle=registers`.
 - If you know your design does not read and write to the same address simultaneously, specify `syn_ramstyle=no_rw_check` to ensure that the

synthesis tool does not unnecessarily create bypass logic for resolving conflicts.

4. Synthesize the design.

The tool first compiles the design and infers the RAMs, which it represents as abstract technology-independent primitives like RAM1 and RAM2. You can view these RAMs in the RTL view, which is a graphic, technology-independent representation of your design after compilation:



It is important that the compiler first infers the RAM, because the tool only maps the inferred RAM primitives to technology-specific block RAM. Any RAM that is not inferred is mapped to registers. You can view the mapped RAMs in the Technology view, which is a graphic representation of your design after synthesis, and shows the design mapped to technology-specific resources.

Simple Dual-Port Block RAM Inference

Simple dual-port RAMs (SDP) are block RAMs with one port dedicated to read operations and one port dedicated to write operations. SDP RAMs offer the unique advantage of combining ports and using them to pack double the data width and address width.

The synthesis tools map SDP RAMs to RAM primitives in the architecture. A unique set of addresses, clocks, and enable signals are used for each port. The synthesis tool might also set the `RAM_MODE` property on the RAM to indicate the RAM mode.

The inference of simple dual-port RAM is dependent on the size of the address and data. The RAM must follow the coding guidelines listed below to be inferred.

- The read and write addresses must be different

- The read and write clocks can be different
- The enable signals can be different

Here is an example where the tool infers SDP RAM:

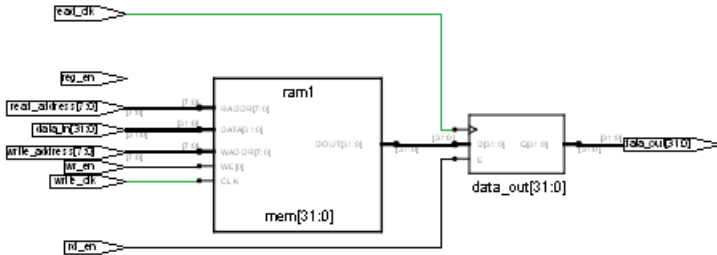
```
module Read_First_RAM (
    read_clk,
    read_address,
    data_in,
    write_clk,
    rd_en,
    wr_en,
    reg_en,
    write_address,
    data_out);

    parameter address_width = 8;
    parameter data_width = 32;
    parameter depth = 256;
    input read_clk, write_clk;
    input rd_en;
    input wr_en;
    input reg_en;
    input [address_width-1:0] read_address, write_address;
    input [data_width-1:0] data_in;
    output [data_width-1:0] data_out;
    //wire [data_width-1:0] data_out;
    reg [data_width-1:0] mem [depth-1 : 0]/* synthesis
    syn_ramstyle="no_rw_check"
    */;
    reg [data_width-1:0] data_out;

    always @(posedge write_clk)
    if(wr_en)
        mem[write_address] <= data_in;

    always @(posedge read_clk)
    if(rd_en)
        data_out <= mem[read_address];

endmodule
```



Dual-Port RAM Inference

Dual-port RAM is configured to have read and/or write operations from both ports of the RAM. One such configuration is a RAM with one port for both read and write operations and another dedicated read-only port. A unique set of addresses, clocks, and enable signals are used for each port. The synthesis tool sets properties on the RAM to indicate the RAM mode.

To infer dual-port block RAM, the RAM must follow the coding rules described below.

- The read and write addresses must be different
- The read and write clocks can be different
- The enable signals can be different

True Dual-Port RAM Inference

True dual-port RAMs (TDP) are block RAMs with two write ports and two read ports. The compiler extracts a RAM2 primitive for RAMs with two write ports or two read ports and the tool maps this primitive to TDP RAM. The ports operate independently, with different clocks, addresses and enables.

The synthesis tool also sets the RAM_MODE property on the RAM to indicate the RAM mode.

The compiler infers TDP block RAM based on the write processes. The implementation depends on whether the write enables use one process or multiple processes:

- When all the writes are made in one process, there are no address conflicts, and the compiler generates an `nram` that is later mapped to either true dual-port block RAM. The following coding results in an `nram` with two write ports, one with write address `waddr0` and the other with write address `waddr1`:

```
always @(posedge clk)
begin
    if(we1) mem[waddr0] <= data1;
    if(we2) mem[waddr1] <= data2;
end
```

- When the writes are made in multiple processes, the software does not infer a multiport RAM unless you explicitly specify the `syn_ramstyle` attribute with a value that indicates the kind of RAM to implement, or with the `no_rw_check` value. If the attribute is not specified as such, the software does not infer an `nram`, but infers a RAM with multiple write ports. You get a warning about simulation mismatches when the two addresses are the same.

In the following case, the compiler infers an `nram` with two write ports because the `syn_ramstyle` attribute is specified. The writes associated with `waddr0` and `waddr1` are `we1` and `we2`, respectively.

```
reg [1:0] mem [7:0] /* synthesis syn_ramstyle="no_rw_check" */;
always @(posedge clk1)
begin
    if(we1) mem[waddr0] <= data1;
end

always @(posedge clk2)
begin
    if(we2) mem[waddr1] <= data2;
end
```


True Dual-Port Byte-Enabled RAM Inference

The procedure below describes how to specify RAM where you can read/write each byte into a specific address location independently, and how to implement it as block RAM. See the Synopsys article 2560210, *Verilog RTL Coding Style for True Dual-Port Byte-Enabled RAM*, for an example.

1. Instantiate the true dual-port RAM n number of times, where n is the number of bytes for a particular RAM address.

In the following example, `ram_dp` is instantiated twice because there are two bytes in the address:

```
ram_dp u1 (clk1, clk2, dia[7:0] , addra, wea[0], doa[7:0] , dib[7:0] , addrb, web[0],  
          dob[7:0]);  
ram_dp u2 (clk1, clk2, dia[15:8], addra, wea[1], doa[15:8], dib[15:8], addrb,  
          web[1], dob[15:8]);
```

2. To map the true dual-port RAM into a block RAM, add the `syn_ramstyle="block_ram"` attribute to the true dual-port RAM module.
3. Run compile.

The RTL schematic shows two instantiations, as specified.

4. Run map.

After synthesis, check the resource utilization report to make sure that two block RAMs were inferred, as specified.

Initializing RAMs

You can specify startup values for RAMs and pass them on to the place-and-route tools. See the following topics for ways to set the initial values:

- [Initializing RAMs in Verilog](#), on page 408
- [Initializing RAMs in VHDL](#), on page 409
- [Initializing RAMs with \\$readmemb and \\$readmemh](#), on page 412

Initializing RAMs in Verilog

In Verilog, you specify startup values using initial statements, which are procedural `assign` statements guaranteed by the language to be executed by the simulator at the start of simulation. This means that any assignment to a variable within the body of the initial statement is treated as if the variable was initialized with the corresponding LHS value. You can initialize memories using the built-in load memory system tasks `$readmemb` (binary) and `$readmemh` (hex).

The following procedure is the recommended method for specifying initial values:

1. Create a data file with an initial value for every address in the memory array. This file can be a binary file or a hex file. See [Initialization Data File, on page 208](#) in the *Reference Manual* for details of the formats for these files.
2. Do the following in the Verilog file to define the module:
 - Include the appropriate task enable statement, `$readmemb` or `$readmemh`, in the initial statement for the module:

```
$readmemb ("fileName", memoryName [, startAddress [, stopAddress]]);
```

```
$readmemh ("fileName", memoryName [, startAddress [, stopAddress]]);
```

Use `$readmemb` for a binary file and use `$readmemh` for a hex file. For descriptions of the syntax, see [Initial Values for RAMs, on page 204](#) in the *Reference Manual*.

- Make sure the array declaration matches the order in the initial value data file you specified. As the file is read, each number encountered is assigned to a successive word element of the memory. The software

starts with the left-hand address in the memory declaration, and loads consecutive words until the memory is full or the data file has been completely read. The loading order is the order in the declaration. For example, with the following memory definition, the first line in the data file corresponds to address 0:

```
reg [7:0] mem_up [0:63]
```

With this next definition, the first line in the data file applies to address 63:

```
reg [7:0] mem_down [63:0]
```

3. To forward-annotate initial values, use the `$readmemb` or `$readmemh` statements, as described in [Initializing RAMs with `\$readmemb` and `\$readmemh`, on page 412](#).

See [Example 1: RAM Initialization, on page 205](#) in the *Reference Manual* for an example of a Verilog single-port RAM.

Initializing RAMs in VHDL

There are two ways to initialize the RAM in the VHDL code: with signal declarations or with variable declarations.

Initializing VHDL Rams with Signal Declarations

The following example shows a single-port RAM that is initialized with signal initialization statements. For alternative methods, see [Initializing VHDL Rams with Variable Declarations, on page 411](#).

```
library ieee;
use ieee.std_logic_1164.all;
use ieee.std_logic_unsigned.all;

entity w_r2048x28 is
port (
    clk : in  std_logic;
    adr : in  std_logic_vector(10 downto 0);
    di  : in  std_logic_vector(26 downto 0);
    we  : in  std_logic;
    dout : out std_logic_vector(26 downto 0));
end;
```

© 2024 Synopsys, Inc.
410

Initializing VHDL Rams with Variable Declarations

The following example shows a RAM that is initialized with variable declarations. For alternative methods, see [Initializing VHDL Rams with Signal Declarations, on page 409](#) and [Initializing RAMs with \\$readmemb and \\$readmemh, on page 412](#).

```

library ieee;
use ieee.std_logic_1164.all;
use ieee.numeric_std.all;

entity one is
generic (data_width : integer := 6;
        address_width : integer := 3
);

port (data_a :in std_logic_vector(data_width-1 downto 0);
      raddr1  :in unsigned(address_width-2 downto 0);
      waddr1  :in unsigned(address_width-1 downto 0);
      we1     :in std_logic;
      clk     :in std_logic;
      out1    :out std_logic_vector(data_width-1 downto 0));
end;

architecture rtl of one is
    type mem_array is array(0 to 2**(address_width) -1) of
        std_logic_vector(data_width-1 downto 0);
begin

    WRITE1_RAM : process (clk)
        variable mem : mem_array := (1 => "111101", others => (1=>'1',
            others => '0'));
    begin
        if rising_edge(clk) then
            out1 <= mem(to_integer(raddr1));
            if (we1 = '1') then
                mem(to_integer(waddr1)) := data_a;
            end if;
        end if;
    end process WRITE1_RAM;
end rtl;

```

Initializing RAMs with \$readmemb and \$readmemh

1. Create a data file with an initial value for every address in the memory array. This file can be a binary file or a hex file. See [Initialization Data File, on page 208](#) in the *Reference Manual* for details.
2. Include one of the task enable statements, \$readmemb or \$readmemh, in the initial statement for the module:

```
$readmemb ("fileName", memoryName [, startAddress [, stopAddress]]);  
$readmemh ("fileName", memoryName [, startAddress [, stopAddress]]);
```

Use \$readmemb for a binary file and \$readmemh for a hex file. For details about the syntax, see [Initial Values for RAMs, on page 204](#) in the *Reference Manual*.

CHAPTER 10

Specifying Design-Level Optimizations

This chapter covers techniques for optimizing your design using built-in tools or attributes. For vendor-specific optimizations, see [Chapter 16, *Optimizing for Lattice Designs*](#). It describes the following:

- [Tips for Optimization](#), on page 414
- [Retiming](#), on page 418
- [Preserving Objects from Being Optimized Away](#), on page 425
- [Optimizing Fanout](#), on page 430
- [Sharing Resources](#), on page 434
- [Inserting I/Os](#), on page 435
- [Optimizing State Machines](#), on page 437
- [Inserting Probes](#), on page 442

Tips for Optimization

The software automatically makes efficient trade-offs to achieve the best results. However, you can optimize your results by using the appropriate control parameters. This section describes general design guidelines for optimization. The topics have been categorized as follows:

- [General Optimization Tips](#), on page 414
- [Optimizing for Area](#), on page 415
- [Optimizing for Timing](#), on page 416

General Optimization Tips

This section contains general optimization tips that are not directly area or timing-related. For area optimization tips, see [Optimizing for Area, on page 415](#). For timing optimization, see [Optimizing for Timing, on page 416](#).

- In your source code, remove any unnecessary priority structures in timing-critical designs. For example, use CASE statements instead of nested IF-THEN-ELSE statements for priority-independent logic.
- If your design includes safe state machines, use the `syn_encoding` attribute with a value of `safe`. This ensures that the synthesized state machines never lock in an illegal state.
- For FSMs coded in VHDL using enumerated types, use the same encoding style (`syn_enum_encoding` attribute value) on both the state machine enumerated type and the state signal. This ensures that there are no discrepancies in the type of encoding to negatively affect the final circuit.
- Make sure that the source code supports inferencing or instantiation by using architecture-specific resources like memory blocks.
- Some designs benefit from hierarchical optimization techniques. To enable hierarchical optimization on your design, set the `syn_hier` attribute to `firm`.
- For accurate results with timing-driven synthesis, explicitly define clock frequencies with a constraint, instead of using a global clock frequency.

Optimizing for Area

This section contains information on optimizing to reduce area. Optimizing for area often means larger delays, and you will have to weigh your performance needs against your area needs to determine what works best for your design. For tips on optimizing for performance, see [Optimizing for Timing, on page 416](#). General optimization tips are in [General Optimization Tips, on page 414](#).

- Increase the fanout limit when you set the implementation options. A higher limit means less replicated logic and fewer buffers inserted during synthesis, and a consequently smaller area. In addition, as P&R tools typically buffer high fanout nets, there is no need for excessive buffering during synthesis. See [Setting Fanout Limits, on page 430](#) for more information.
- Enable the Resource Sharing option when you set implementation options. With this option checked, the software shares hardware resources like adders, multipliers, and counters wherever possible, and minimizes area. This is a global setting, but you can also specify resource sharing on an individual basis for lower-level modules. See [Sharing Resources, on page 434](#) for details.
- For designs with large FSMs, use the gray or sequential encoding styles, because they typically use the least area. For details, see [Specifying FSMs with Attributes and Directives, on page 395](#).
- If you are mapping into a CPLD and do not meet area requirements, set the default encoding style for FSMs to sequential instead of oneshot. For details, see [Specifying FSMs with Attributes and Directives, on page 395](#).
- For small CPLD designs (less than 20K gates), you might improve area by using the `syn_hier` attribute with a value of `flatten`. When specified, the software optimizes across hierarchical boundaries and creates smaller designs.

Optimizing for Timing

This section contains information on optimizing to meet timing requirements. Optimizing for timing is often at the expense of area, and you will have to balance the two to determine what works best for your design. For tips on optimizing for area, see [Optimizing for Area, on page 415](#). General optimization tips are in [General Optimization Tips, on page 414](#).

- Use realistic design constraints, about 10 to 15 percent of the real goal. Over-constraining your design can be counter-productive because you can get poor implementations. Typically, you set timing constraints like clock frequency, clock-to-clock delay paths, I/O delays, register I/O delays and other miscellaneous path delays. Use clock, false path, and multi-cycle path constraints to make the constraints realistic.
- Enable the Retiming option. This optimization moves registers into I/O buffers if this is permitted by the technology and the design. However, it may add extra registers when clouds of logic are balanced across more than one register-to-register timing path. Extra registers are only added in parallel within the timing path and only if no extra latency is added by the additional registers. For example, if registers are moved across a 2x1 multiplexer, the tool adds two new registers to accommodate the select and data paths.

You can set this option globally or on specific registers. When it is enabled, it automatically enables pipelining as well. See [Retiming, on page 418](#) for details.

- Enable the Pipelining option. With this optimization enabled, the tool moves existing registers into a ROM or multiplier. Unlike retiming, it does not add any new logic. Pipelining reduces routing and delay and the extra instance delay of the external register by moving it into the ROM or multiplier and making it a built-in register.
- Select a balanced fanout constraint. A large constraint creates nets with large fanouts, and a low fanout constraint results in replicated logic. See [Setting Fanout Limits, on page 430](#) for information about setting limits and using the `syn_maxfan` attribute. You can use this in conjunction with the `syn_replicate` attribute that controls register duplication and buffering.
- Control register duplication and buffering criteria with the `syn_replicate` attribute. The tool automatically replicates registers during optimization, and you can use this attribute globally or locally on a specific register to turn off register duplication. See [Controlling Buffering and Replication](#),

[on page 432](#) for a description. Use `syn_replicate` in conjunction with the `syn_maxfan` attribute that controls fanout.

- If the critical path goes through arithmetic components, try disabling Resource Sharing. You can get faster times at the expense of increased area, but use this technique carefully. Adding too many resources can cause longer delays and defeat your purpose.
- If the P&R and synthesis tools report different critical paths, use a timing constraint with the `-route` option. With this option, the software adds route delay to its calculations when trying to meet the clock frequency goal. Use realistic values for the constraints.
- For FSMs, use the onehot encoding style, because it is often the fastest implementation. If a large output decoder follows an FSM, gray or sequential encoding could be faster.
- For designs with black boxes, characterize the timing models accurately, using the `syn_tpd`, `syn_tco`, and `syn_tso` directives.
- If you see warnings about feedback muxes being created for signals when you compile your source code, make sure to assign set/resets for the signals. This improves performance by eliminating the extra mux delay on the input of the register.
- Make sure that you pass your timing constraints to the place-and-route tools, so that they can use the constraints to optimize timing.

Retiming

Retiming improves the timing performance of sequential circuits without modifying the source code. It automatically moves registers (register balancing) across combinational gates or LUTs to improve timing while maintaining the original behavior as seen from the primary inputs and outputs of the design. Retiming moves registers across gates or LUTs, but does not change the number of registers in a cycle or path from a primary input to a primary output. However, it can change the total number of registers in a design.

The retiming algorithm retimes only edge-triggered registers. It does not retime level-sensitive latches. Note that registers associated with RAMS, DSPs, and the mapping for generated clocks may be moved, regardless of the Retiming option setting. The Retiming option is not available if it does not apply to the family you are using.

These sections contain details about using retiming.

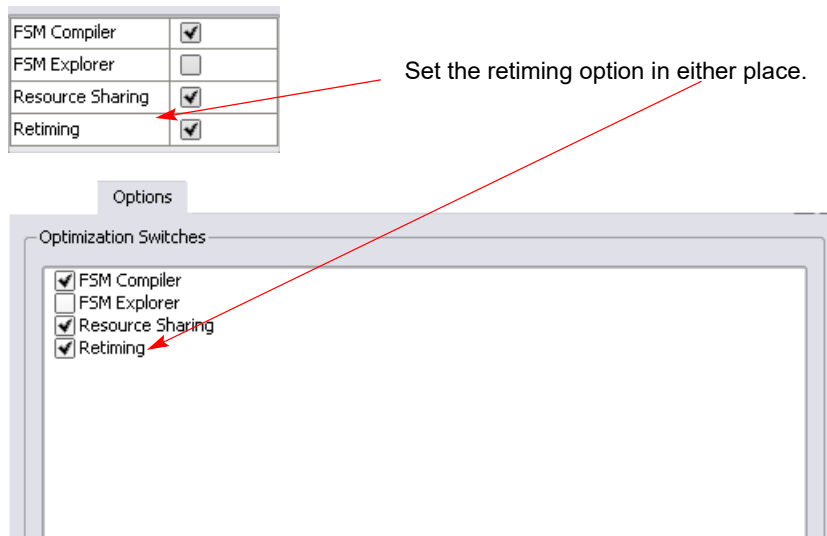
- [Controlling Retiming](#), on page 418
- [Retiming Example](#), on page 420
- [Retiming Report](#), on page 421
- [How Retiming Works](#), on page 422

Controlling Retiming

The following procedure shows you how to use retiming.

1. To enable retiming for the whole design, check the Retiming check box.

You can set the Retiming option from the button panel in the Project window, or with the Project->Implementation Options command (Options tab). The option is only available in certain technologies.



For Lattice designs, retiming is a superset of pipelining, so when you select Retiming, you automatically select Pipelining. See [Retiming, on page 418](#) for more information.

Retiming works globally on the design, and moves edge-triggered registers as needed to balance timing.

2. To enable retiming on selected registers, use either of the following techniques:
 - Check the Retiming checkbox and attach the `syn_allow_retiming` attribute with a value of 0 or false to any registers you do not want the software to move. This attribute specifies that the register cannot be moved for retiming. Refer to [How Retiming Works, on page 422](#) for a list of the components the retiming algorithm will move.
 - Do not check the Retiming checkbox. Attach the `syn_allow_retiming` attribute with a value of 1 or true to any registers you want the software to consider for retiming. You can do this in the SCOPE interface or in the source code. This attribute marks the register as one that can be moved during retiming, but does not necessarily force it to be moved during retiming. If you apply the attribute to an FSM, RAM or SRL that is decomposed into flip-flops and logic, the software applies the attribute to all the resulting flip-flops.

3. You can also fine-tune retiming using attributes:
 - To preserve the power-on state of flip-flops without sets or resets (FD or FDE) during retiming, set `syn_preserve=1` or `syn_allow_retiming=0` on these flip-flops.
 - To force flip-flops to be packed in I/O pads, set `syn_useioff=1` as a global attribute. This will prevent the flip-flops from being moved during retiming.
4. Set other options for the run. Retiming might affect some constraints and attributes. See [How Retiming Works, on page 422](#) for details.
5. Click Run to start synthesis.

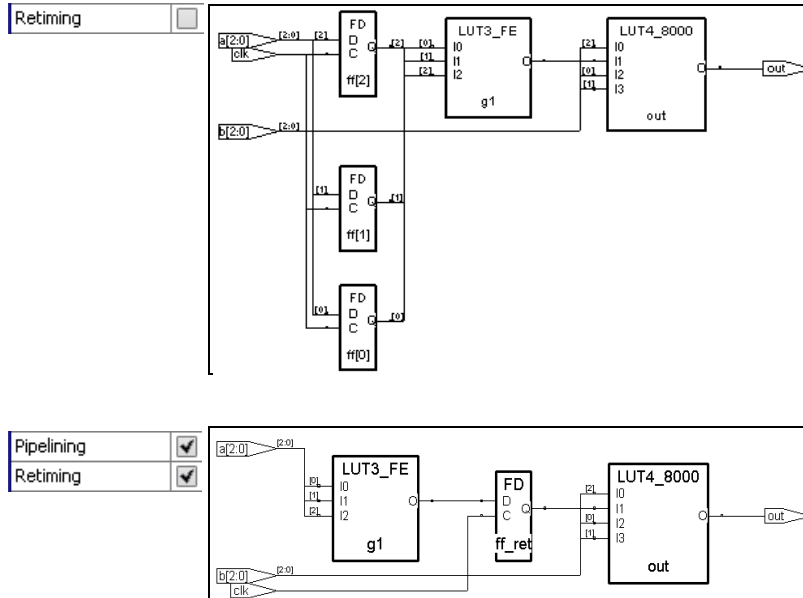
After the LUTs are mapped, the software moves registers to optimize timing. See [Retiming Example, on page 420](#) for an example. The software honors other attributes you set, like `syn_preserve`, `syn_useioff`, and `syn_ramstyle`. See [How Retiming Works, on page 422](#) for details.

Note that the tool might retime registers associated with RAMs, DSPs, and generated clocks, regardless of whether the Retiming option is on or off.

The log file includes a retiming report that you can analyze to understand the retiming changes. It contains a list of all the registers added or removed because of retiming. Retimed registers have a `_ret` suffix added to their names. See [Retiming Report, on page 421](#) for more information about the report.

Retiming Example

The following example shows a design with retiming disabled and enabled.



The top figure shows two levels of logic between the registers and the output, and no levels of logic between the inputs and the registers.

The bottom figure shows the results of retiming the three registers at the input of the OR gate. The levels of logic from the register to the output are reduced from two to one. The retimed circuit has better performance than the original circuit. Timing is improved by transferring one level of logic from the critical part of the path (register to output) to the non-critical part (input to register).

Retiming Report

The retiming report is part of the log file, and includes the following:

- The number of registers added, removed, or untouched by retiming.
- Names of the original registers that were moved by retiming and which no longer exist in the Technology view.
- Names of the registers created as a result of retiming, and which did not exist in the RTL view. The added registers have a `_ret` suffix.

How Retiming Works

This section describes how retiming works when it moves sequential components (flip-flops). Registers associated with RAMs, DSPs, and the mapping for fixing generated clocks might be moved, whether Retiming is enabled or not. Here are some implications and results of retiming:

- Flip-flops with no control signals (resets, presets, and clock enables) are moved. Flip-flops with minimal control logic can also be retimed. Multiple flip-flops with reset, set or enable signals that need to be retimed together are only retimed if they have exactly the same control logic.
- The software does not retime the following combinational sequential elements: flip-flops with both set and reset, flip-flops with attributes like `syn_preserve`, flip-flops packed in I/O pads, level-sensitive latches, registers that are instantiated in the code, SRLs, and RAMs. If a RAM with combinational logic has `syn_ramstyle` set to `registers`, the registers can be retimed into the combinational logic.
- Retimed flip-flops are only moved through combinational logic. The software does not move flip-flops across the following objects: black boxes, sequential components, tristates, I/O pads, instantiated components, carry and cascade chains, and keepbufs.
- You might not be able to crossprobe retimed registers between the RTL and the Technology view, because there may not be a one-to-one correspondence between the registers in these two views after retiming. A single register in the RTL view might now correspond to multiple registers in the Technology view.
- Retiming affects or is affected by, these attributes and constraints:

Attribute/Constraint	Effect
False path constraint	Does not retime flip-flops with different false path constraints. Retimed registers affect timing constraints.
Multicycle constraint	Does not retime flip-flops with different multicycle constraints. Retimed registers affect timing constraints.
Register constraint	Does not maintain <code>set_reg_input_delay</code> and <code>set_reg_output_delay</code> constraints. Retimed registers affect timing constraints.

Attribute/Constraint	Effect
from/to timing exceptions	If you set a timing constraint using a from/to specification on a register, it is not retimed. The exception is when using a max_delay constraint. In this case, retiming is performed but the constraint is not forward annotated. (The max_delay value would no longer be valid.)
syn_hier=macro	Does not retime registers in a macro with this attribute.
syn_keep	Does not retime across keepbufs generated because of this attribute.
syn_hier=macro	Does not retime registers in a macro with this attribute.
syn_preserve	Does not retime flip-flops with this attribute set.
syn_probe	Does not retime net drivers with this attribute. If the net driver is a LUT or gate, no flip-flops are retimed across it.
syn_reference_clock	On a critical path, does not retime registers with different syn_reference_clock values together, because the path effectively has two different clock domains.
syn_useioff	Does not override attribute-specified packing of registers in I/O pads. If the attribute value is false, the registers can be retimed. If the attribute is not specified, the timing engine determines whether the register is packed into the I/O block.
syn_allow_retiming	Registers are not retimed if the value is 0.

- Retiming does not change the simulation behavior (as observed from primary inputs and outputs) of your design, However if you are monitoring (probing) values on individual registers inside the design, you might need to modify your test bench if the probe registers are retimed.

- Beginning with the C-2009.09-SP1 release, the behavior for retiming unconstrained I/O pads has changed. If retiming is enabled, registers connected to unconstrained I/O pins are not retimed by default. If you want to revert back to how retiming I/O paths was previously implemented, you can:
 - Globally turn on the Use clock period for unconstrained IO switch from the Constraints tab of the Implementation Options panel.
 - Add constraints to all input/output ports.
 - Separately constrain each I/O pin as required.

Preserving Objects from Being Optimized Away

Synthesis can collapse or remove nets during optimization. If you want to retain a net for simulation, probing, or for a different synthesis implementation, you must specify this with an attribute. Similarly, the software removes duplicate registers or instances with unused output. If you want to preserve this logic for simulation or analysis, you must use an attribute. The following table lists the attributes to use in each situation. For details about the attributes and their syntax, see the *Attributes Reference Manual*.

To Preserve...	Use...	Result
Nets	<code>syn_keep</code> on wire or reg (Verilog), or signal (VHDL).	Keeps net for simulation, a different synthesis implementation, or for passing to the place-and-route tool.
Nets for probing	<code>syn_probe</code> on wire or reg (Verilog), or signal (VHDL)	Preserves internal net for probing.
Shared registers	<code>syn_keep</code> on input wire or signal of shared registers	Preserves duplicate driver cells, prevents sharing. See Using <code>syn_keep</code> for Preservation or Replication , on page 426 for details on the effects of applying <code>syn_keep</code> to different objects.
Sequential components	<code>syn_preserve</code> on reg or module (Verilog), signal or architecture (VHDL)	Preserves logic of constant-driven registers, keeps registers for simulation, prevents sharing
FSMs	<code>syn_preserve</code> on reg or module (Verilog), signal (VHDL)	Prevents the output port or internal signal that holds the value of the state register from being optimized
Instantiated components	<code>syn_noprune</code> on module or component (Verilog), architecture or instance (VHDL)	Keeps instance for analysis, preserves instances with unused outputs

See the following for more information:

- [Using `syn\_keep` for Preservation or Replication](#), on page 426
- [Controlling Hierarchy Flattening](#), on page 428
- [Preserving Hierarchy](#), on page 429

Using `syn_keep` for Preservation or Replication

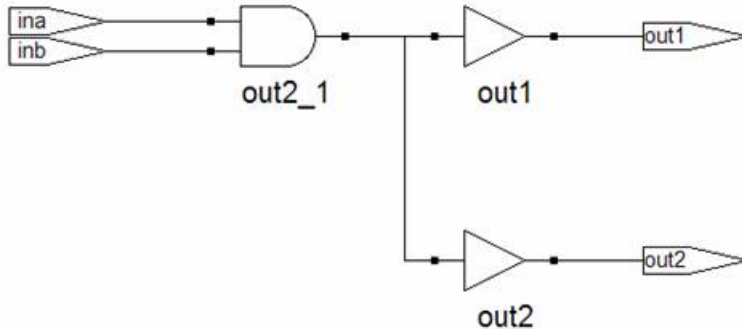
By default the tool considers replicated logic redundant, and optimizes it away. If you want to maintain the redundant logic, use `syn_keep` to preserve the logic that would otherwise be optimized away.

The following Verilog code specifies a replicated AND gate:

```
module redundant1(ina,inb,out1);
input  ina,inb;
output out1,out2;
wire out1;
wire out2;

assign out1 = ina & inb;
assign out2 = ina & inb;
endmodule
```

The compiler implements the AND function by replicating the outputs `out1` and `out2`, but optimizes away the second AND gate because it is redundant.



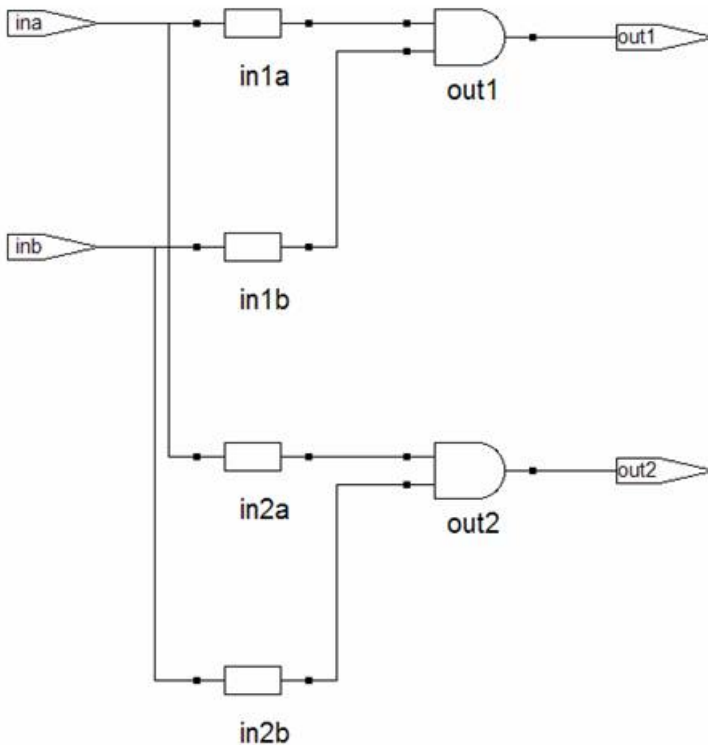
To replicate the AND gate in the previous example, apply `syn_keep` to the input wires, as shown below:

```
module redundant1d(ina,inb,out1,out2);
input ina,inb;
output out1,out2;
wire out1;
wire out2;

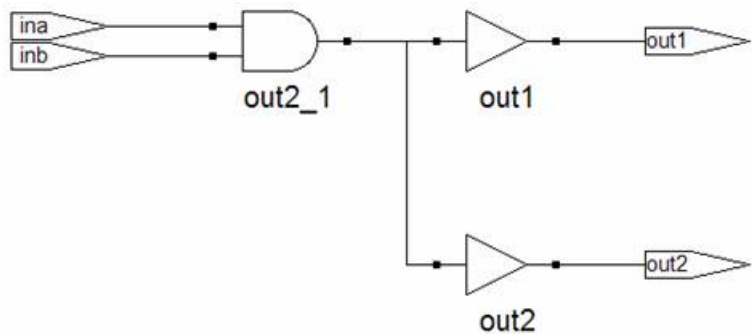
wire in1a /*synthesis syn_keep = 1*/;
wire in1b /*synthesis syn_keep = 1*/;
wire in2a /*synthesis syn_keep = 1*/;
wire in2b /*synthesis syn_keep = 1 */;
```

```
assign in1a = ina;  
assign in1b = inb;  
assign in2a = ina;  
assign in2b = inb;  
assign out1 = in1a & in1b;  
assign out2 = in2a & in2b;  
endmodule
```

Setting `syn_keep` on the input wires ensures that the second AND gate is preserved:



You must set `syn_keep` on the input wires of an instance if you want to preserve the logic, as in the replication of this AND gate. If you set it on the outputs, the instance is not replicated, because `syn_keep` preserves the nets but not the function driving the net. If you set `syn_keep` on the outputs in the example, you get only one AND gate, as shown in the next figure.



Controlling Hierarchy Flattening

Optimization flattens hierarchy. To control the flattening, use the `syn_hier` attribute as described here. You can also use the attribute to prevent flattening, as described in [Preserving Hierarchy, on page 429](#).

1. Attach the `syn_hier` attribute with the value you want to the module or architecture you want to preserve.

To...	Value...
Flatten all levels below, but not the current level	<code>flatten</code>
Remove the current level of hierarchy without affecting the lower levels	<code>remove</code>
Remove the current level of hierarchy and the lower levels	<code>flatten, remove</code>
Flatten the current level (if needed for optimization)	<code>soft</code>

You can also add the attribute in SCOPE instead of the HDL code. If you use SCOPE to enter the attribute, make sure to use the `v:` syntax. For details, see [syn_hier, on page 122](#) in the *Attribute Reference Manual*.

The software flattens the design as directed. If there is a lower-level `syn_hier` attribute, it takes precedence over a higher-level one.

2. If you want to flatten the entire design, use the `syn_netlist_hierarchy` attribute set to `false`, instead of the `syn_hier` attribute.

This flattens the entire netlist and does not preserve any hierarchical boundaries. See [syn_netlist_hierarchy, on page 158](#) in the *Attribute Reference Manual* for the syntax.

Preserving Hierarchy

The synthesis process includes cross-boundary optimizations that can flatten hierarchy. To override these optimizations, use the `syn_hier` attribute as described here. You can also use this attribute to direct the flattening process as described in [Controlling Hierarchy Flattening, on page 428](#).

1. Attach the `syn_hier` attribute to the module or architecture you want to preserve. You can also add the attribute in SCOPE. If you use SCOPE to enter the attribute, make sure to use the `v:` syntax.
2. Set the attribute value:

To...	Value...
Preserve the interface but allow cell packing across the boundary	firm
Preserve the interface and contents with no exceptions	macro
Flatten lower levels but preserve the interface of the specified design unit	flatten, firm

The software flattens the design as directed. If there is a lower-level `syn_hier` attribute, it takes precedence over a higher-level one.

Optimizing Fanout

You can optimize your results with attributes and directives, some of which are specific to the technology you are using. Similarly, you can use specify objects or hierarchy that you want to preserve during synthesis. For a complete list of all the directives and attributes, see the *Attribute Reference Manual*. This section describes the following:

- [Setting Fanout Limits](#), on page 430
- [Controlling Buffering and Replication](#), on page 432

Setting Fanout Limits

Optimization affects net fanout. If your design has critical nets with high fanout, you can set fanout limits. You can only do this for certain technologies. For details specific to individual technologies, see the *Reference Manual*.

1. To set a global fanout limit for the whole design, do either of the following:
 - Select Project-> Implementation Options->Device and type a value for the Fanout Guide option.
 - Apply the `syn_maxfan` attribute to the top-level view or module.

The value sets the number of fanouts for a given driver, and affects all the nets in the design. The defaults vary, depending on the technology. Select a balanced fanout value. A large constraint creates nets with large fanouts, and a low fanout constraint results in replicated or buffered logic. Both extremes affect routing and design performance. The right value depends on your design. The same value of 32 might result in fanouts of 11 or 12 and large delays on the critical path in one design or in excessive replication in another design.

The software uses the value as a soft limit, or a guide. It traverses the inverters and buffers to identify the fanout, and tries to ensure that all fanouts are under the limit by replicating or buffering where needed (see [Controlling Buffering and Replication](#), on page 432 for details). However, the synthesis tool does not respect the fanout limit absolutely; it ignores the limit if the limit imposes constraints that interfere with optimization.

2. To override the global fanout guideline and set a soft fanout limit at a lower level, set the `syn_maxfan` attribute on modules, views, or non-primitive instances.

These limits override the more global limits for that object. However, these limits still function as soft limits, and are replicated or buffered, as described in [Controlling Buffering and Replication, on page 432](#).

Attribute specified on ...	Effect
Module or view	Soft limit for the module; overrides the global setting.
Non-primitive instance	Soft limit; overrides global and module settings
Clock nets or asynchronous control nets	Soft limit.

3. To set a hard or absolute limit, set the `syn_maxfan` attribute on a port, net, register, or primitive instance.

Fanouts that exceed the hard limit are buffered or replicated, as described in [Controlling Buffering and Replication, on page 432](#).

4. To preserve net drivers from being optimized, attach the `syn_keep` or `syn_preserve` attributes.

For example, the software does not traverse a `syn_keep` buffer (inserted as a result of the attribute), and does not optimize it. However, the software can optimize implicit buffers created as a result of other operations; for example, it does not respect an implicit buffer created as a result of `syn_direct_enable`.

5. Check the results of buffering and replication in the following:
 - The log file (click View Log). The log file reports the number of buffered and replicated objects and the number of segments created for the net.
 - The HDL Analyst views. The software might not follow DRC rules when buffering or replicating objects, or when obeying hard fanout limits.

Controlling Buffering and Replication

To honor fanout limits (see [Setting Fanout Limits, on page 430](#)) and reduce fanout, the software either replicates components or adds buffers. The tool uses buffering to reduce fanout on input ports, and uses replication to reduce fanout on nets driven by registers or combinational logic. The software first tries replication, replicating the net driver and splitting the net into segments. This increases the number of register bits in the design. When replication is not possible, the software buffers the signals. Buffering is more expensive in terms of intrinsic delay and resource consumption. The following table summarizes the behavior.

Replicates When...	Creates Buffers When...
syn_replicate is 1	syn_replicate is 0. The syn_replicate attribute is used only to turn off the replication.
	syn_maxfan is set on a port/net that is driven by a port or I/O pad
	The net driver has a syn_keep or syn_preserve attribute
	The net driver is not a primitive gate or register

You can control whether high fanout nets are buffered or replicated, using the techniques described here:

- To use buffering instead of replication, set syn_replicate with a value of 0 globally, or on modules or registers. The syn_replicate attribute prevents replication, so that the software uses buffering to satisfy the fanout limit. For example, you can prevent replication between clock boundaries for a register that is clocked by clk1 but whose fanin cone is driven by clk2, even though clk2 is an unrelated clock in another clock group.
- To specify that high-fanout clock ports should not be buffered, set syn_noclockbuf globally, or on individual input ports. Use this if you want to save clock buffer resources for nets with lower fanouts but tighter constraints.
- Inverters merged with fanout loads increase fanout on the driver during placement and routing. A distinction is made between a keep buffer created as the result of the syn_keep attribute being applied by the user (explicit keep buffer) and a keep buffer that exists as the result of another attribute (implicit keep buffer). For example, the syn_direct_enable

attribute inserts a keep buffer. When a `syn_maxfan` attribute is applied to the output of an explicit keep buffer, the signal is buffered (the keep buffer is not traversed so that the driver is not replicated). When the `syn_maxfan` attribute is applied to the output of an implicit keep buffer, the keep buffer is traversed and the driver is replicated.

- Turn off buffering and replication entirely, by setting `syn_maxfan` to a very high number, like 1000.

Sharing Resources

One of the ways to optimize area is to use resource sharing in the compiler. With resource sharing, the software uses the same arithmetic operators for mutually exclusive statements; for example, with the branches of a case statement. Conversely, you can improve timing by disabling resource sharing, but at the expense of increased area.

Compiler resource sharing is on by default. You can set it globally and then override the global setting on individual modules

1. To disable resource sharing globally for the whole design, use one of the methods below.

Leave the default setting to improve area; disable the option to improve timing.

- Select Project->Implementation Options->Options, disable Resource Sharing. Alternatively, disable the Resource Sharing button on the left side of the Project view.
- Apply the `syn_sharing` directive to the top-level module or architecture in the source code. See [syn_sharing](#), on page 247 of the *Attribute Reference Manual* for syntax and examples.

```
Verilog module top(out, in, clk_in) /* synthesis syn_sharing = "off" */;
```

```
VHDL architecture rtl of top is
    attribute syn_sharing : string;
    attribute syn_sharing of rtl : architecture is "false";
```

- Edit your project file and include the following command: `set_option -resource_sharing 0`

When you save the project file, it includes the Tcl `set_option -resource_sharing` command.

You cannot specify `syn_sharing` from the SCOPE interface, because it is a compiler directive, and works during the compilation stage of synthesis. The resource sharing setting does not affect the mapper, so even if resource sharing is disabled, the tool can share resources during the mapping phase to optimize the design and improve results.

2. To specify resource sharing on an individual basis or override the global setting, specify the `syn_sharing` attribute for the lower-level module/architecture.

Inserting I/Os

You can control I/O insertion globally, or on a port-by-port basis.

1. To control the insertion of I/O pads at the top level of the design, use the Disable I/O Insertion option as follows:

- Select Project->Implementation Options and click the Device panel.
- Enable the option (checkbox on) if you want to do a preliminary run and check the area taken up by logic blocks, before synthesizing the entire design.

Do this if you want to check the area your blocks of logic take up, before you synthesize an entire FPGA. If you disable automatic I/O insertion, you do not get *any* I/O pads in your design, unless you manually instantiate them.

- Leave the Disable I/O Insertion checkbox empty (disabled) if you want to automatically insert I/O pads for all the inputs, outputs and bidirectionals.

When this option is set, the software inserts I/O pads for inputs, outputs, and bidirectionals in the output netlist. Once inserted, you can override the I/O pad inserted by directly instantiating another I/O pad.

- For the most control, enable the option and then manually instantiate the I/O pads for specific pins, as needed.

For more information about using the Disable I/O Insertion option with Lattice iCE devices, see [syn_insert_pad](#), on page 136.

2. For Lattice designs, you can force I/O pads to be inserted for input ports that do not drive logic with the `syn_force_pads` attribute:
 - To force I/O pad insertion at the module level, set the `syn_force_pads` attribute on the module. Set the attribute value to 1. To disable I/O pad insertion at the module level, set the `syn_force_pads` attribute for the module to 0.

- To force I/O pad insertion on an individual port, set the `syn_force_pads` attribute on the port with a value to 1. To disable I/O insertion for a port, set the attribute on the port with a value of 0.

Enable this attribute to preserve user-instantiated pads, insert pads on unconnected ports, insert bi-directional pads on bi-directional ports instead of converting them to input ports, or insert output pads on unconnected outputs.

If you do not set the `syn_force_pads` attribute, the synthesis design optimizes any unconnected I/O buffers away.

Optimizing State Machines

You can optimize state machines with the symbolic FSM Compiler tool. The Symbolic FSM Compiler is an advanced state machine optimizer, it automatically recognizes state machines in your design and optimizes them. Unlike other synthesis tools that treat state machines as regular logic, the FSM Compiler extracts the state machines as symbolic graphs, and then optimizes them by re-encoding the state representations and generating a better logic optimization starting point for the state machines.

For more information, see the following:

- [Deciding when to Optimize State Machines](#), on page 437
- [Running the FSM Compiler](#), on page 438

Deciding when to Optimize State Machines

The FSM Compiler is an automatic tool for encoding state machines, but you can also specify FSMs manually with attributes. For more information about using attributes, see [Specifying FSMs with Attributes and Directives](#), on page 395.

Here are the main reasons to use the FSM Compiler:

- To generate better results for your state machines

The software uses optimization techniques that are specifically tuned for FSMs, like reachability analysis for example. The FSM Compiler also lets you convert an encoded state machine to another encoding style (to improve speed and area utilization) without changing the source. For example, you can use a onehot style to improve results.

- To debug the state machines

State machine description errors result in unreachable states, so if you have errors, you will have fewer states. You can check whether your source code describes your state machines correctly. You can also use the FSM Viewer to see a high-level bubble diagram and crossprobe from there. For information about the FSM Viewer, see [Using the FSM Viewer \(Standard\)](#), on page 353.

Running the FSM Compiler

You can run the FSM Compiler tool on the whole design or on individual FSMs. See the following:

- [Running the FSM Compiler on the Whole Design](#), on page 438
- [Running the FSM Compiler on Individual FSMs](#), on page 439

Running the FSM Compiler on the Whole Design

1. Enable the compiler by checking the Symbolic FSM Compiler box in one of these places:
 - The main panel on the left side of the project window
 - The Options tab of the dialog box that comes up when you click the Add Implementation/New Impl or Implementation Options buttons
2. To set a specific encoding style for a state machine, define the style with the `syn_encoding` attribute, as described in [Specifying FSMs with Attributes and Directives](#), on page 395.

If you do not specify a style, the FSM Compiler picks an encoding style based on the number of states.

3. Click Run to run synthesis.

The software automatically recognizes and extracts the state machines in your design, and instantiates a state machine primitive in the netlist for each FSM it extracts. It then optimizes all the state machines in the design, using techniques like reachability analysis, next state logic optimization, state machine re-encoding and proprietary optimization algorithms. Unless you specified an encoding style, the tool automatically selects the encoding style. If you did specify a style, the tool uses that style.

In the log file, the FSM Compiler writes a report that includes a description of each state machine extracted and the set of reachable states for each state machine.

4. Select View->View Log File and check the log file for descriptions of the state machines and the set of reachable states for each one. You see text like the following:


```

Extracted state machine for register cur_state
State machine has 7 reachable states with original encodings of:
0000001
0000010
0000100
0001000
0010000
0100000
1000000
....
original code -> new code
0000001 -> 0000001
0000010 -> 0000010
0000100 -> 0000100
0001000 -> 0001000
0010000 -> 0010000
0100000 -> 0100000
1000000 -> 1000000

```

5. Check the state machine implementation in the RTL and Technology views and in the FSM viewer.
 - In the RTL view you see the FSM primitive with one output for each state.
 - In the Technology view, you see a level of hierarchy that contains the FSM, with the registers and logic that implement the final encoding.
 - In the FSM viewer you see a bubble diagram and mapping information. For information about the FSM viewer, see [Using the FSM Viewer \(Standard\)](#), on page 353.
 - In the `statemachine.info` text file, you see the state transition information.

Running the FSM Compiler on Individual FSMs

If you have state machines that you do not want automatically optimized by the FSM Compiler, you can use one of these techniques, depending on the number of FSMs to be optimized. You might want to exclude state machines from automatic optimization because you want them implemented with a specific encoding or because you do not want them extracted as state machines. The following procedure shows you how to work with both cases.

1. If you have just a few state machines you do not want to optimize, do the following:
 - Enable the FSM Compiler by checking the box in the button panel of the Project window.
 - If you do not want to optimize the state machine, add the `syn_state_machine` directive to the registers in the Verilog or VHDL code. Set the value to 0. When synthesized, these registers are not extracted as state machines.

```
Verilog  reg [3:0] curstate /* synthesis syn_state_machine=0 */;
```

```
VHDL    signal curstate : state_type;
        attribute syn_state_machine : boolean;
        attribute syn_state_machine of curstate : signal is
        false;
```

- If you want to specify a particular encoding style for a state machine, use the `syn_encoding` attribute, as described in [Specifying FSMs with Attributes and Directives](#), on page 395. When synthesized, these registers have the specified encoding style.
- Run synthesis.

The software automatically recognizes and extracts all the state machines, except the ones you marked. It optimizes the FSMs it extracted from the design, honoring the `syn_encoding` attribute. It writes out a log file that contains a description of each state machine extracted, and the set of reachable states for each FSM.

2. If you have many state machines you do not want optimized, do this:
 - Disable the compiler by disabling the Symbolic FSM Compiler box in one of these places: the main panel on the left side of the project window or the Options tab of the dialog box that comes up when you click the Add Implementation or Implementation Options buttons. This disables the compiler from optimizing any state machine in the design. You can now selectively turn on the FSM compiler for individual FSMs.
 - For state machines you want the FSM Compiler to optimize automatically, add the `syn_state_machine` directive to the individual state registers in the VHDL or Verilog code. Set the value to 1. When synthesized, the FSM Compiler extracts these registers with the default encoding styles according to the number of states.

Verilog	<code>reg [3:0] curstate /* synthesis syn_state_machine=1 */;</code>
VHDL	<code>signal curstate : state_type; attribute syn_state_machine : boolean; attribute syn_state_machine of curstate : signal is true;</code>

- For state machines with specific encoding styles, set the encoding style with the `syn_encoding` attribute, as described in [Specifying FSMs with Attributes and Directives, on page 395](#). When synthesized, these registers have the specified encoding style.
- Run synthesis.

The software automatically recognizes and extracts only the state machines you marked. It automatically assigns encoding styles to the state machines with the `syn_state_machine` attribute, and honors the encoding styles set with the `syn_encoding` attribute. It writes out a log file that contains a description of each state machine extracted, and the set of reachable states for each state machine.

3. Check the state machine in the log file, the RTL and technology views, and the FSM viewer. For information about the FSM viewer, see [Using the FSM Viewer \(Standard\), on page 353](#).

Inserting Probes

Probes are extra wires that you insert into the design for debugging. When you insert a probe, the signal is represented as an output port at the top level. You can specify probes in the source code or by interactively attaching an attribute.

Specifying Probes in the Source Code

To specify probes in the source code, you must add the `syn_probe` attribute to the net. You can also add probes interactively, using the procedure described in [Adding Probe Attributes Interactively, on page 443](#).

1. Open the source code file.
2. For Verilog source code, attach the `syn_probe` attribute as a comment on any internal signal declaration:

```
module alu(out, opcode, a, b, sel);
    output [7:0] out;
    input [2:0] opcode;
    input [7:0] a, b;
    input sel;
    reg [7:0] alu_tmp /* synthesis syn_probe=1 */;
    reg [7:0] out;
//Other code
```

The value 1 indicates that probe insertion is turned on. For detailed information about Verilog attributes and examples of the files, see the *Attribute Reference Manual*.

To define probes for part of a bus, specify where you want to attach the probes; for example, if you specify `reg [1:0]` in the previous code, the software only inserts two probes.

3. For VHDL source code, add the `syn_probe` attribute as follows:

```
architecture rtl of alu is
    signal alu_tmp : std_logic_vector(7 downto 0);
    attribute syn_probe : boolean;
    attribute syn_probe of alu_tmp : signal is true;
--other code;
```

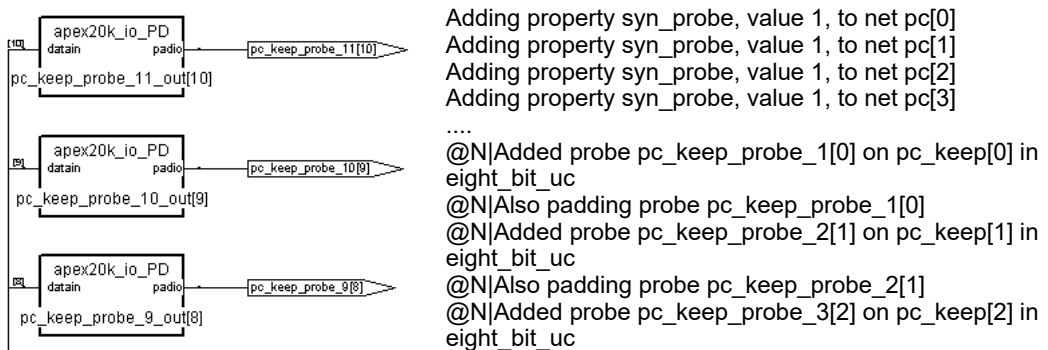
For detailed information about VHDL attributes and sample files, see the *Attribute Reference Manual*.

4. Run synthesis.

The software looks for nets with the `syn_probe` attribute and creates probes and I/O pads for them.

5. Check the probes in the log file (*.srr) and the Technology view.

This figure shows some probes and probe entries in the log file.



Adding Probe Attributes Interactively

The following procedure shows you how to insert probes by adding the `syn_probe` attribute through the SCOPE interface. Alternatively, you can add the attribute in the source code, as described in [Specifying Probes in the Source Code](#), on page 442.

1. Open the SCOPE window and click Attributes.
2. Push down as necessary in an RTL view, and select the net for which you want to insert a probe point.

Do not insert probes for output or bidirectional signals. If you do, you see warning messages in the log file.

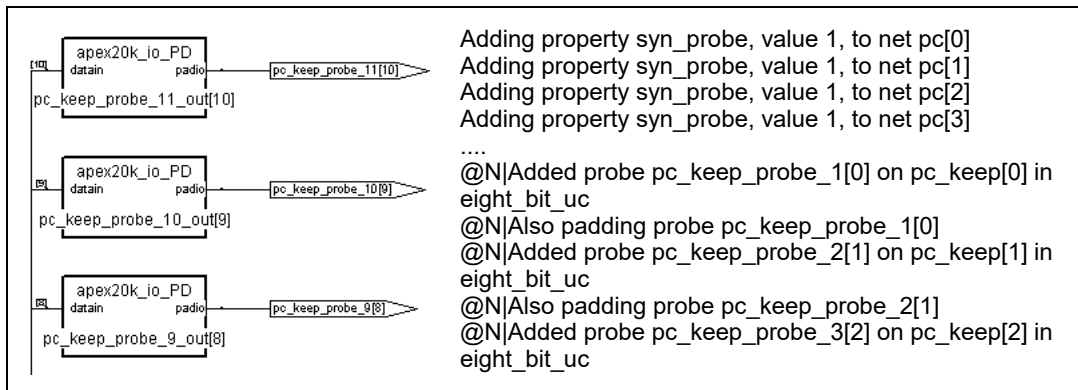
3. Do the following to add the attribute:
 - Drag the net into a SCOPE cell.

- Add the prefix `n:` to the net name in the SCOPE window. If you are adding a probe to a lower-level module, the name is created by concatenating the names of the hierarchical instances.
- If you want to attach probes to part but not all of a bus, make the change in the Object column. For example, if you enter `n:UC_ALU.longq[4:0]` instead of `n:UC_ALU.longq[8:0]`, the software only inserts probes where specified.
- Select `syn_probe` in the Attribute column, and type 1 in the Value column.
- Add the constraint file to the project list.

4. Rerun synthesis.

5. Open a Technology view and check the probe wires that have been inserted. You can use the Ports tab of the Find form to locate the probes.

The software adds I/O pads for the probes. The following figure shows some of the pads in the Technology view and the log file entries.



CHAPTER 11

Working with Compile Points

The following sections describe compile points and how to use them in logic synthesis iterative flows:

- [Compile Point Basics](#), on page 448
- [Compile Point Synthesis Basics](#), on page 456
- [Synthesizing Compile Points](#), on page 469
- [Resynthesizing Incrementally](#), on page 480

Compile Point Basics

Compile points are RTL partitions of the design that you define before synthesizing the design. Compile points can be defined manually, or the tool can generate them automatically. The software treats each compile point as a block, and can synthesize, optimize, place, and route the compile points independently. Compile points can be nested.

See the following topics for some details about compile points:

- [Advantages of Compile Point Design](#), on page 448
- [Nested Compile Points](#), on page 451
- [Compile Point Types](#), on page 452

Advantages of Compile Point Design

Designing with compile points makes it more efficient to work with the increasingly larger designs of today and the corresponding team approach to design. They offer several advantages, which are described here:

- [Compile Points and Design Flows](#), next
- [Runtime Savings](#), on page 449
- [Design Preservation](#), on page 449

Compile Points and Design Flows

Compile points improve the efficacy of both top-down and bottom-up design flows:

- In a traditional bottom-up design flow, compile points make it possible to easily divide up the design effort between designers or design teams. The compile points can be worked on separately and individually. The compile point synthesis flow eliminates the need to maintain the complex error-prone scripts for stitching, modeling, and ordering required by the traditional bottom-up design flow.
- From a top-down design flow perspective, compile points make it easier to work on the top-level design. You can mark compile points that are still being developed as black boxes, and synthesize the top level with what you have. You can also customize the compile point type settings

for individual compile points to take advantage of cross-boundary optimizations.

You can also synthesize incrementally, because the tool does not resynthesize compile points that are unchanged when you resynthesize the design. This saves runtime and also preserves parts of the design that are done while the rest of the design is completed.

See [Compile Point Synthesis, on page 463](#) for a description of the synthesis process with compile points.

Runtime Savings

Compile points are the required foundation for multiprocessing and incremental synthesis, both of which translate directly to runtime savings:

- Multiprocessing runs synthesis as multiple parallel processes, using the compile points as the partitions that are synthesized in parallel on different processors.
- Incremental synthesis uses compile points to determine which portions of the design to resynthesize, only resynthesizing the compile points that have been modified. See [Resynthesizing Compile Points Incrementally, on page 480](#).

Design Preservation

Using compile points addresses the need to maintain the overall stability of a design while portions of the design evolve. When you use compile points to partition the design, you can isolate one part from another. This lets you preserve some compile points, and only resynthesize those that need to be rerun. These scenarios describe some design situations where compile points can be used to isolate parts of the design and run incremental synthesis:

- During the initial design phase, design modules are still being designed. Use compile points to preserve unchanged design modules and evaluate the effects of modifications to parts of the design that are still changing.
- During design integration, use compile points to preserve the main design modules and only allow the glue logic to be remapped.
- If your design contains IP, synthesize the IP, and use compile points to preserve them while you run incremental synthesis on the rest of the design.

- In the final stages of the design, use compile points to preserve design modules that do not need to be updated while you work through minor RTL changes in some other part of the design.

Manual Compile Points

You can create compile points manually. Manual compile points require setup, but provide more control because they let you define the partition boundaries and constraints instead of the tool.

- Manual compile points (MCP)

Manual compile points provide more control. You can specify boundary constraints for each compile point individually. You can separate completed parts of the design from parts that are still being designed, or fine-tune the compile points to take advantage of as many cross-boundary optimizations as possible. For example, you can ensure that a critical path does not cross a compile point boundary, thus ensuring synthesis results with optimal performance.

Guidelines for Using Manual Compile Points

The table lists some guidelines:

Use Manual Compile Points...

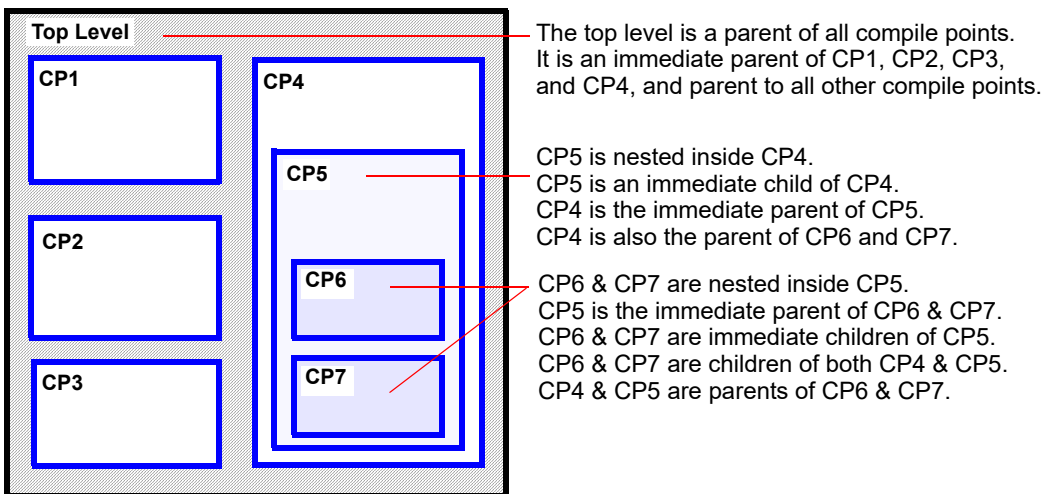
When you know the design in detail. Create manual compile points to get better QoR. Good candidates for manual compile points include the following:
• Completed modules with registered interfaces, where you want to preserve the design • Modules created to include an entire critical path, so as to get the best performance. • Modules that are less likely to be affected by cross boundary optimizations like constant propagation and register absorption.

When you do not want further optimizations to a completed compile point.
When you want more control to determine cross-boundary optimizations on an individual basis.

Nested Compile Points

A design can have any number of compile points, and compile points can be nested inside other compile points. In the following figure, compile point CP6 is nested inside compile point CP5, which is nested inside compile point CP4.

To simplify things, the term *child* is used to refer to a compile point that is contained inside another compile point; the term *parent* is used to refer to a container compile point that contains a child. These terms are not used in their strict sense of direct, immediate containment: If a compile point A is nested in B, which is nested in C, then A and B are both considered children of C, and C is a parent of both A and B. The top level is considered the parent of all compile points. In the figure above, both CP5 and CP6 are children of CP4; both CP4 and CP5 are parents of CP6; CP5 is an immediate child of CP4 and an immediate parent of CP6.



Compile Point Types

Compile point designs do not have as good QoR as designs without them because the boundaries limit optimizations. Cross-boundary optimizations typically improve area and timing, at the expense of runtime. The compile point type determines whether boundary optimizations are allowed. For manual compile points, you define the type. See [Defining the Compile Point Type](#), on page 475 for details.

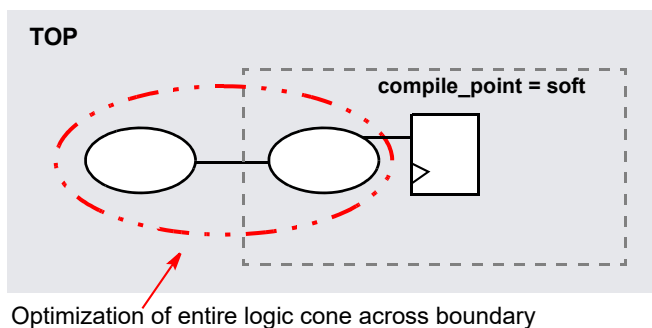
These are descriptions of the soft, hard, locked, and locked,partition compile types:

- Soft

Compile point boundaries can be reoptimized during top-level mapping. Timing optimizations like sizing, buffering, and DRC logic optimizations can modify boundary instances of the compile point and combine them with functions from the next higher level of the design. The compile point interface can also be modified. Multiple instances are uniquified. Any optimization changes can propagate both ways: into the compile point and from the compile point to its parent.

Using soft mode usually yields the best quality of results, because the software can utilize boundary optimizations. On the other hand, soft compile points can take a longer time to run than the same design with hard or locked compile points. Unless they are at the leaf level, soft compile points are not processed in parallel. Upper levels that contain soft compile points cannot be processed until the lower level has been mapped, with the top level processed last.

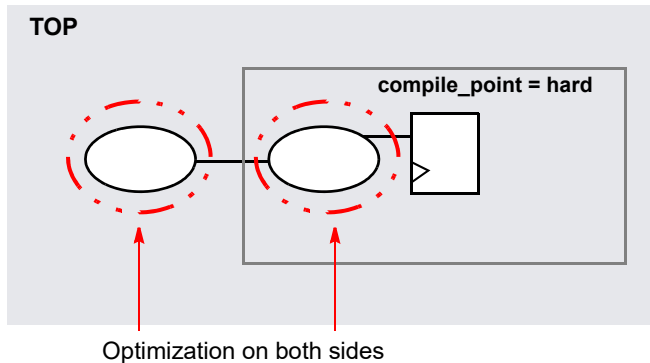
The following figure shows the soft compile point with a dotted boundary to show that logic can be moved in or out of the compile point.



- Hard

For hard compile points, the compile point boundary can be reoptimized during top-level mapping and instances on both sides of the boundary can be modified by timing and DRC optimizations using top-level constraints. However, the boundary is not modified. Any changes can propagate in either direction while the compile point boundary (port/interface) remains unchanged. Multiple instances are uniquified. For performance improvements, constant propagation and removal of unused logic optimizations are performed across hard compile points.

In the following figure, the solid boundary on the hard compile point indicates that no logic can be moved in or out of the compile point.



The hard compile point type allows for optimizations on both sides of the boundary without changing the boundary. There is a trade-off in quality of results to keep the boundaries. Using hard also allows for hierarchical equivalence checking for the compile point module.

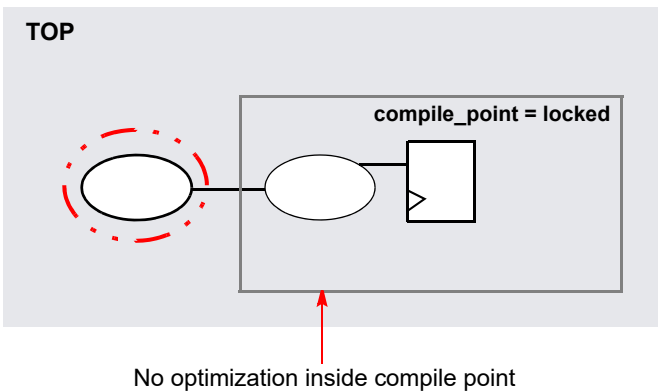
- Locked

This is the default compile point type for manual compile points. With a locked compile point, the tool does not make any interface changes or reoptimize the compile point during top-level mapping. An interface logic model (ILM) of the compile point is created (see [Interface Logic Models, on page 459](#)) and included for the top-level mapping. The ILM remains unchanged during top-level mapping.

The locked value indicates that all instances of the same compile point are identical and unaffected by top-level constraints or critical paths. As a result, multiple instances of the compile point module remain identical even though the compile point is uniquified. The Technology view (srm

file) shows unique names for the multiple instances, but in the final Verilog netlist (vma file) the original module names for the multiple instances are restored.

Timing optimization can only modify instances outside the compile point. Although the compile point is used to time the top-level netlist, changes do not propagate into or out of a locked compile point. The following figure shows a solid boundary for the locked compile point to indicate that no logic is moved in or out of the compile point during top-level mapping.



This mode has the largest trade-off in terms of QoR, because there are no boundary optimizations. So, it is very important to provide accurate constraints for locked compile points. The following table lists some advantages and limitations with the locked compile point:

Advantages	Limitations
Consumes smallest amount of memory. Used for large designs because of this memory advantage.	Interface timing
Provides most runtime advantage compared to other compile point types.	Constant propagation

Advantages	Limitations
Allows for obtaining stable results for a completed part of the design.	BUFG insertion
Allows for hierarchical place and route with multiple output netlists for each compile point and the top-level output netlist.	GSR hookup
Allows for hierarchical simulation.	IO pads, like IBUFs and OBUFs, should not be instantiated within compile points

- Locked, partition

You can also specify a compile point type to be locked, partition. This mode offers place-and-route runtime advantages and lets you converge on stable results for a completed design. However, this mode has the largest trade-off of quality of results because boundary optimizations are not allowed.

Compile Point Type Summary

The following table summarizes how the tool handles different compile points during synthesis:

Features	Compile Point Type		
	Soft	Hard	Locked
Boundary optimizations	Yes	Limited	No
Uniquification of multiple instance modules	Yes	Yes	Limited
Compile point interface (port definitions)	Modified	Not modified	Not modified
Hierarchical simulation	No	no	Yes
Hierarchical equivalence checking	No	Yes	Yes
Interface Logic Model (created/used)	No	No	Yes

Compile Point Synthesis Basics

This section describes the compile point constraint files and timing models, and describes the steps the tool goes through to synthesize compile points. See the following for details:

- [Compile Point Constraint Files](#), on page 456
- [Interface Logic Models](#), on page 459
- [Interface Timing for Compile Points](#), on page 459
- [Compile Point Synthesis](#), on page 463
- [Incremental Compile Point Synthesis](#), on page 464
- [Forward-annotation of Compile Point Timing Constraints](#), on page 467

For step-by-step information about how to use compile points, see [Synthesizing Compile Points](#), on page 469.

Compile Point Constraint Files

A compile point design can contain two levels of constraint files, as described below:

- The constraint file at the top level

This is a required file, and contains constraints that apply to the entire design. This file also contains the definitions of the compile points in the design. The `define_compile_point` command is automatically written to the top-level constraint file for each compile point you define.

The following figure shows that this design has one locked compile point, `prgm_cntr`. It uses the following syntax to define the compile point:

```
define_compile_point {v:work.prgm_cntr} -type {locked}
```



```

13
14 ##### BEGIN Collections - (Populated from tab in SCOPE, do not edit)
15 ##### END Collections
16
17 ##### BEGIN Clocks - (Populated from tab in SCOPE, do not edit)
18 create_clock {p:clock} -period {10}
19
20 ##### END Clocks
21
22 ##### BEGIN "Generated Clocks" - (Populated from tab in SCOPE, do not edit)
23 ##### END "Generated Clocks"
24
25 ##### BEGIN Inputs/Outputs - (Populated from tab in SCOPE, do not edit)
26 ##### END Inputs/Outputs
27
28 ##### BEGIN "Delay Paths" - (Populated from tab in SCOPE, do not edit)
29 ##### END "Delay Paths"
30
31 ##### BEGIN Attributes - (Populated from tab in SCOPE, do not edit)
32 ##### END Attributes
33
34 ##### BEGIN "I/O Standards" - (Populated from tab in SCOPE, do not edit)
35 ##### END "I/O Standards"
36
37 ##### BEGIN "Compile Points" - (Populated from tab in SCOPE, do not edit)
38 define_compile_point {v:work.prgm_cntr} -type {locked}
39 ##### END "Compile Points"
40

```

- Constraints files at the compile point level

These constraint files are optional, and are used for better control over manual compile points.

The compile point constraints are specific to the compile point and only apply within it. If your design has manual compile points, you can define corresponding compile point constraint files for them. See [Setting Constraints at the Compile Point Level, on page 476](#) for a step-by-step procedure.

When compile point constraints are defined, the tool uses them to synthesize the compile point, not automatic interface timing. Note that depending on the compile point type, the tool might further optimize the compile points during top-down synthesis of the top level to improve timing performance and overall design results, but the compile point itself is synthesized with the defined compile point constraints.

The first command in a compile point constraint file is `define_current_design`, and it specifies the compile point module for the contained constraints. This command sets the context for the constraint file. The

remainder of the file is similar to the top-level constraint file. For example:

```
define_current_design {work.pgrm_cntr}
```

```

12 define_current_design {work.pgrm_cntr}
13 ##### END Header
14
15 ##### BEGIN Collections - (Populated from tab in SCOPE, do not edit)
16 ##### END Collections
17
18 ##### BEGIN Clocks - (Populated from tab in SCOPE, do not edit)
19 create_clock {p:clock} -period {11.1111}
20
21 ##### END Clocks
22
23 ##### BEGIN "Generated Clocks" - (Populated from tab in SCOPE, do not edit)
24 ##### END "Generated Clocks"
25
26 ##### BEGIN Inputs/Outputs - (Populated from tab in SCOPE, do not edit)
27 ##### END Inputs/Outputs
28
29 ##### BEGIN "Delay Paths" - (Populated from tab in SCOPE, do not edit)
30 ##### END "Delay Paths"
31
32 ##### BEGIN Attributes - (Populated from tab in SCOPE, do not edit)
33 ##### END Attributes
34
35 ##### BEGIN "I/O Standards" - (Populated from tab in SCOPE, do not edit)
36 ##### END "I/O Standards"
37
38 ##### BEGIN "Compile Points" - (Populated from tab in SCOPE, do not edit)
39 ##### END "Compile Points"

```

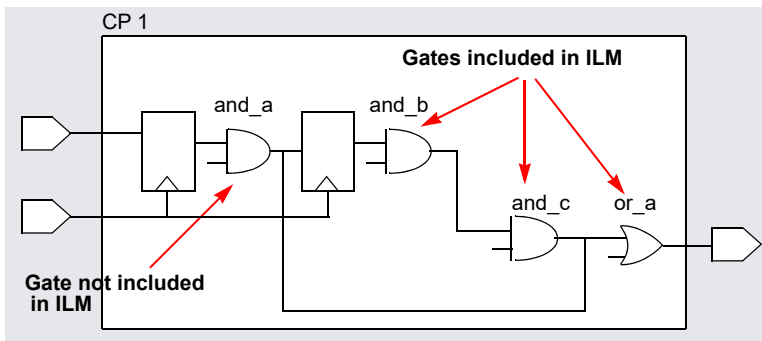
If your design has some compile points with their own constraint files and others without them, the tool uses the defined compile point constraints when it synthesizes those compile points. For the other compile points without defined constraints, it uses automatic interface timing, as described in [Interface Timing for Compile Points](#), on page 459.

Interface Logic Models

The interface logic model (ILM) of a locked or hard compile point is a timing model that contains only the interface logic necessary for accurate timing. An ILM is a partial gate-level netlist that represents the original design accurately while requiring less memory during mapping. Using ILMs improves the runtime for static timing analysis without compromising timing accuracy.

The tool does not do any timing optimizations on an ILM. The interface logic is preserved with no modifications. All logic required to recreate timing at the top level is included in the ILM. ILM logic includes any paths from an input/inout port to an internal register, an internal register to an output/inout port, and an input/inout port to an output/inout port.

The tool removes internal register-to-register paths, as shown in this example. In this design, `and_a` is not included in the ILM because the timing path that goes through `and_a` is an internal register-to-register path.



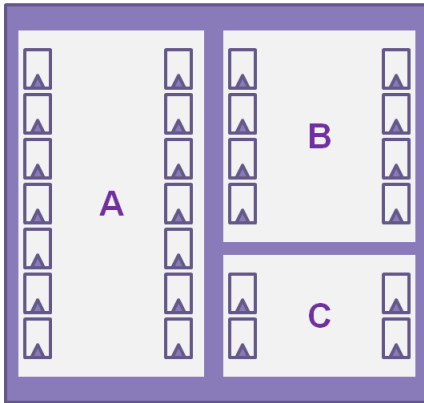
Interface Timing for Compile Points

By default, the synthesis tool automatically infers timing constraints for all compile points from the top-level constraints. However, if a compile point has its own constraint file, the tool applies those compile point-specific constraints to synthesize the compile point.

- For automatic interface timing, the tool derives constraints from the top level and uses them to synthesize the compile point. The top level is synthesized at the same time as the other compile points.

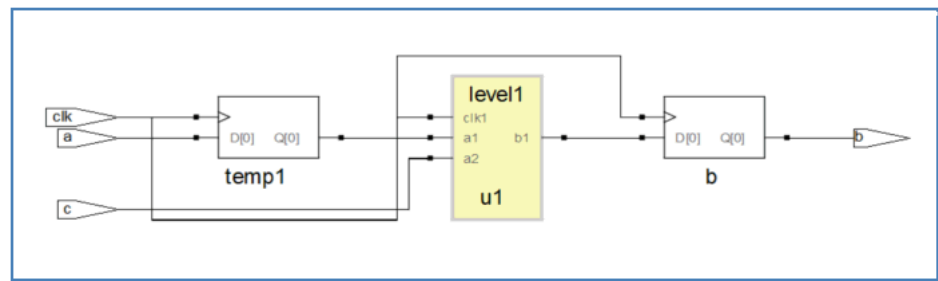
- When there are compile point constraint files, the tool first synthesizes the compile point using the constraints in the compile point constraints file and then synthesizes the top level using the top-level constraints.

When it synthesizes a compile point, the tool considers all other compile points as black boxes and only uses their interface timing information. In the following figure, when the tool is synthesizing compile point A, it applies relevant timing information to the boundary registers of B and C, because it treats them as black boxes.

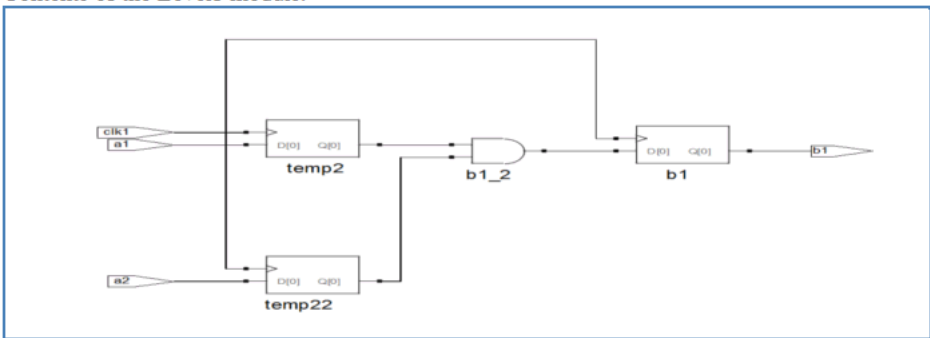


Interface Timing Example

The design below shows how the interface timing works on compile points.



Contents of level1 Module

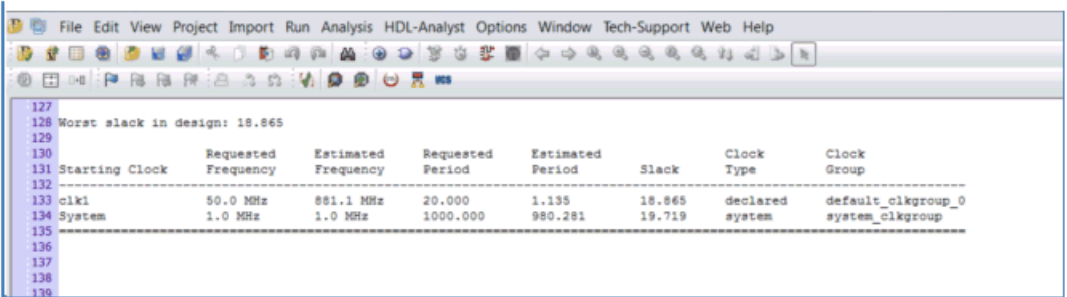


Interface Timing Off

Interface timing is off for a compile point when you define constraints for it in a compile point constraints file. In this example, the following frequencies are defined for the level1 compile point shown above:

Clock	Period	Constraints File
Top-level clock	10 ns	Top-level constraint file
Compile point-level clock	20 ns	Compile point constraint file

When interface timing is off, the compile point log file (`src`) reports the clock period for the compile point as 20 ns, which is the compile point period.

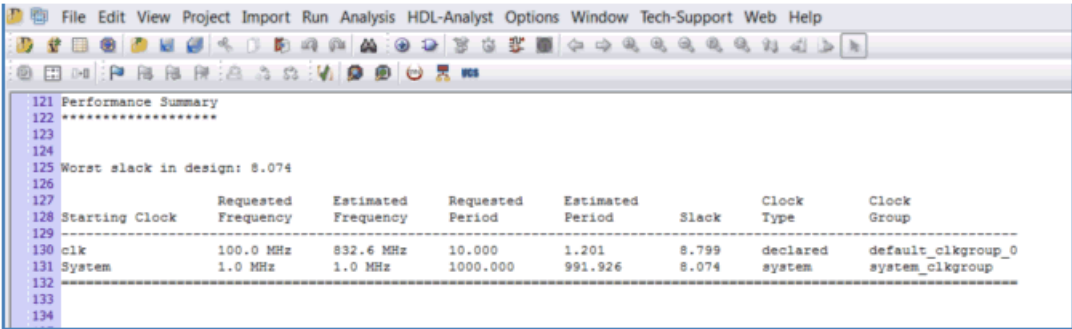


The screenshot shows the HDL-Analyst window with a menu bar (File, Edit, View, Project, Import, Run, Analysis, HDL-Analyst, Options, Window, Tech-Support, Web, Help) and a toolbar. The main display area shows the following text and table:

```
127
128 Worst slack in design: 18.865
129
130
131 Starting Clock      Requested Frequency    Estimated Frequency    Requested Period    Estimated Period    Slack    Clock Type    Clock Group
132 -----
133 clk1                50.0 MHz              881.1 MHz             20.000             1.135              18.865    declared    default_clkgroup_0
134 System              1.0 MHz               1.0 MHz               1000.000           980.281            19.719    system      system_clkgroup
135
136
137
138
139
```

Interface Timing On

For automatic interface timing to run on a compile point (interface timing on), there must not be a compile-point level constraints file. When interface timing is on, the compile point log file (srr) reports the clock period for the top-level design, which is 10 ns:



The screenshot shows the HDL-Analyst window with a menu bar (File, Edit, View, Project, Import, Run, Analysis, HDL-Analyst, Options, Window, Tech-Support, Web, Help) and a toolbar. The main display area shows the following text and table:

```
121 Performance Summary
122 *****
123
124
125 Worst slack in design: 8.074
126
127
128 Starting Clock      Requested Frequency    Estimated Frequency    Requested Period    Estimated Period    Slack    Clock Type    Clock Group
129 -----
130 clk                100.0 MHz             832.6 MHz             10.000             1.201              8.799    declared    default_clkgroup_0
131 System              1.0 MHz               1.0 MHz               1000.000           991.926            8.074    system      system_clkgroup
132
133
134
```

Compile Point Synthesis

During synthesis, the tool first synthesizes the compile points and then maps the top level. By default, synthesis stops if the tool encounters an error while synthesizing a compile point. You can specify that the tool ignore the error (compile on error option) and continue synthesizing other compile points. If you use this option, the tool black boxes the compile point with the error and continues with the rest of the design. The rest of this section describes the process that the tool goes through to synthesize compile points; for step-by-step information about what you need to do to use compile points, see [Synthesizing Compile Points, on page 469](#).

Manual Compile Point Synthesis

The following headings describe the process of manual compile point synthesis:

Stage 1: Bottom-up Compile Point Synthesis

The tool synthesizes compile points individually from the bottom up. If you have enabled multiprocessing, it synthesizes the compile points in parallel using multiple processing jobs. For nested compile points, it starts with the compile point at the lowest level of hierarchy and works up the hierarchy.

A compile point stands on its own, and is optimized separately from its parent environment (the compile point container or the top level). This means that critical paths from a higher level do not propagate downwards, and they are unaffected by them.

If you have specified compile point-level constraints, the tool uses them to synthesize the compile point; if not, it uses automatic interface timing propagated from the top level. For compile point synthesis, the tool assumes that all other compile points are black boxes, and only uses the interface information.

When defined, compile point constraints apply within the compile point. For manual compile points, it is recommended that you set constraints on locked compile points, but setting constraints is optional for soft and hard compile points.

By default, synthesis stops if the tool encounters an error while synthesizing a compile point.

Stage 2: Top-Level Synthesis

Once all the compile points have been synthesized, the tool synthesizes the entire design from the top down, using the model information generated for each compile point and constraints defined in the top-level constraints file. You do not need to duplicate compile point constraints at a higher level, because the tool takes the compile point timing models into account when it synthesizes a higher level. Note that if you run standalone timing analysis on a compile point, the timing report reflects the top-level constraints and not the compile point constraints, although the tool used compile point level constraints to synthesize the compile point.

The software writes out a single output netlist and one constraint file for the entire design. See [Forward-annotation of Compile Point Timing Constraints, on page 467](#) for a description of the constraints that are forward-annotated.

Incremental Compile Point Synthesis

The tool treats compile points as blocks for incremental synthesis. On subsequent synthesis runs, the tool runs incrementally and only resynthesizes those compile points that have changed, and the top level. The synthesis tool automatically detects design changes and resynthesizes compile points only if necessary. For example, it does not resynthesize a compile point if you only add or change a source code comment, because this change does not really affect the design functionality.

The tool resynthesizes a compile point that has already been synthesized, in any of these cases:

- The HDL source code defining the compile point is changed in such a way that the design logic is changed.
- The constraints applied to the compile point are changed.
- Any of the options on the Device panel of the Implementation Options dialog box, except Update Compile Point Timing Data, are changed. In this case the entire design is resynthesized, including all compile points.
- You intentionally force the resynthesis of your entire design, including all compile points, with the Run -> Resynthesize All command.
- The Update Compile Point Timing Data device mapping option is enabled and at least one child of the compile point (at any level) has been remapped. The option requires that the parent compile point be resynthesized using the updated timing model of the child. This includes the possibility that

the child was remapped earlier, while the option was disabled. The newly enabled option requires that the updated timing model of the child be taken into account, by resynthesizing the parent.

For each compile point, the software creates a subdirectory named for the compile point, in which it stores intermediate files that contain hierarchical interface timing and resource information that is used to synthesize the next level. Once generated, the model file is not updated unless there is an interface design change or you explicitly specify it. If you happen to delete these files, the associated compile point will be resynthesized and the files regenerated.

Checking Partition Timestamps

The synthesis tool forward-annotates partition timestamps for each manual compile point in the design.vm netlist. You can use this information to check whether the compile point was resynthesized in an incremental run of the tool.

1. Define compile point constraints on the modules in the design.

For example:

```
define_compile_point {viewName} -type {locked, partition}
-cpfile {fileName}
```

2. Run the standard synthesis flow. Synplify Pro writes timestamps for each compile point in the designName_partition.tcl file.

For example:

- Partition timestamp is annotated on the module viewName which was inferred as a manual compile point in the design.vm netlist.

```
(* PARTITION_TIMESTAMP = "1707268302"*) module viewName
```

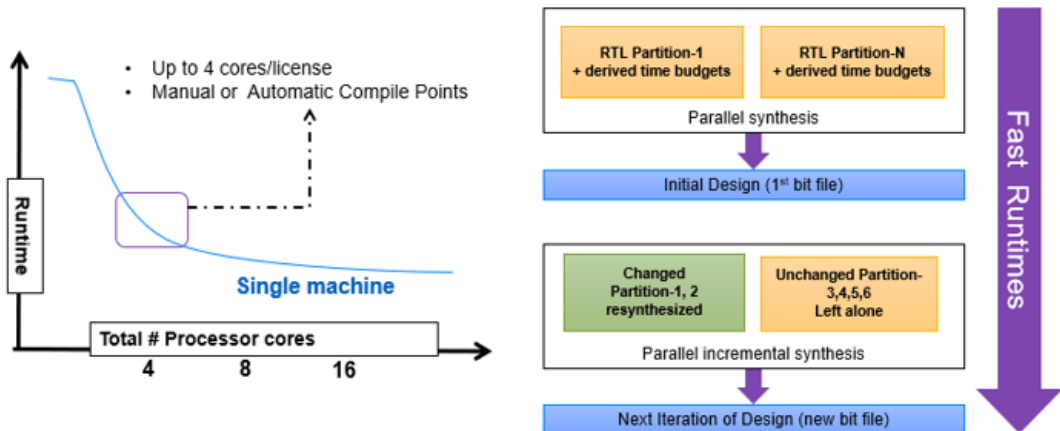
- Timestamps are displayed in the design_partition.tcl file:

```
set_partition_info -name partitionName -timestamp timestamp
```

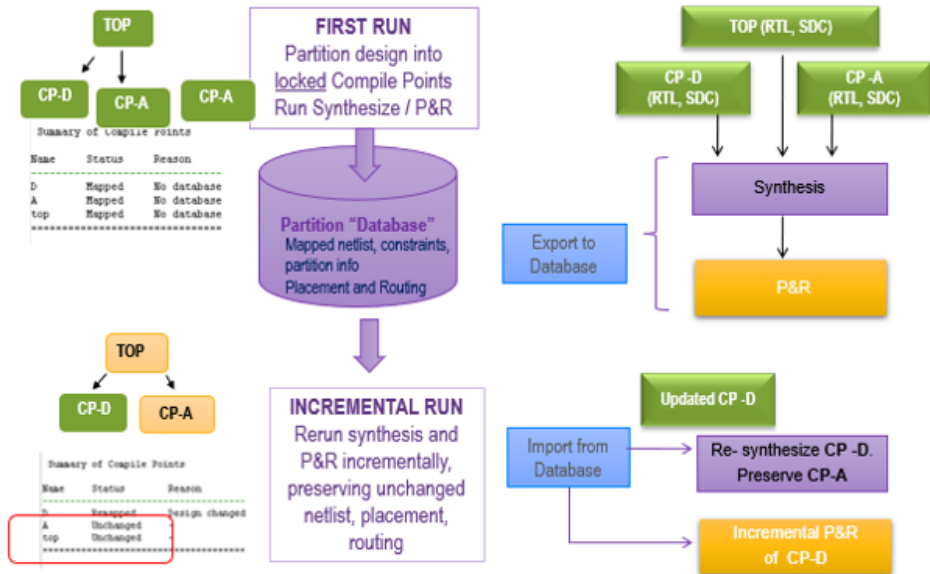
For an incremental synthesis run, only the affected compile points display new timestamps, while unaffected compile points retain the same timestamps. Check the Compile Point Summary report available in the log file.

Note: Generation of partition file is enabled for all compile points identified in the Automatic Compile Point (ACP) flow. You do not need to set any manual partitions.

Compile Point Flow with Multiprocessing and Incremental Design



Design Preservation with Compile Points



Forward-annotation of Compile Point Timing Constraints

In addition to a top-level constraint file, each compile point can have its own constraint file. Constraints are forward-annotated to placement and routing from the top-level as well as the compile point-level files. However, not all compile point constraints are forward-annotated, as explained below. For example, constraints on top-level ports are always forward annotated, but compile point port constraints are not forward annotated.

- Top-level constraints are forward-annotated.
- Constraints applied to the interface (ports and bit ports) of the compile point are not forward-annotated. These include `input_delays`, `output_delays`, and clock definitions on the ports. Such constraints are only used to map the compile point itself, not its parents. They are not used in the final timing report, and they are not forward-annotated.
- Constraints applied to instances inside the compile point are forward-annotated

Constraints like timing exceptions and internal clocks are used to map the compile point and its parents. They are used in the final timing report, and they are forward-annotated.

Synthesizing Compile Points

This section describes the synthesis process with manual compile points in your design:

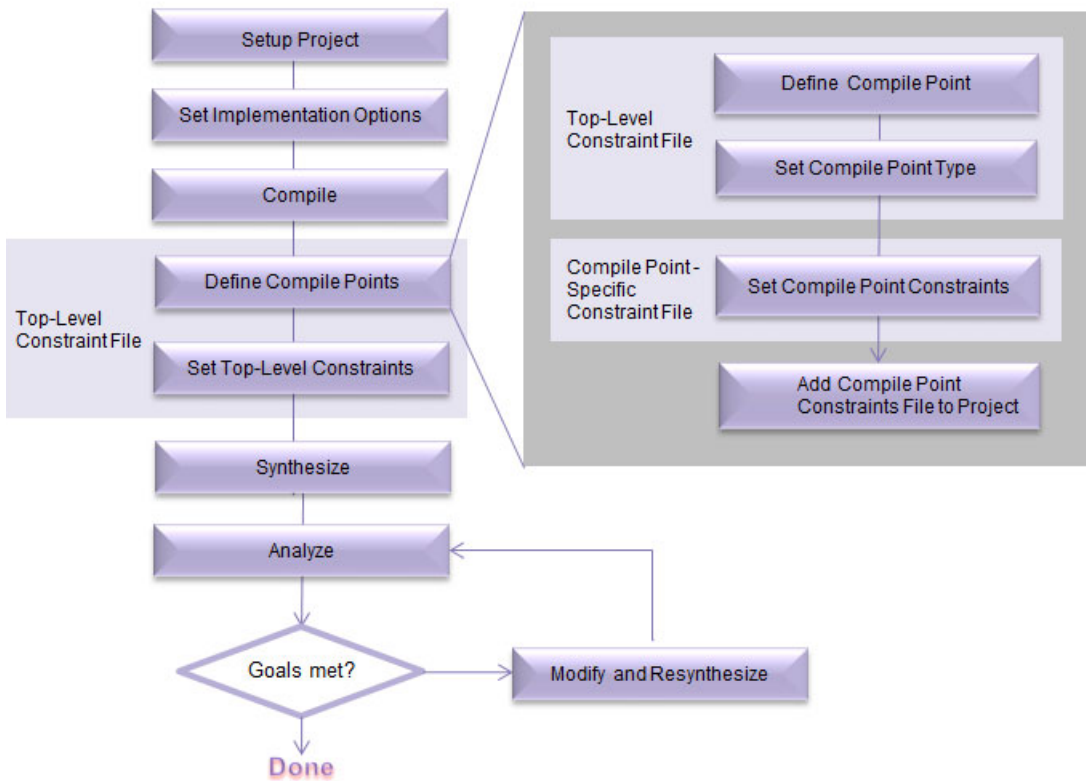
- [The Manual Compile Point Flow](#), on page 469
- [Creating a Top-Level Constraints File for Compile Points](#), on page 472
- [Defining Manual Compile Points](#), on page 473
- [Setting Constraints at the Compile Point Level](#), on page 476
- [Analyzing Compile Point Results](#), on page 478

The Manual Compile Point Flow

Using manual compile points is most advantageous in the following situations, where you

- Have to work with a large design
- Experience long runtimes, or need to reduce synthesis runtime
- Require the maximum QoR from logic synthesis
- Can adjust design methodology to get the best results from the tools

The following figure summarizes the process for using manual compile points in your design.



This procedure describes the steps in more detail:

1. Set up the project.
 - Create the project and add RTL and IP files to the project, as usual.
 - Target a device and technology for which compile points are supported.
 - Set other options as usual.
2. Compile the design (F7) to initialize the constraints file.
3. Do the following in the top-level constraint file:
 - Define compile points in the top-level constraint file. See [Creating a Top-Level Constraints File for Compile Points, on page 472](#). Note that by default, the tool automatically calculates the interface timing for all compile points.

- Set timing constraints and attributes in the top-level constraint file:

Constraint	Apply to...	Example
Clock	All clocks in the design.	<code>create_clock {p:clk} -name clk -period 100 -clockgroup cg1</code>
I/O constraints	All top-level port constraints. Register the compile point I/O boundaries to improve timing.	<code>set_input_delay {p:a} {1} -clock {clk:r}</code>
Timing exceptions	All timing exceptions that are outside the compile point module, or that might be partially in the compile point modules.	<code>set_false_path -from {i:reg1} -to {i:reg2}</code>
Attributes	All attributes that are applicable to the rest of the design, not within the compile points.	<code>define_attribute {i:statemachine_1} syn_encoding {sequential}</code>

4. Set compile point-specific constraints as needed in a separate, compile point-level constraint file.

See [Setting Constraints at the Compile Point Level, on page 476](#) for a step-by-step procedure. After setting the compile point constraints, add the compile point constraint file to the project.

5. Synthesize the design.

The tool synthesizes the compile points separately and then synthesizes the top level. See [Compile Point Synthesis, on page 463](#) for details about the process.

- The first time it runs synthesis, the tool maps the entire design.
- For subsequent synthesis runs, the tool only maps compile points that were modified since the last run. It preserves unchanged compile points.

You can also run synthesis on individual compile points, without synthesizing the whole design.

6. Analyze the synthesis results using the top-level srr log file.

See [Analyzing Compile Point Results, on page 478](#) for details.

- 7. If you do not meet your design goals, make necessary changes to the RTL, constraints, or synthesis controls, and re-synthesize the design.

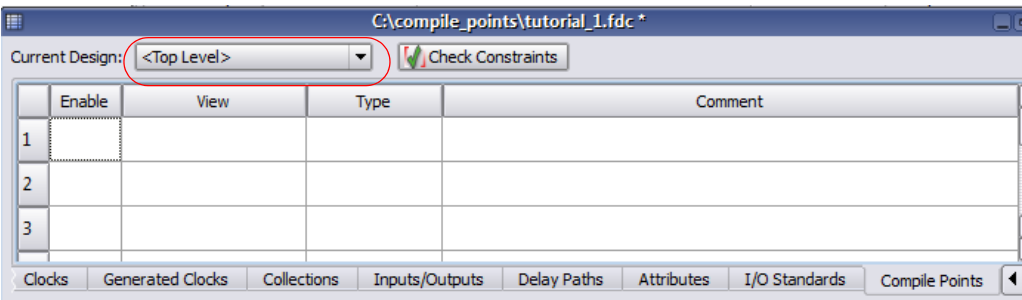
The tool runs incremental synthesis on the modified parts of the design, as described in [Incremental Compile Point Synthesis, on page 464](#). See [Resynthesizing Compile Points Incrementally, on page 480](#) for a detailed procedure.

Creating a Top-Level Constraints File for Compile Points

All compile points require a top-level constraints file. If you have manual compile points, define them in this file. The top-level file also contains design-level constraints. The following procedure describes how to create a top-level constraints file for a compile point design.

- 1. Create the top-level constraints file.
 - To define compile points in an existing top-level constraint file, open a SCOPE window by double-clicking the file in the Project view.
 - To define compile points in a new top-level constraint file, click the SCOPE icon. Click on the FPGA Constraints (SCOPE) button.

The SCOPE window opens. It includes a Current Design field, where you can specify constraints for the top-level design from the drop-down menu and define manual compile points.



- 2. Set top-level constraints like input/output delays, clock frequencies or multicycle paths.

You do not have to redefine compile point constraints at the top level as the tool uses them to synthesize the compile points.

- 3. Define manual compile points if needed.

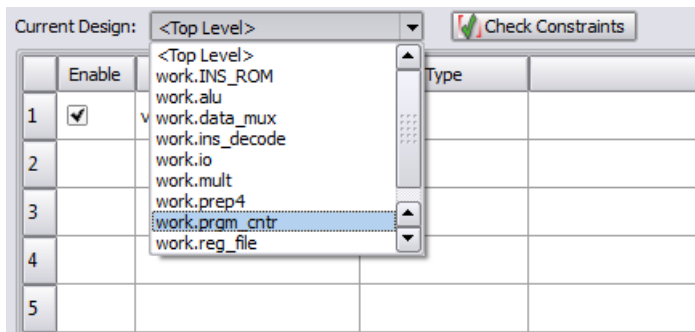
See [Defining Manual Compile Points, on page 473](#) for details.

4. Save the top-level constraints file and add it to the project.

Defining Manual Compile Points

Compile points and constraints are both saved in a constraint file, so this step can be combined with the setting of constraints, as convenient. This procedure only describes how to define compile points. You define compile points in a top-level constraint file. You can add the compile point definitions to an existing top-level constraint file or create a new file.

1. From the Current Design field, select the module for which you want to create the compile point.



2. Click the Compile Points tab in the top-level constraints file.

See [Creating a Top-Level Constraints File for Compile Points, on page 472](#) if you need information about creating this file.

3. Set the module you want as a compile point.

Do this by either selecting a module from the drop-down list in the View column, or dragging the instance from the HDL Analyst RTL view to the View column. The equivalent Tcl command is `define_compile_point`, as shown in this example:

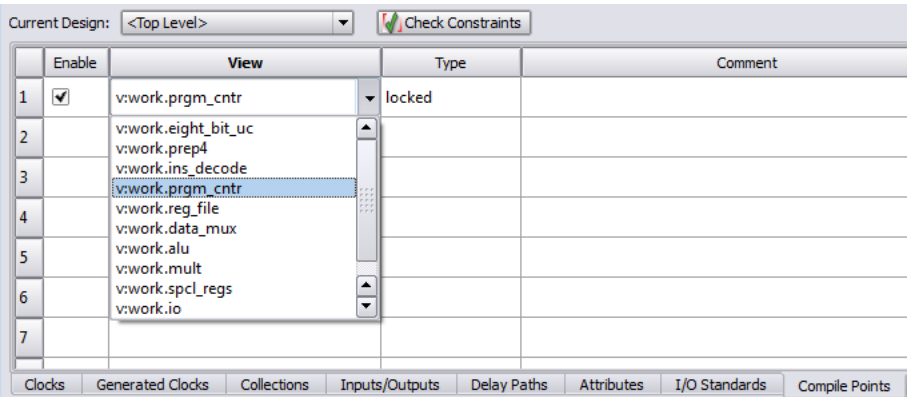
```
define_compile_point {v:work.m3} -type {black_box}
```

You can get a list of all the modules from which you can select and designate compile points with the Tcl find command, as shown here:

```
c_print [find -hier -view {*} -filter ((!@is_black_box) && (@is_verilog == 1 || @is_vhdl == 1))] -file view.txt
```

- 4. Set the Type to locked, hard, or soft, according to your design goals. See [Defining the Compile Point Type, on page 475](#) for details.

This tags the module as a compile point. The following figure shows the prgm_cntr module set as a locked compile point:



- 5. Save the top-level constraint file.

You can now open the compile point constraint file and define constraints for the compile point, as needed for manual compile points. See [Setting Constraints at the Compile Point Level, on page 476](#) for details.

Defining the Compile Point Type

The compile point type you select depends on your design goals. For descriptions of the various compile point types, see [Compile Point Types, on page 452](#). This procedure shows you how to set the compile point type in the top-level constraint file when you define the compile points:

1. When runtime is the main objective and QoR is not a primary concern, set the compile point type as follows on the SCOPE Compile Points tab:

Situation	Compile Point Type
RTL is almost ready	locked
RTL is still being built but the module interface is ready	black_box If you define black box compile points, you must update the value and rerun synthesis after you have completed the RTL for the compile point.

The following example shows the Tcl command and the equivalent version in the in the SCOPE GUI:

```
define_compile_point {v:work.user_top} -type {locked}
```

	Enable	View	Type
1	☒	v:work.user_top	locked

2. When runtime and QoR are both important, do the following to ensure the best performance while still saving runtime:
 - Register the I/O boundaries for the compile points.
 - As far as possible, put the entire critical path into the same compile point.
 - Set each compile point type individually, using these compile point types:

Situation	Compile Point Type
Need boundary optimizations	soft
Do not need boundary optimizations	locked

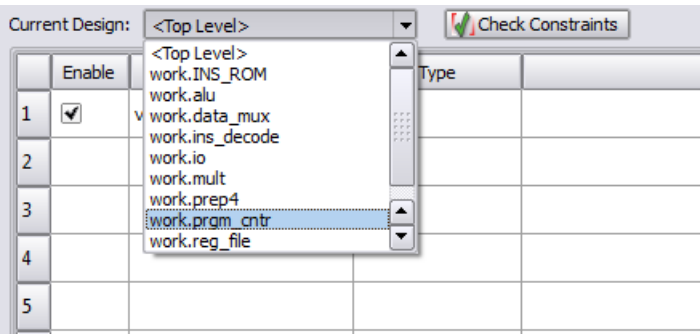
- 3. If your goal is design preservation, set the compile point you want to preserve to locked.

Setting Constraints at the Compile Point Level

You can specify constraints for each compile point in individual constraint files. (See [Compile Point Constraint Files](#), on page 456 for a description of the files.) It is recommended that you specify constraints for each locked manual compile point, but you do not need to set them for soft and hard compile points.

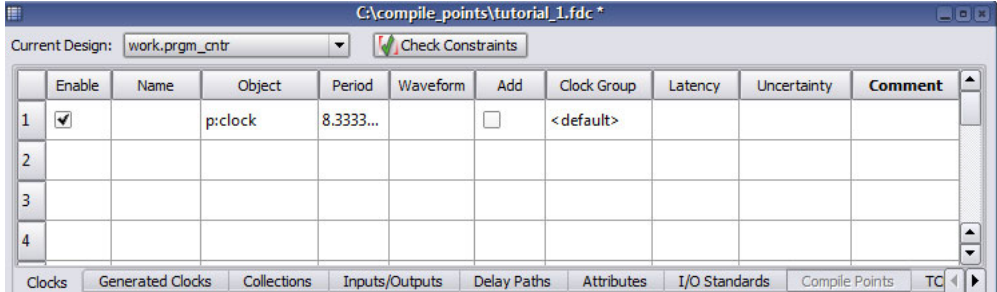
When you specify compile point constraints, the tool synthesizes the compile point using the compile point timing models instead of automatic interface timing from the top level. This procedure explains how to create a (compile point constraint file, and set constraints for the compile point:

- 1. In an open project, click the SCOPE icon (). Click on the FPGA Constraints (SCOPE) button. The New Constraints File dialog box opens.
- 2. From the Current Design field, select the module for which you want to create the compile point.



- 3. Check that you are in the right file.

A default name for the compile point file appears in the banner of the SCOPE window. Unlike the top-level constraint file, the Compile Point tab in the SCOPE UI is greyed out when the constraint file is for a compile point.



4. Set constraints for the compile point. In particular, do the following:
 - Define clocks for the compile point.
 - Specify I/O delay constraints for non-registered I/O paths that may be critical or near critical.
 - Set port constraints for the compile point that are needed for top-level mapping.

The tool uses the compile point constraints you define to synthesize the compile point. Compile point port constraints are not used at the parent level, because compile point ports do not exist at that level.

You can specify SCOPE attributes for the compile point as usual. See [Using Attributes with Compile Points, on page 477](#) for some exceptions.

5. Save the file and add it to the project. When prompted, click Yes to add the constraint file to the top-level design project.

Otherwise, use Save As to write a file such as, *moduleName.fdc* to the current directory. The hierarchical paths for compile point modules in the constraint file are specified at the compile point level; not the top-level design.

Using Attributes with Compile Points

You can use attributes as usual when you set constraints for compile points. The following sections describe some caveats and exceptions:

- `syn_hier`

When you use `syn_hier` on a compile point, the only valid value is `flatten`. All other values of this attribute are ignored for compile points. The

syn_hier attribute behaves normally for all other module boundaries that are not defined as compile points.

- syn_allowed_resources

Apply the syn_allowed_resources attribute globally or to a compile point to specify its allowed resources. When a compile point is synthesized, the resources of its siblings and parents cannot be taken into account because it stands alone as an independent synthesis unit. This attribute limits dedicated resources such as block RAMs or DSPs that the compile point can use, so that there are adequate resources available during the top-down flow.

Analyzing Compile Point Results

The software writes all timing and area results to a single log file in the implementation directory. You can check this file and the RTL and Technology views to determine if your design has met the goals for area and performance. You can also view and isolate the critical paths, search for and highlight design objects and crossprobe between the schematics and source files.

1. Check that the design meets the target frequency for the design. Use the Watch window or check the log file.
2. Open the log file and check the following:
 - Check top-level and compile point boundary timing. You can also check this visually using the RTL and Technology view schematics. If you find negative slack, check the critical path. If the critical path crosses the compile point boundary, you might need to improve the compile point constraints.
 - If the design was resynthesized, check the Summary of Compile Points section to see if compile points were preserved or remapped.

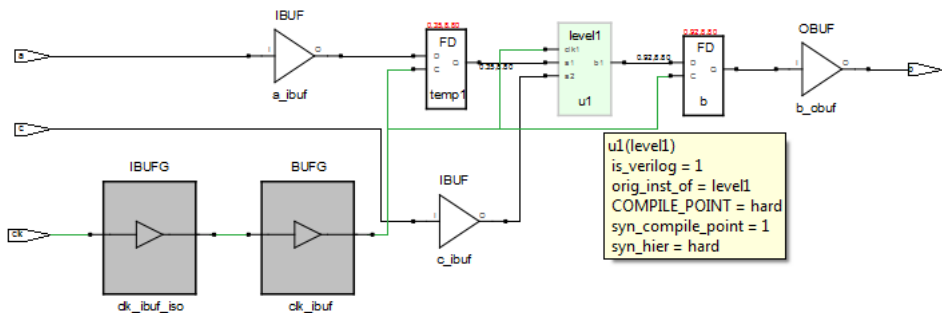
Summary of Compile Points :							

name	Status	Reason	Start Time	End Time	Realtime	CPU Time	
user_top	Remapped	Nagging options changed	Mon Mar 14 04:43:42 2011	Mon Mar 14 04:44:27 2011	0h:00m:44s	0h:00m:33s	

- Review all warnings and determine which should be addressed and which can be ignored.

- Review the area report in the log file and determine if the cell usage is acceptable for your design.
 - Check all DRC information.
3. Check other files:
- Check the individual compile point module log files. The tool creates a separate directory for each compile point module under the implementation directory. Check the compile point log file in this directory for synthesis information about the compile point synthesis run.
 - Check the compile point timing report. This report is located in the compile point results directory of the implementation directory for each compile point.
4. Check the RTL and Technology view schematics for a graphic view of the design logic. Even though instantiations of compile points do not have unique names in the output netlist, they have unique names in the Technology view. This is to facilitate timing analysis and the viewing of critical paths.

Note: Compile points of type {hard} are easily located in the Technology view with the color green.



5. Fix any errors.

Remember that the mapper reports an error if synthesis at a parent level requires that interface changes be made to a locked compile point. The software does not change the compile point interface, even if changes are required to fix DRC violations.

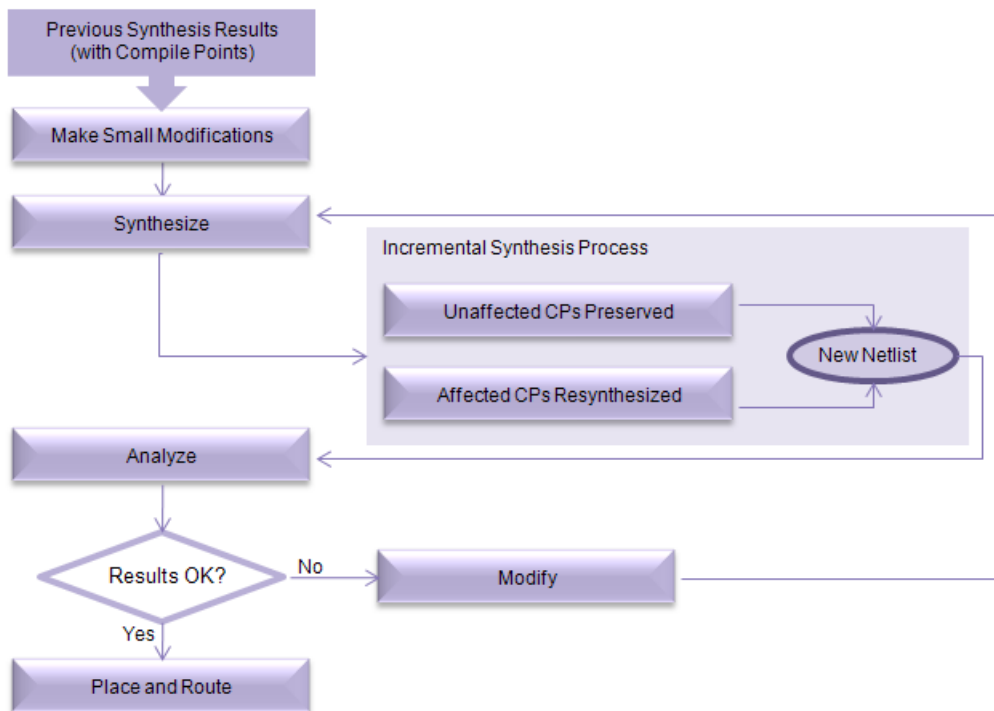
Resynthesizing Incrementally

Incremental synthesis can significantly reduce runtime on subsequent runs. It can also help with design stabilization and preservation. The following describe the incremental synthesis process, and how compile points are used in incremental synthesis within the tool and with other tools:

- [Incremental Compile Point Synthesis](#), on page 464
- [Resynthesizing Compile Points Incrementally](#), on page 480

Resynthesizing Compile Points Incrementally

The following figure illustrates how compile points (CP) are used in incremental synthesis.



1. To synthesize a design incrementally, make the changes you need to fix errors or improve your design.

- Define new compile point constraints or modify existing constraints in the existing constraint file or in a new constraint file for the compile point. Save the file.
- If necessary, reset implementation options. Click Implementation Options and modify the settings (operating conditions, optimization switches, and global frequency).

To obtain the best results, define any required constraints and set the proper implementation options for the compile point before resynthesizing.

2. Click Run to resynthesize the design.

When a design is resynthesized, compile points are not resynthesized unless source code logic, implementation options, or constraints have been modified. If there are no compile point interface changes, the software synthesizes the immediate parent using the previously generated model file for the compile point. See [Incremental Compile Point Synthesis, on page 464](#) for details.

3. Check the log file for changes.

The following figure illustrates incremental synthesis by comparing compile point summaries. After the first run, a syntax change was made in the mult module, and a logic change in the comb_logic module. The figure shows that incremental synthesis resynthesizes comb_logic (logic change), but does not resynthesize mult because the logic did not change even though there was a syntax change. Incremental synthesis re-uses the mapped file generated from the previous run to incrementally synthesize the top level.

First Run Log Summary

Summary of Compile Points		
Name	Status	Reason
mult	Mapped	No database
comb_logic	Mapped	No database
alu	Mapped	No database
eight_bit_uc	Mapped	No database
=====		

Incremental Run Log Summary

Summary of Compile Points		
Name	Status	Reason
mult	Unchanged	-
comb_logic	Remapped	Design changed
alu	Unchanged	-
eight_bit_uc	Unchanged	-
=====		

Syntax changes only; not resynthesized

Logic changes; compile point resynthesized

4. To force the software to generate a new model file for the compile point, click Implementation Options on the Device tab and enable Update Compile Point Timing Data. Click Run.

The software regenerates the model file for each compile point when it synthesizes the compile points. The new model file is used to synthesize the parent. The option remains in effect until you disable it.
5. To override incremental synthesis and force the software to resynthesize all compile points whether or not there have been changes made, use the Run->Resynthesize All command.

You might want to force resynthesis to propagate changes from a locked compile point to its environment, or resynthesize compile points one last time before tape out. When you use this option, incremental synthesis is disabled for the current run only. The Resynthesize All command does not regenerate model files for the compile points unless there are interface changes. If you enable Update Compile Point Timing Data and select Resynthesize All, you can resynthesize the entire design and regenerate the compile point model files, but synthesis will take longer than an incremental synthesis run.

CHAPTER 12

Working with IP Input

This chapter describes how to work with IP from different sources. It describes the following:

- [Generating IP with SYNCORE](#), on page 484
- [The Synopsys FPGA IP Encryption Flow](#), on page 524
- [Working with Encrypted IP](#), on page 529
- [Using Hyper Source](#), on page 543
- [Working with Lattice IP](#), on page 551

Generating IP with SYNCore

You can use the SYNCore IP wizard to generate FIFO, RAM, ROM, adder/subtractor, and counter implementations. See the following for more information.

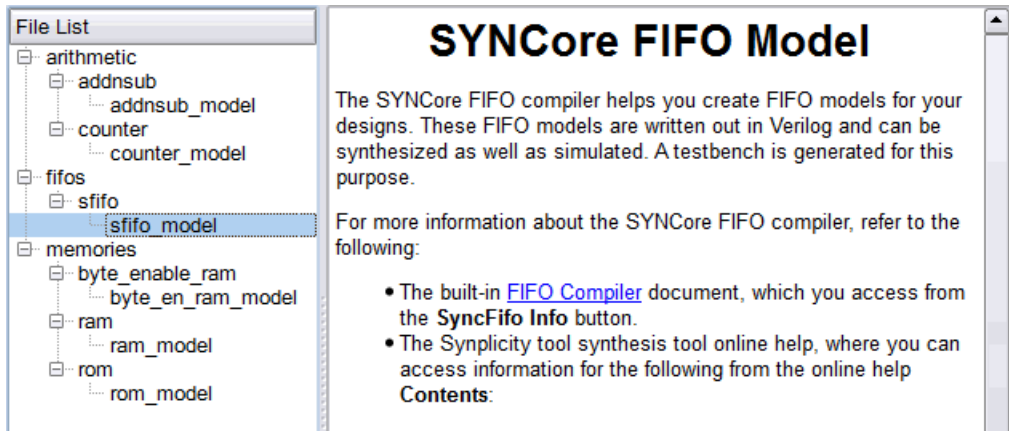
- [Specifying FIFOs with SYNCore](#), on page 484
- [Specifying RAMs with SYNCore](#), on page 490
- [Specifying Byte-Enable RAMs with SYNCore](#), on page 497
- [Specifying ROMs with SYNCore](#), on page 504
- [Specifying Adder/Subtractors with SYNCore](#), on page 509
- [Specifying Counters with SYNCore](#), on page 516

Specifying FIFOs with SYNCore

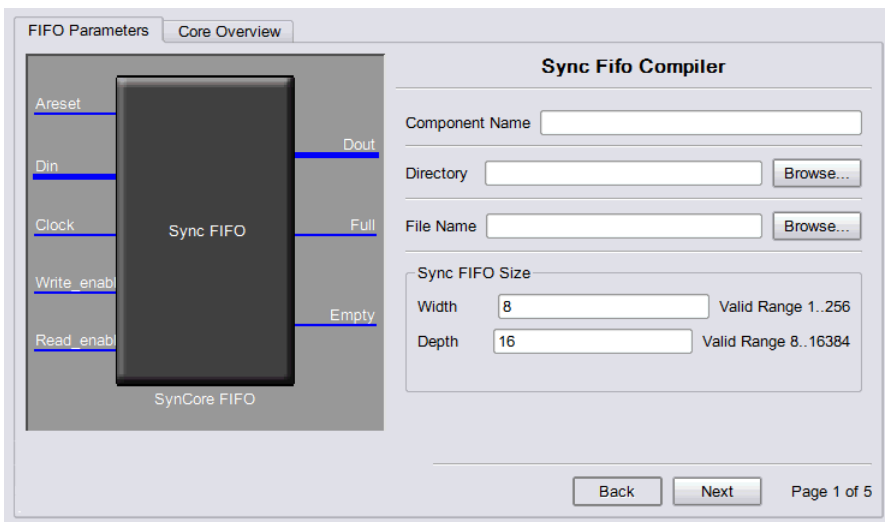
The SYNCore IP Wizard helps you generate Verilog code for your FIFO implementations. The following procedure shows you how to generate Verilog code for a FIFO using the SYNCore IP wizard.

Note: The SYNCore FIFO model uses Verilog 2001. When adding a FIFO model to a Verilog-95 design, be sure to enable the Verilog 2001 check box on the Verilog tab of the Implementation Options dialog box or include a `set_option -vlog_std v2001` statement in your project file to prevent a syntax error.

1. Start the wizard.
 - From the synthesis tool GUI, select Run->Launch SYNCore or click the Launch SYNCore icon  to start the SYNCore IP wizard.



- In the window that opens, select `sfifo_model` and click Ok. This opens the first screen of the wizard.



- Specify the parameters you need in the five pages of the wizard. For details, refer to [Specifying SYNCore FIFO Parameters, on page 488](#).

The FIFO symbol on the left reflects the parameters you set.

3. After you have specified all the parameters you need, click the **Generate** button (lower left).

The tool displays a confirmation message (TCL execution successful!) and writes the required files to the directory you specified in the parameters. The HDL code is in Verilog.

The FIFO generated is a synchronous FIFO with symmetric ports and with the same clock controlling both the read and write operations. Data is written or read on the rising edge of the clock. All resets are synchronous with the clock. All edges (clock, enable, and reset) are considered positive.

SYNCORE also generates a testbench for the FIFO that you can use for simulation. The testbench covers a limited set of vectors for testing.

You can now close the SYNCORE wizard.

4. Add the FIFO you generated to your design.
 - Use the **Add File** command to add the Verilog design file that was generated and the `syncore_sfifo.v` file to your project. These files are in the directory for output files that you specified on page 1 of the wizard.
 - Use a text editor to open the `instantiation_file.vin` template file, which is located in the same directory. Copy the lines that define the memory, and paste them into your top-level module. The following shows a template file (in red text) inserted into a top-level module.

```
module top (  
    input Clk,  
    input [15:0] DataIn,  
    input WrEn,  
    input RdEn,  
    output Full,  
    output Empty,  
    output [15:0] DataOut  
);  
  
    fifo_a32 <instanceName>(   
        .Clock(Clock)  
        , .Din (Din)  
        , .Write_enable(Write_enable)  
        , .Read_enable(Read_enable)  
        , .Dout(Dout)  
        , .Full(Full)  
        , .Empty(Empty)  
    )  
endmodule
```



template

5. Edit the template port connections so that they agree with the port definitions in your top-level module as shown in the example below (the updated connection names are shown in red). You can also assign a unique name to each instantiation.

```
module top (  
    input Clk,  
    input [15:0] DataIn,  
    input WrEn,  
    input RdEn,  
  
    output Full,  
    output Empty,  
    output [15:0] DataOut  
);  
  
    fifo_a32 busfifo(  
        .Clock(Clk)  
        , .Din(DataIn)  
        , .Write_enable(WrEn)  
        , .Read_enable(RdEn)  
        , .Dout(DataOut)  
        , .Full(Full)  
        , .Empty(Empty)  
    )  
endmodule
```

Note that currently, the FIFO models will not be implemented with the dedicated FIFO blocks available in certain technologies.

Specifying SYNCORE FIFO Parameters

The following elaborates on the parameter settings for SYNCORE FIFOs. The status, handshaking, and programmable flags are optional. For descriptions of the parameters, see [SYNCore FIFO Wizard, on page 403](#) in the *Reference Manual*. For timing diagrams, see [SYNCore FIFO Compiler, on page 218](#) in the *Reference Manual*.

1. Start the SYNCORE wizard, as described in [Specifying FIFOs with SYNCORE, on page 484](#).
2. Do the following on page 1 of the FIFO wizard:
 - In Component Name, specify a name for the FIFO. Do not use spaces.
 - In Directory, specify a directory where you want the output files to be written. Do not use spaces.

- In Filename, specify a name for the Verilog output file with the FIFO specifications. Do not use spaces.
 - Click Next. The wizard opens another page where you can set parameters.
3. For a FIFO with no status, handshaking, or programmable flags, use the default settings. You can generate the FIFO, as described in [Specifying FIFOs with SYNCORE, on page 484](#).
 4. To set an almost full status flag, do the following on page 2 of the FIFO wizard:
 - Enable Almost Full.
 - Set associated handshaking flags for the signal as desired, with the Overflow Flag and Write Acknowledge options.
 - Click Next when you are done.
 5. To set an almost empty status flag, do the following on page 3:
 - Enable Almost Empty.
 - Set associated handshaking flags for the signal as desired, with the Underflow Flag and Read Acknowledge options.
 - Click Next when you are done.
 6. To set a programmable full flag, do the following:
 - Make sure you have enabled Full on page 2 of the wizard and set any handshaking flags you require.
 - Go to page 4 and enable Programmable Full.
 - Select one of the four mutually exclusive configurations for Programmable Full on page 4. See [Programmable Full, on page 229](#) in the *Reference Manual* for details.
 - Click Next when you are done.
 7. To set a programmable empty flag, do the following:
 - Make sure you have enabled Empty on page 3 of the wizard and set any handshaking flags you require.
 - Go to page 5 and enable Programmable Empty.
 - Select one of the four mutually exclusive configurations for Programmable Empty on page 5. See [Programmable Empty, on page 232](#) in the *Reference Manual* for details.

You can now generate the FIFO and add it to the design, as described in [Specifying FIFOs with SYNCore, on page 484](#).

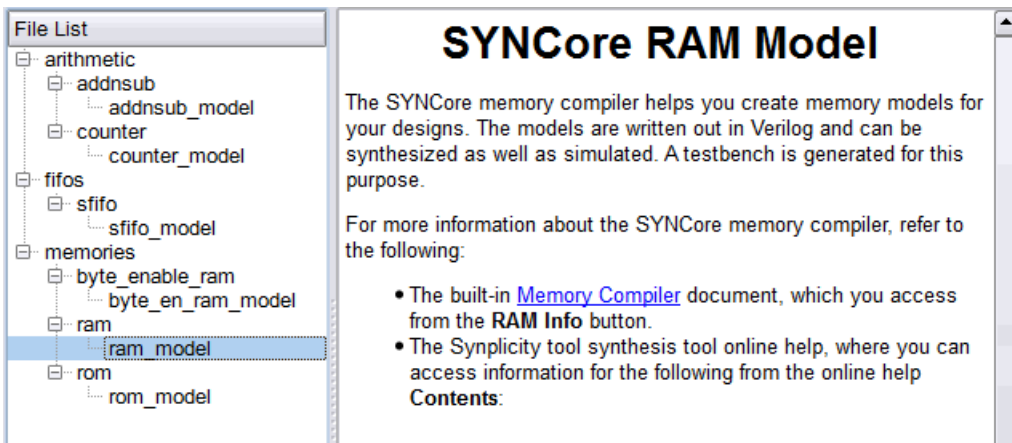
Specifying RAMs with SYNCore

The SYNCore IP wizard helps you generate Verilog code for your RAM implementation requirements. The following procedure shows you how to generate Verilog code for a RAM using the SYNCore IP wizard.

Note: The SYNCore RAM model uses Verilog 2001. When adding a RAM model to a Verilog-95 design, be sure to enable the Verilog 2001 check box on the Verilog tab of the Implementation Options dialog box or include a `set_option -vlog_std v2001` statement in your project file to prevent a syntax error.

1. Start the wizard.

- From the synthesis tool GUI, select Run->Launch SYNCore or click the Launch SYNCore icon  to start the SYNCore IP wizard.



- In the window that opens, select `ram_model` and click Ok. This opens the first screen of the wizard.

2. Specify the parameters you need in the wizard.

- For details about the parameters for a single-port RAM, see [Specifying Parameters for Single-Port RAM, on page 493](#).

- For details about the parameters for a dual-port RAM, see [Specifying Parameters for Dual-Port RAM, on page 494](#). Note that dual-port implementations are only supported for some technologies.

The RAM symbol on the left reflects the parameters you set.

The default settings for the tool implement a block RAM with synchronous resets, and where all edges (clock, enable, and reset) are considered positive.

3. After you have specified all the parameters you need, click the Generate button in the lower left corner.

The tool displays a confirmation message is displayed (TCL execution successful!) and writes the required files to the directory you specified in the parameters. The HDL code is in Verilog.

SYNCore also generates a testbench for the RAM. The testbench covers a limited set of vectors.

You can now close the SYNCore Memory Compiler.

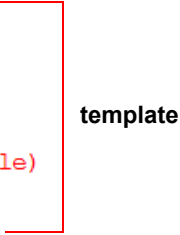
4. Edit the RAM files if necessary.

- The default RAM has a `no_rw_check` attribute enabled. If you do not want this, edit `syncore_ram.v` and comment out the ``define SYN_MULTI_PORT_RAM` statement, or use ``undef SYN_MULTI_PORT_RAM`.
- If you want to use the synchronous RAMs available in the target technology, make sure to register either the read address or the outputs.

5. Add the RAM you generated to your design.

- Use the Add File command to add the Verilog design file that was generated and the `syncore_ram.v` file to your project. These files are in the directory for output files that you specified on page 1 of the wizard.
- Use a text editor to open the `instantiation_file.vin` template file, which is located in the same directory. Copy the lines that define the memory, and paste them into your top-level module. The following figure shows a template file (in red text) inserted into a top-level module.

```
module top (  
  
    input ClkA,  
    input [7:0] AddrA,  
    input [15:0] DataInA,  
    input WrEnA,  
  
    output [15:0] DataOutA  
  
);  
  
myram2 <InstanceName> (  
    .PortAClk (PortAClk)  
    , .PortAAddr (PortAAddr)  
    , .PortADDataIn (PortADDataIn)  
    , .PortAWriteEnable (PortAWriteEnable)  
    , .PortADDataOut (PortADDataOut)  
);  
  
endmodule
```



6. Edit the template port connections so that they agree with the port definitions in your top-level module as shown in the example below (the updated connection names are shown in red). You can also assign a unique name to each instantiation.

```
module top (  
  
    input ClkA,  
    input [7:0] AddrA,  
    input [15:0] DataInA,  
    input WrEnA,  
  
    output [15:0] DataOutA  
  
);  
  
myram2 decoderram(  
    .PortAClk(ClkA)  
    , .PortAAddr(AddrA)  
    , .PortADataIn(DataInA)  
    , .PortAWriteEnable(WrEnA)  
    , .PortADataOut(DataOutA)  
);  
  
endmodule
```

Specifying Parameters for Single-Port RAM

To create a single-port RAM with the SYNCORE Memory Compiler, you need to specify a single read/write address (single port) and a single clock. You only need to configure Port A. The following procedure lists what you need to specify. For descriptions of each parameter, refer to [SYNCORE RAM Wizard, on page 412](#) in the *Reference Manual*.

1. Start the SYNCORE RAM wizard, as described in [Specifying RAMs with SYNCORE, on page 490](#).
2. Do the following on page 1 of the RAM wizard:
 - In Component Name, specify a name for the memory. Do not use spaces.
 - In Directory, specify a directory where you want the output files to be written. Do not use spaces.

- In Filename, specify a name for the Verilog file that will be generated with the RAM specifications. Do not use spaces.
- Enter data and address widths.
- Enable Single Port, to specify that you want to generate a single-port RAM. This automatically enables Single Clock.
- Click Next. The wizard opens another page where you can set parameters for Port A.

The RAM symbol dynamically updates to reflect the parameters you set.

3. Do the following on page 2 of the RAM wizard:

- Set Use Write Enable to the setting you want.
- Set Register Read Address to the setting you want.
- Set Synchronous Reset to the setting you want. Register Outputs is always enabled
- Specify the read access you require for the RAM.

You can now generate the RAM by clicking Generate, as described in [Specifying RAMs with SYNCORE, on page 490](#). You do not need to specify any parameters on page 3, as this is a single-port RAM and you do not need to specify Port B. All output files are in the directory you specified on the first page of the wizard.

For details about setting dual-port RAM parameters, see [Specifying Parameters for Dual-Port RAM, on page 494](#). For read/write timing diagrams, see [Read/Write Timing Sequences, on page 243](#) of the *Reference Manual*.

Specifying Parameters for Dual-Port RAM

The following procedure shows you how to set parameters for dual-port memory in the SYNCORE wizard. Dual-port RAMs are only supported for some technologies. For information about generating single-port RAMs, see [Specifying Parameters for Single-Port RAM, on page 493](#). It shows you how to generate these common RAM configurations:

- One read access and one write access
- Two read accesses and one write access
- Two read accesses and two write accesses

For the corresponding read/write timing diagrams, see [Read/Write Timing Sequences](#), on page 243 of the *Reference Manual*.

1. Start the SYNCORE RAM wizard, as described in [Generating IP with SYNCORE](#), on page 484.
2. Do the following on page 1 of the RAM wizard:
 - In Component Name, specify a name for the memory. Do not use spaces.
 - In Directory, specify a directory where you want the output files to be written. Do not use spaces.
 - In Filename, specify a name for the Verilog file that will be generated with the RAM specifications. Do not use spaces.
 - Enter data and address widths.
 - Enable Dual Port, to specify that you want to generate a dual-port RAM.
 - Specify the clocks.

For a single clock...	Enable Single Clock.

For separate clocks for each of the ports...	Enable Separate Clocks For Each Port.

- Click Next. The wizard opens another page where you can set parameters for Port A.
3. Do the following on page 2 of the RAM wizard to specify settings for Port A:

- Set parameters according to the kind of memory you want to generate:

One read & one write	Enable Read Only Access.

Two reads & one write	Enable Read and Write Access. Specify a setting for Use Write Enable.

Two reads & two writes	Enable Read and Write Access. Specify a setting for Use Write Enable. Specify a read access option for Port A.

- Specify a setting for Register Read Address.

- Set Synchronous Reset to the setting you want. Register Outputs is always enabled.
 - Click Next. The wizard opens another page where you can set parameters for Port B. The page and the parameters are identical to the previous page, except that the settings are for Port B instead of Port A.
4. Specify the settings for Port B on page 3 of the wizard according to the kind of memory you want to generate:

One read & one write	Enable Write Only Access. Set Use Write Enable to the setting you want.
Two reads & one write	Enable Read Only Access. Specify a setting for Register Read Address.
Two reads & two writes	Enable Read and Write Access. Specify a setting for Use Write Enable. Specify a setting for Register Read Address. Set Synchronous Reset to the setting you want. Note that Register Outputs is always enabled. Select a read access option for Port B.

The RAM symbol on the left reflects the parameters you set. All output files are written to the directory you specified on the first page of the wizard.

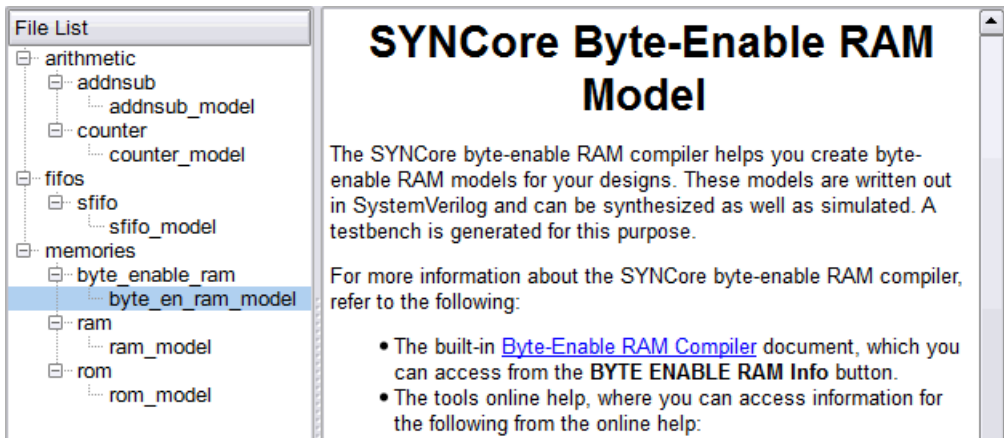
You can now generate the RAM by clicking Generate, as described in [Generating IP with SYNCORE, on page 484](#), and add it to your design.

Specifying Byte-Enable RAMs with SYNCORE

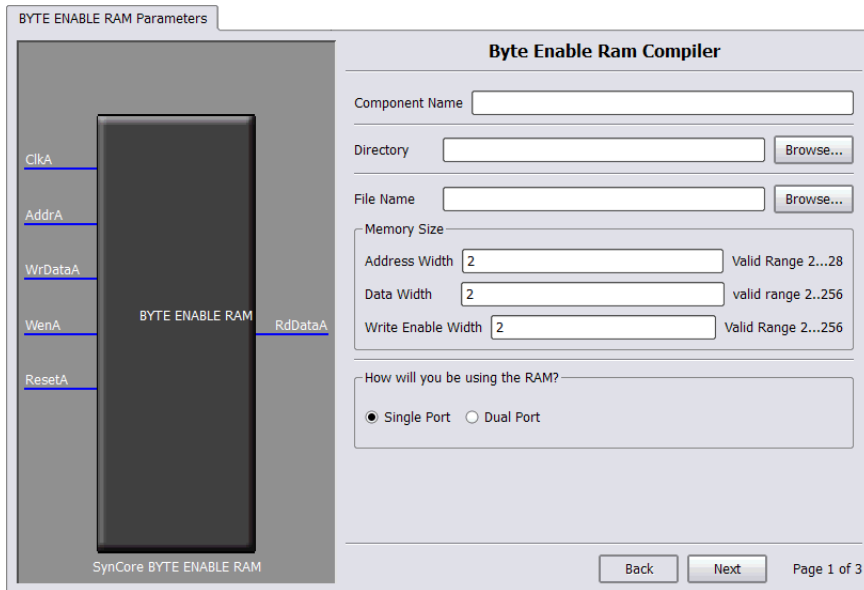
The SYNCORE IP wizard helps you generate SystemVerilog code for your byte-enable RAM implementation requirements. The following procedure shows you how to generate SystemVerilog code for a byte-enable RAM using the SYNCORE IP wizard.

Note: The SYNCORE byte-enable RAM model uses SystemVerilog. When adding a byte-enable RAM to your design, be sure to enable the System Verilog check box on the Verilog tab of the Implementation Options dialog box or include a `set_option -vlog_std sysv` statement in your project file to prevent a syntax error.

1. Start the wizard.
 - From the FPGA synthesis tool GUI, select Run->Launch SYNCORE or click the Launch SYNCORE icon  to start the SYNCORE IP wizard.



- In the window that opens, select `byte_en_ram_model` and click Ok to open the first page (page1) of the wizard.



2. Specify the parameters you need in the wizard. For details about the parameters, see [Specifying Byte-Enable RAM Parameters, on page 501](#). The BYTE ENABLE RAM symbol on the left reflects any parameters you set.

3. After you have specified all the parameters you need, click the Generate button in the lower left corner. The tool displays a confirmation message (TCL execution successful!) and writes the required files to the directory you specified on page 1 of the wizard. The HDL code is in SystemVerilog.

SYNCore also generates a test bench for the byte-enable RAM component. The test bench covers a limited set of vectors. You can now close the SYNCore byte-enable RAM compiler.

4. Edit the generated files for the byte-enable RAM component if necessary.

5. Add the byte-enable RAM that you generated to your design.

- On the Verilog tab of the Implementation Options dialog box, make sure that SystemVerilog is enabled.
- Use the Add File command to add the Verilog design file that was generated (the filename entered on page 1 of the wizard) and the

syncore_*.v file to your project. These files are in the directory for output files that you specified on page 1 of the wizard.

- Use a text editor to open the instantiation_file.vin template file. This file is located in the same output files directory. Copy the lines that define the byte-enable RAM and paste them into your top-level module.
- Edit the template port connections so that they agree with the port definitions in the top-level module; also change the instantiation name to agree with the component name entered on page 1. The following figure shows a template file inserted into a top-level module with the updated component name and port connections in red.

```

module top
    (input ClockA,
     input [3:0] AddA
     input [31:0] DataIn
     input WrEnA,
     input Reset
     output [31:0] DataOut
    )|

INST_TAG

SP_RAM #
    (.ADD_WIDTH(4),
     .WE_WIDTH(2),
     .RADDR_LTNCY_A(1), // 0 - No Latency, 1 - 1 Cycle Latency
     .RDATA_LTNCY_A(1), // 0 - No Latency, 1 - 1 Cycle Latency
     .RST_TYPE_A(1), // 0 - No Reset, 1 synchronous
     .RST_RDATA_A({32{1'b1}}),
     .DATA_WIDTH(32)
    )

4x32spram
    (// Output Ports
     .RdDataA(DataIn),
     // Input Ports
     .WrDataA(DataOut),
     .WenA(WeEnA),
     .AddrA(AddA),
     .ResetA(Reset),
     .ClkA(ClockA)
    );

```

Port List

Port A interface signals are applicable for both single-port and dual-port configurations; Port B signals are applicable for dual-port configuration only.

Name	Type	Description
ClkA	Input	Clock input for Port A
WenA	Input	Write enable for Port A; present when Port A is in write mode
AddrA	Input	Memory access address for Port A
ResetA	Input	Reset for memory and all registers in core; present with registered read data when Reset is enabled; active low (cannot be changed)
WrDataA	Input	Write data to memory for Port A; present when Port A is in write mode
RdDataA	Output	Read data output for Port A; present when Port A is in read or read/write mode
ClkB	Input	Clock input for Port B; present in dual-port mode
WenB	Input	Write enable for Port B; present in dual-port mode when Port B is in write mode
AddrB	Input	Memory access address for Port B; present in dual-port mode
ResetB	Input	Reset for memory and all registers in core; present in dual-port mode when read data is registered and Reset is enabled; active low (cannot be changed)
WrDataB	Input	Write data to memory for Port B; present in dual-port mode when Port B is in write mode
RdDataB	Output	Read data output for Port B; present in dual-port mode when Port B is in read or read/write mode

Specifying Byte-Enable RAM Parameters

When creating a single-port, byte-enable RAM with the SYNCORE IP wizard, you must specify a single read address and a single clock; you only need to configure the Port A parameters on page 2 of the wizard.

When creating a dual-port, byte-enable RAM, you must additionally configure the Port B parameters on page 3 of the wizard.

The following procedure lists the parameters you need to specify. For descriptions of each parameter, refer to [Parameter List, on page 250](#) in the Reference Manual.

1. Start the SYNCORE byte-enable RAM wizard as described in [Specifying Byte-Enable RAMs with SYNCORE, on page 497](#).
2. Do the following on page 1 of the byte-enable RAM wizard:
 - Specify a name for the memory in the Component Name field; do not use spaces.
 - Specify a directory name in the Directory field where you want the output files to be written; do not use spaces.
 - Specify a name in the File Name field for the SystemVerilog file to be generated with the byte-enable RAM specifications; do not use spaces.
 - Enter a value for the address width of the byte-enable RAM; the maximum depth of memory is limited to 2^{256} .
 - Enter a value for the data width for the byte-enable RAM; data width values range from 2 to 256.
 - Enter a value for the write enable width; write-enable width values range from 1 to 4.
 - Select Single Port to generate a single-port, byte-enable RAM or select Dual Port to generate a dual-port, byte-enable RAM.
 - Click Next to open page 2 of the wizard.

The Byte Enable RAM symbol dynamically updates to reflect the parameters that you set.

3. Do the following on page 2 (configuring Port A) of the wizard:
 - Select the Port A configuration. Only Read and Write Access mode is valid for single-port configurations; this mode is selected by default.
 - Set Pipelining Address Bus and Output Data according to your application. By default, read data is registered; you can register both the address and data registers.
 - Set the Configure Reset Options. Enabling the checkbox enables the synchronous reset for read data. This option is enabled only when the read data is registered. Reset is active low and cannot be changed.
 - Configure output reset data value options under Specify output data on reset; reset data can be set to default value of all '1' s or to a user-defined decimal value. Reset data value options are disabled when the reset is not enabled for Port A.
 - Set Write Enable for Port A value; default for the write-enable level is active high.
4. If you are generating a dual-port, byte-enable RAM, set the Port B parameters on page 3 (note that the Port B parameters are only enabled when Dual Port is selected on page 1).

The Port B parameters are identical to the Port A parameters on page 2. When using the dual-port configuration, when one port is configured for read access, the other port can only be configured for read/write access or write access.

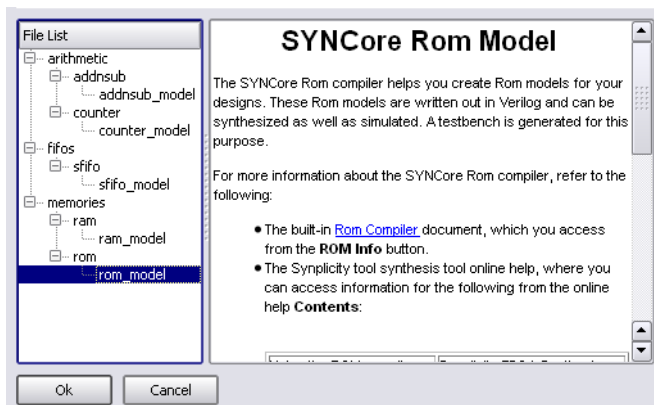
5. Generate the byte-enable RAM by clicking Generate. Add the file to your project and edit the template file as described in [Specifying Byte-Enable RAMs with SYNCORE, on page 497](#). For read/write timing diagrams, see [Read/Write Timing Sequences, on page 247](#) of the reference manual.

Specifying ROMs with SYNCore

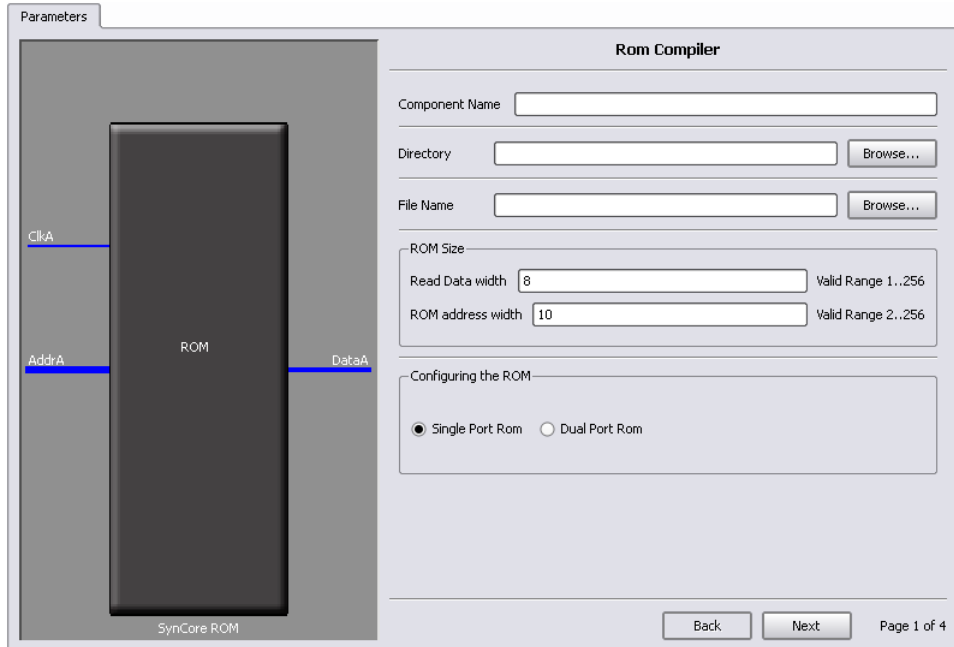
The SYNCore IP wizard helps you generate Verilog code for your ROM implementation requirements. The following procedure shows you how to generate Verilog code for a ROM using the SYNCore IP wizard.

Note: The SYNCore ROM model uses Verilog 2001. When adding a ROM model to a Verilog-95 design, be sure to enable the Verilog 2001 check box on the Verilog tab of the Implementation Options dialog box or include a `set_option -vlog_std v2001` statement in your project file to prevent a syntax error.

1. Start the wizard.
 - From the FPGA synthesis tool GUI, select Run->Launch SYNCore or click the Launch SYNCore icon  to start the SYNCore IP wizard.



- In the window that opens, select `rom_model` and click Ok to open page 1 of the wizard.



2. Specify the parameters you need in the wizard. For details about the parameters, see [Specifying ROM Parameters, on page 508](#). The ROM symbol on the left reflects any parameters you set.
3. After you have specified all the parameters you need, click the **Generate** button in the lower left corner. The tool displays a confirmation message (TCL execution successful!) and writes the required files to the directory you specified on page 1 of the wizard. The HDL code is in Verilog.

SYNCORE also generates a testbench for the ROM. The testbench covers a limited set of vectors.

You can now close the SYNCORE ROM Compiler.

4. Edit the ROM files if necessary. If you want to use the synchronous ROMs available in the target technology, make sure to register either the read address or the outputs.
5. Add the ROM you generated to your design.
 - Use the **Add File** command to add the Verilog design file that was generated and the `syncore_rom.v` file to your project. These files are in

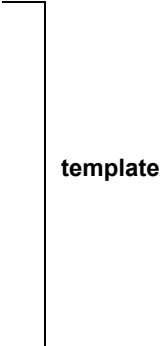
the directory for output files that you specified on page 1 of the wizard.

- Use a text editor to open the `instantiation_file.vin` template file. This file is located in the same output files directory. Copy the lines that define the ROM, and paste them into your top-level module. The following figure shows a template file (in red text) inserted into a top-level module.

```
module test_rom_style(z,a,clk,en,rst);
input clk,en,rst;
output reg [3:0] z;
input [6:0] a;

my1stROM    <InstanceName> (
    // Output Ports
    .DataA(DataA),

    // Input Ports
    .ClkA(ClkA),
    .EnA(EnA),
    .ResetA(ResetA),
    .AddrA(AddrA)
);
```



6. Edit the template port connections so that they agree with the port definitions in the top-level module as shown in the example below. You can also assign a unique name to each instantiation.

```
module test_rom_style(z,a,clk,en,rst);
input clk,en,rst;
output reg [3:0] z;
input [6:0] a;

my1stROM decode_rom(
    // Output Ports
    .DataA(z),

    // Input Ports
    .ClkA(clk),
    .EnA(en),
    .ResetA(rst),
    .AddrA(a)
);
```

Port List

PortA interface signals are applicable for both single-port and dual-port configurations; PortB signals are applicable for dual-port configuration only.

Name	Type	Description
ClkA	Input	Clock input for Port A
EnA	Input	Enable input for Port A
AddrA	Input	Read address for Port A
ResetA	Input	Reset or interface disable pin for Port A
DataA	Output	Read data output for Port A
ClkB	Input	Clock input for Port B
EnB	Input	Enable input for Port B
AddrB	Input	Read address for Port B
ResetB	Input	Reset or interface disable pin for Port B
DataB	Output	Read data output for Port B

Specifying ROM Parameters

If you are creating a single-port ROM with the SYNCORE IP wizard, you need to specify a single read address and a single clock, and you only need to configure the Port A parameters on page 2. If you are creating a dual-port ROM, you must additionally configure the Port B parameters on page 3. The following procedure lists what you need to specify. For descriptions of each parameter, refer to [SYNCORE RAM Wizard, on page 412](#) in the *Reference Manual*.

1. Start the SYNCORE ROM wizard, as described in [Specifying ROMs with SYNCORE, on page 504](#).
2. Do the following on page 1 of the ROM wizard:
 - In Component Name, specify a name for the memory. Do not use spaces.
 - In Directory, specify a directory where you want the output files to be written. Do not use spaces.
 - In Filename, specify a name for the Verilog file that will be generated with the ROM specifications. Do not use spaces.
 - Enter values for Read Data width and ROM address width (minimum depth value is 2; maximum depth of the memory is limited to 2^{256}).
 - Select Single Port Rom to indicate that you want to generate a single-port ROM or select Dual Port Rom to generate a dual-port ROM.
 - Click Next. The wizard opens page 2 where you set parameters for Port A.

The ROM symbol dynamically updates to reflect any parameters you set.

3. Do the following on page 2 (Configuring Port A) of the RAM wizard:
 - For synchronous ROMs, select Register address bus AddrA and/or Register output data bus DataA to register the read address and/or the outputs. Selecting either checkbox enables the Enable for Port A checkbox which is used to select the Enable level.
 - Set the Configure Reset Options. Enabling the checkbox enables the type of reset (asynchronous or synchronous) and allows an output data pattern (all 1's or a specified pattern) to be defined on page 4.
4. If you are generating a dual-port ROM, set the port B parameters on page 3 (the page 3 parameters are only enabled when Dual Port Rom is selected on page 1).

5. On page 4, specify the location of the ROM initialization file and the data format (Hexadecimal or Binary). ROM initialization is supported using memory-coefficient files. The data format is either binary or hexadecimal with each data entry on a new line in the memory-coefficient file (specified by parameter INIT_FILE). Supported file types are `txt`, `mem`, `dat`, and `init` (recommended).
6. Generate the ROM by clicking Generate, as described in [Specifying ROMs with SYNCORE, on page 504](#) and add it to your design. All output files are in the directory you specified on page 1 of the wizard.

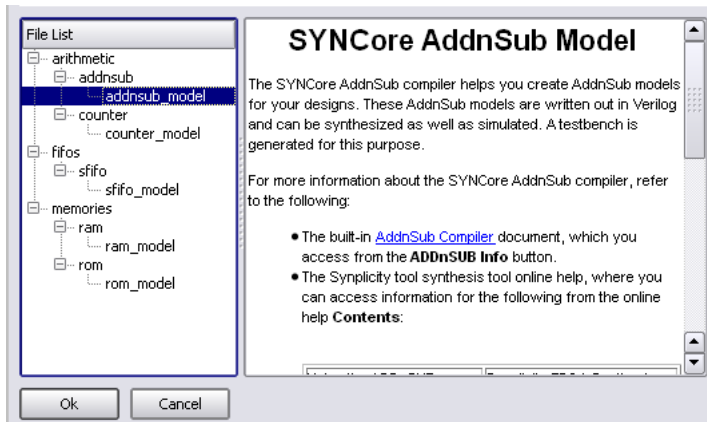
For read/write timing diagrams, see [Read/Write Timing Sequences, on page 243](#) of the *Reference Manual*.

Specifying Adder/Subtractors with SYNCORE

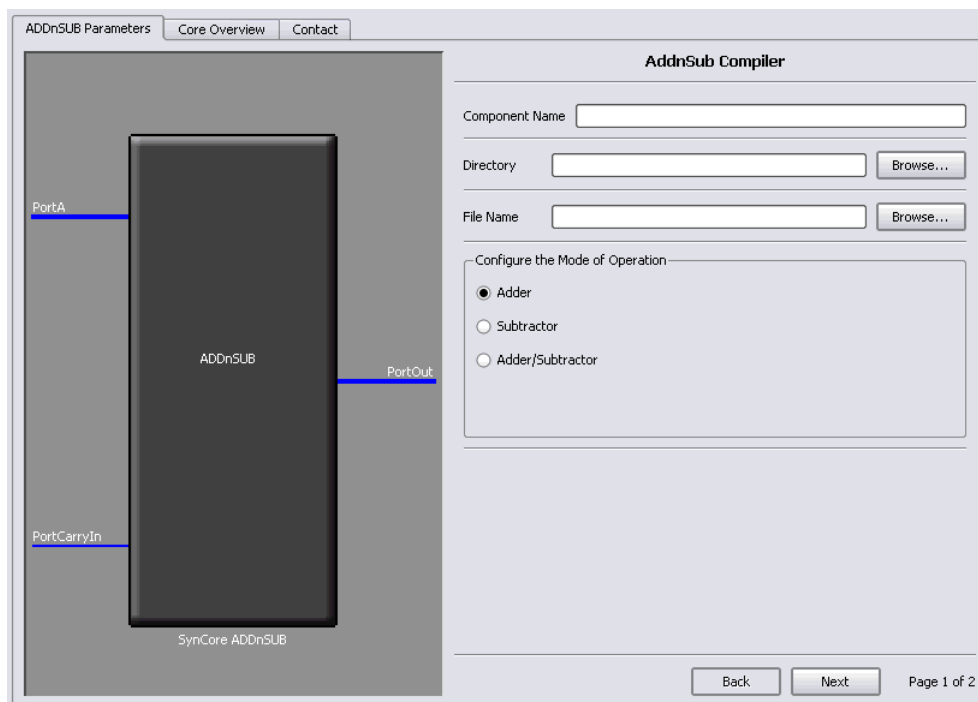
The SYNCORE IP wizard helps you generate Verilog code for your adder/subtractor implementation requirements. The following procedure shows you how to generate Verilog code for an adder/subtractor using the SYNCORE IP wizard.

Note: The SYNCORE adder/subtractor models use Verilog 2001. When adding an adder/subtractor model to a Verilog-95 design, be sure to enable the Verilog 2001 check box on the Verilog tab of the Implementation Options dialog box or include a `set_option -vlog_std v2001` statement in your project file to prevent a syntax error.

1. Start the wizard.
 - From the FPGA synthesis tool GUI, select Run->Launch SYNCORE or click the Launch SYNCORE icon  to start the SYNCORE IP wizard.



- In the window that opens, select `addnsub_model` and click Ok to open page1 of the wizard.



2. Specify the parameters you need in the wizard. For details about the parameters, see [Specifying Adder/Subtractor Parameters, on page 515](#). The ADDnSUB symbol on the left reflects any parameters you set.
3. After you have specified all the parameters you need, click the Generate button in the lower left corner.

The tool displays a confirmation message (TCL execution successful!) and writes the required files to the directory you specified on page 1 of the wizard. The HDL code is in Verilog.

The SYNCORE wizard also generates a testbench for your adder/subtractor. The testbench covers a limited set of vectors. You can now close the wizard.

4. Add the adder/subtractor you generated to your design.
 - Edit the adder/subtractor files if necessary.
 - Use the Add File command to add the Verilog design file that was generated and the `syncore_addnsub.v` file to your project. These files are in the directory for output files that you specified on page 1 of the wizard.
 - Use a text editor to open the `instantiation_file.v` template file. This file is located in the same output files directory. Copy the lines that define the adder/subtractor and paste them into your top-level module. The following figure shows a template file (in red text) inserted into a top-level module.

```

module top (
    output [15 : 0] Out,
    input Clk,
    input [15 : 0] A,
    input CEA,
    input RSTA,
    input [15 : 0] B,
    input CEB,
    input RSTB,
    input CEOut,
    input RSTOut,
    input ADDnSUB,
    input CarryIn );

    My_ADDnSUB <InstanceName> (
        // Output Ports
        .PortOut(PortOut),
        // Input Ports
        .PortClk(PortClk),
        .PortA(PortA),
        .PortCEA(PortCEA),
        .PortRSTA(PortRSTA),
        .PortB(PortB),
        .PortCEB(PortCEB),
        .PortRSTB(PortRSTB),
        .PortCEOut(PortCEOut),
        .PortRSTOut(PortRSTOut),
        .PortADDnSUB(PortADDnSUB),
        .PortCarryIn(PortCarryIn) );

endmodule

```

template

5. Edit the template port connections so that they agree with the port definitions in your top-level module as shown in the example below (the updated connection names are shown in red). You can also assign a unique name to each instantiation.

```

module top (
    output [15 : 0] Out,
    input Clk,
    input [15 : 0] A,
    input CEA,
    input RSTA,

```



```
        input [15 : 0] B,
        input CEB,
        input RSTB,
        input CEOut,
        input RSTOut,
        input ADDnSUB,
        input CarryIn);

My_ADDnSUB ADDnSUB_inst (
// Output Ports
    .PortOut (Out),
// Input Ports
    .PortClk (Clk),
    .PortA (A),
    .PortCEA (CEA),
    .PortRSTA (RSTA),
    .PortB (B),
    .PortCEB (CEB),
    .PortRSTB (RSTB),
    .PortCEOut (CEOut),
    .PortRSTOut (RSTOut),
    .PortADDnSUB (ADDnSUB),
    .PortCarryIn (CarryIn));
endmodule
```

Port List

The following table lists the port assignments for all possible configurations; the third column specifies the conditions under which the port is available.

Port Name	Description	Required/Optional
PortA	Data input for adder/subtractor Parameterized width and pipeline stages	Always present
PortB	Data input for adder/subtractor Parameterized width and pipeline stages	Not present if adder/subtractor is configured as a constant adder/subtractor
PortClk	Primary clock input; clocks all registers in the unit	Always present
PortRstA	Reset input for port A pipeline registers (active high)	Not present if pipeline stage for port A is 0
PortRstB	Reset input for port B pipeline registers (active high)	Not present if pipeline stage for port B is 0 or for constant adder/subtractor
PortADDnSUB	Selection port for dynamic operation	Not present if adder/subtractor configured as standalone adder or subtractor
PortRstOut	Reset input for output register (active high)	Not present if output pipeline stage is 0
PortCEA	Clock enable for port A pipeline registers (active high)	Not present if pipeline stage for port A is 0
PortCEB	Clock enable for port B pipeline registers (active high)	Not present if pipeline stage for port B is 0 or for constant adder/subtractor
PortCarryIn	Carry input for adder/subtractor	Always present
PortCEOut	Clock enable for output register (active high)	Not present if output pipeline stage is 0
PortOut	Data output	Always present

Specifying Adder/Subtractor Parameters

The SYNCORE adder/subtractor can be configured as any of the following:

- Adder
- Subtractor
- Dynamic Adder/Subtractor

If you are creating a constant input adder, subtractor, or a dynamic adder/subtractor with the SYNCORE IP wizard, you must select Constant Value Input and specify a value for port B in the Constant Value/Port B Width field on page 2 of the parameters. The following procedure lists the parameters you need to define when generating an adder/subtractor. For descriptions of each parameter, see [SYNCORE Adder/Subtractor Wizard, on page 423](#) in the *Reference Manual*.

1. Start the SYNCORE adder/subtractor wizard as described in [Specifying Adder/Subtractors with SYNCORE, on page 509](#).
2. Enter the following on page 1 of the wizard:
 - In the Component Name field, specify a name for your adder/subtractor. Do not use spaces.
 - In the Directory field, specify a directory where you want the output files to be written. Do not use spaces.
 - In the Filename field, specify a name for the Verilog file that will be generated with the adder/subtractor definitions. Do not use spaces.
 - Select the appropriate configuration in Configure the Mode of Operation.
3. Click Next. The wizard opens page 2 where you set parameters for port A and port B.
4. Configure Port A and B.
 - In the Configure Port A section, enter a value in the Port A Width field.
 - If you are defining a synchronous adder/subtractor, check Register Input A and then check Clock Enable for Register A and/or Reset for Register A.
 - To configure port B as a constant port, go to the Configure Port B section and check Constant Value Input. Enter the constant value in the Constant Value/Port B Width field.

- To configure port B as a dynamic port, go to the Configure Port B section and check Enable Port B and enter the port width in the Constant Value/Port B Width field.
 - To define a synchronous adder/subtractor, check Register Input B and then check Clock Enable for Register B and/or Reset for Register B.
5. In the Configure Output Port section:
 - Enter a value in the Output port Width field.
 - If you are registering the output port, check Register output Port.
 - If you are defining a synchronous adder/subtractor check Clock Enable for Register PortOut and/or Reset for Register PortOut.
 6. In the Configure Reset type for all Reset Signal section, click Synchronous Reset or Asynchronous Reset as appropriate.

As you enter the page 2 parameters, the ADDnSUB symbol dynamically updates to reflect the parameters you set.

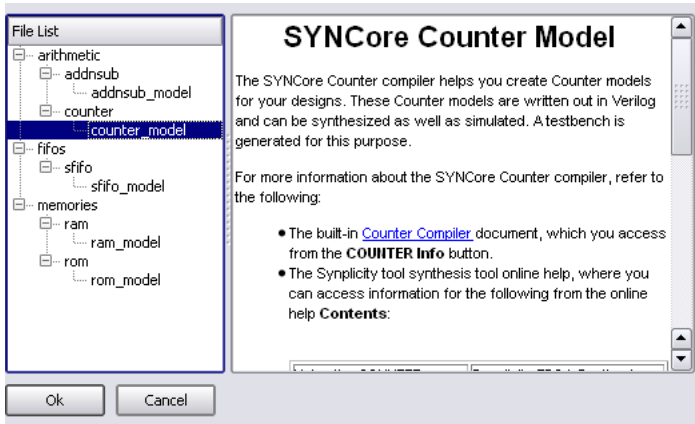
7. Generate the adder/subtractor by clicking the Generate button as described in [Specifying Adder/Subtractors with SYNCORE, on page 509](#) and add it to your design. All output files are in the directory you specified on page 1 of the wizard.

Specifying Counters with SYNCORE

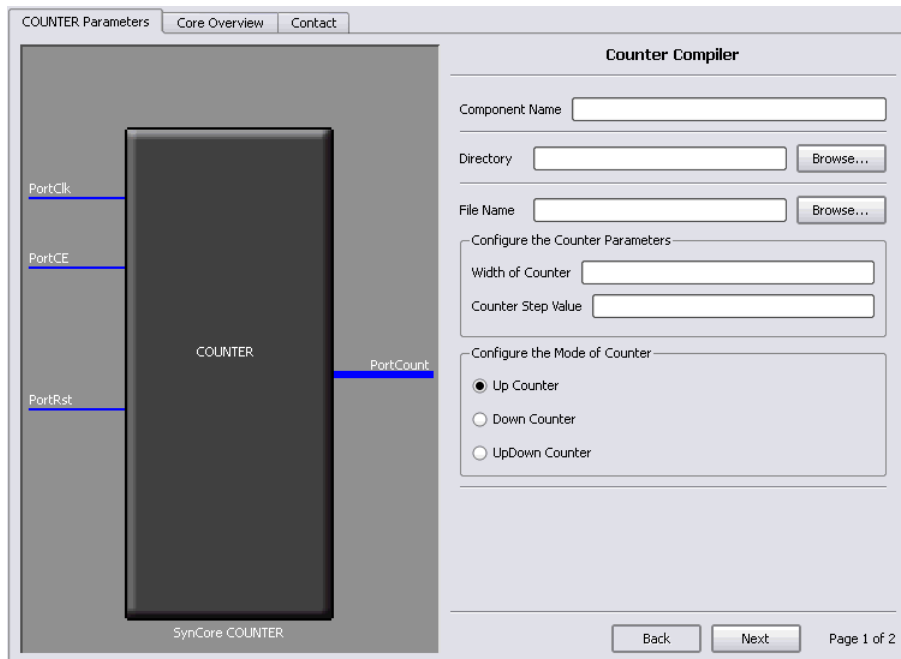
The SYNCORE IP wizard helps you generate Verilog code for your counter implementation requirements. The following procedure shows you how to generate Verilog code for a counter using the SYNCORE IP wizard.

Note: The SYNCORE counter model uses Verilog 2001. When adding a counter model to a Verilog-95 design, be sure to enable the Verilog 2001 check box on the Verilog tab of the Implementation Options dialog box or include a `set_option -vlog_std v2001` statement in your project file to prevent a syntax error.

1. Start the wizard.
 - From the FPGA synthesis tool GUI, select Run->Launch SYNCORE or click the Launch SYNCORE icon  to start the SYNCORE IP wizard.



- In the window that opens, select counter_model and click Ok to open page 1 of the wizard.



- Specify the parameters you need in the wizard. For details about the parameters, see [Specifying Counter Parameters](#), on page 522. The COUNTER symbol on the left reflects any parameters you set.

3. After you have specified all the parameters you need, click the **Generate** button in the lower left corner.

The tool displays a confirmation message (TCL execution successful!) and writes the required files to the directory you specified on page 1 of the wizard. The HDL code is in Verilog.

The SYNCore wizard also generates a testbench for your counter. The testbench covers a limited set of vectors. You can now close the wizard.

4. Add the counter you generated to your design.
 - Edit the counter files if necessary.
 - Use the **Add File** command to add the Verilog design file that was generated and the `syncore_addnsub.v` file to your project. These files are in the directory for output files that you specified on page 1 of the wizard.
 - Use a text editor to open the `instantiation_file.v` template file. This file is located in the same output files directory. Copy the lines that define the counter and paste them into your top-level module. The following figure shows a template file (in red text) inserted into a top-level module.

```

module counter #(
    parameter COUNT_WIDTH = 5,
    parameter STEP = 2,
    parameter RESET_TYPE = 0,
    parameter LOAD = 2,
    parameter MODE = "Dynamic" )
{
    // Output Ports
    output wire [WIDTH-1:0] Count,
    // Input Ports
    input wire Clock,
    input wire Reset,
    input wire Up_Down,
    input wire Load,
    input wire [WIDTH-1:0] LoadValue,
    input wire Enable );

    SynCoreCounter #(
        .COUNT_WIDTH(COUNT_WIDTH),
        .STEP(STEP),
        .RESET_TYPE(RESET_TYPE),
        .LOAD(LOAD),
        .MODE(MODE) )

    SynCoreCounter_inst1 (
        .PortCount (PortCount),
        .PortClk (PortClk),
        .PortRST (PortRST),
        .PortUp_nDown (PortUp_nDown),
        .PortLoad (PortLoad),
        .PortLoadValue (PortLoadValue),
        .PortCE (PortCE) );

endmodule

```

template

```

module counter #(
    parameter COUNT_WIDTH = 5,
    parameter STEP = 2,
    parameter RESET_TYPE = 0,
    parameter LOAD = 2,
    parameter MODE = "Dynamic")
(
    // Output Ports
    output wire [WIDTH-1:0] Count,
    // Input Ports
    input wire Clock,
    input wire Reset,
    input wire Up_Down,
    input wire Load,
    input wire [WIDTH-1:0] LoadValue,
    input wire Enable);

SynCoreCounter #(
    .COUNT_WIDTH(COUNT_WIDTH),
    .STEP(STEP),
    .RESET_TYPE(RESET_TYPE),
    .LOAD(LOAD),
    .MODE(MODE))

SynCoreCounter_inst1 (
    .PortCount(PortCount),
    .PortClk(Clock),
    .PortRST(Reset),
    .PortUp_nDown(Up_Down),
    .PortLoad(Load),
    .PortLoadValue(LoadValue),
    .PortCE(Enable));

endmodule

```



template

5. Edit the template port connections so that they agree with the port definitions in your top-level module as shown in the example below (the updated connection names are shown in red). You can also assign a unique name to each instantiation. You can also assign a unique name to each instantiation.


```

module counter #(
    parameter COUNT_WIDTH = 5,
    parameter STEP = 2,
    parameter RESET_TYPE = 0,
    parameter LOAD = 2,
    parameter MODE = "Dynamic")

(
    // Output Ports
    output wire [WIDTH-1:0] Count,
    // Input Ports
    input wire Clock,
    input wire Reset,
    input wire Up_Down,
    input wire Load,
    input wire [WIDTH-1:0] LoadValue,
    input wire Enable);

SynCoreCounter #(
    .COUNT_WIDTH(COUNT_WIDTH),
    .STEP(STEP),
    .RESET_TYPE(RESET_TYPE),
    .LOAD(LOAD),
    .MODE(MODE))

SynCoreCounter_ins1 (
    .PortCount(Count),
    .PortClk(Clock),
    .PortRST(Reset),
    .PortUp_nDown(Up_Down),
    .PortLoad(Load),
    .PortLoadValue(LoadValue),
    .PortCE(Enable));

endmodule

```

Port List

The following table lists the port assignments for all possible configurations; the third column specifies the conditions under which the port is available.

Port Name	Description	Required/Optional
PortCE	Count Enable input pin with size one (active high)	Always present
PortClk	Primary clock input	Always present
PortLoad	Load Enable input which loads the counter (active high).	Not present for parameter LOAD=0
PortLoadValue	Load value primary input (active high)	Not present for parameter LOAD=0 and LOAD=1
PortRST	Reset input which resets the counter (active high)	Always present
PortUp_nDown	Primary input which determines the counter mode. 0 = Up counter 1 = Down counter	Present only for MODE="Dynamic"
PortCount	Counter primary output	Always present

Specifying Counter Parameters

The SYNCore counter can be configured for any of the following functions:

- Up Counter
- Down Counter
- Dynamic Up/Down Counter

The counter core can have a constant or variable input load or no load value. If you are creating a constant-load counter, you will need to select Enable Load and Load Constant Value on page 2 of the wizard. If you are creating a variable-load counter, you will need to select Enable Load and Use Variable Port Load on page 2. The following procedure lists the parameters you need to define when generating a counter. For descriptions of each parameter, see [SYNCore Counter Wizard](#), on page 427 of the *Reference Manual*.

1. Start the SYNCORE counter wizard, as described in [Specifying Counters with SYNCORE, on page 516](#).
2. Enter the following on page 1 of the wizard:
 - In the Component Name field, specify a name for your counter. Do not use spaces.
 - In the Directory field, specify a directory where you want the output files to be written. Do not use spaces.
 - In the Filename field, specify a name for the Verilog file that will be generated with the counter definitions. Do not use spaces.
 - Enter the width and depth of the counter in the Configure the Counter Parameters section.
 - Select the appropriate configuration in the Configure the Mode of Counter section.
3. Click Next. The wizard opens page 2 where you set parameters for PortLoad and PortLoadValue.
 - Select Enable Load option and the required load option in Configure Load Value section.
 - Select the required reset type in the Configure Reset type section.

The COUNTER symbol dynamically updates to reflect the parameters you set.
4. Generate the counter core by clicking Generate button. All output files are written to the directory you specified on page 1 of the wizard.

The Synopsys FPGA IP Encryption Flow

The Synopsys FPGA IP encryption flow is a design flow that encourages interoperability while protecting IP implementations using encryption/decryption technologies. This flow offers the following advantages: interoperability, protection of IP, reuse of IP, and a standard flow for IP encryption. Currently, Synopsys FPGA synthesis products support the following encryption technologies:

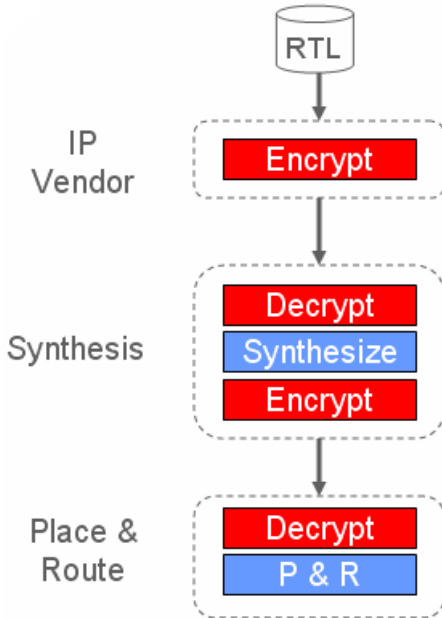
- IEEE 1735-2014 with key-block (Version 1)
- OpenIP
- Synenc-encrypted IP

See the following encryption technologies:

- [Encrypting IP with the encryptP1735.pl Script](#), on page 530
- [Encrypting IP with the encryptIP Script](#), on page 536

Overview of the Synopsys FPGA IP Flow

The complete flow for protecting IP requires a partnership between the IP vendor, Synopsys, and the silicon vendor as illustrated in the following figure. However, depending on the level of agreement between Synopsys and the silicon vendor downstream, the re-encryption of IP following synthesis can vary from the ideal flow shown in the figure.

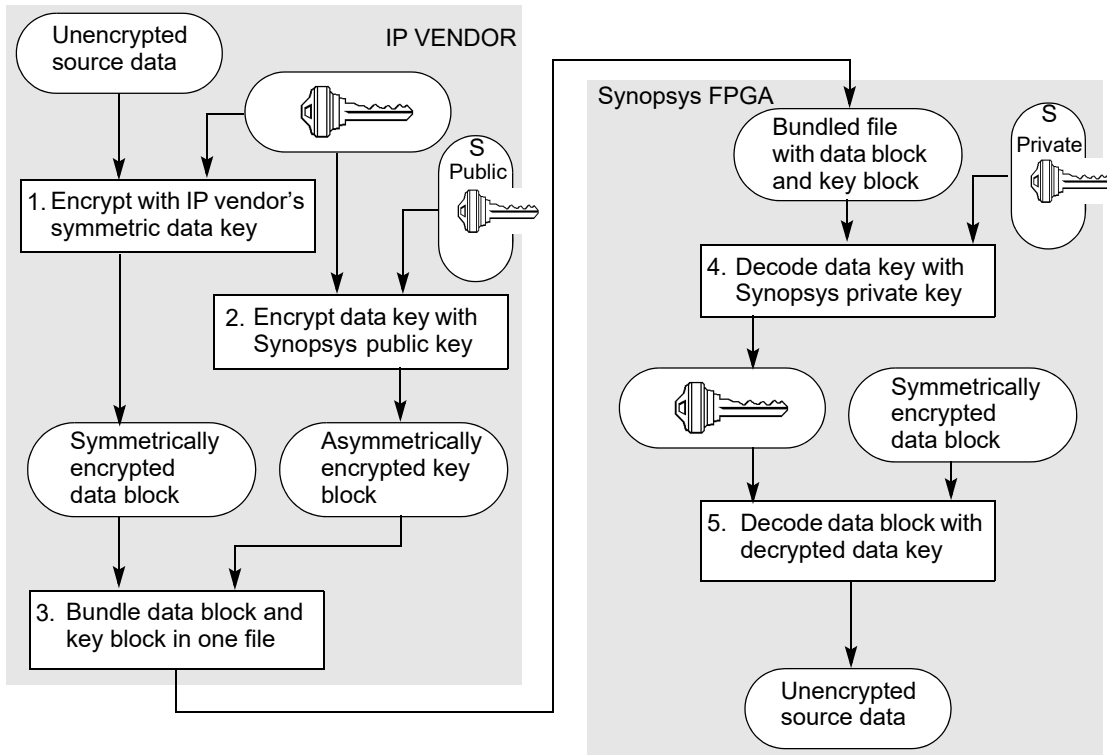


For further details of the hand-offs between vendors and how encryption and decryption are handled, see [Encryption and Decryption, on page 525](#).

Encryption and Decryption

There are two major classes of encryption/decryption algorithms: symmetric, and asymmetric. Each has its own advantages and disadvantages. The approach for the Synopsys FPGA IP flow is a hybrid scheme that uses both asymmetric and symmetric encryption to leverage the strengths of each scheme. The methodology described here can also be used for other design hand-offs. For example, for a handoff from synthesis to place-and-route, the synthesis tool would be in the upstream position occupied by the IP vendor in this flow, and the FPGA vendor would be in the downstream position occupied by the synthesis tool.

The following figure illustrates the steps in this encryption/decryption methodology, showing the handoff from an IP vendor to a Synopsys FPGA synthesis tool.



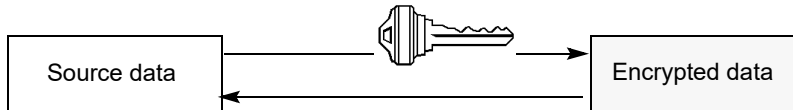
The following describes each of the phases shown in the figure. Note that Synopsys provides the following scripts to simplify and automate the process of encrypting data for the IP vendor.

- IEEE 1735-2014
-
- OpenIP

Data Encryption - Step 1

The IP vendor encrypts the IP data using their own symmetric key. This key is called the *data key*. The result of encoding is a *data block*. Using symmetric encryption offers two advantages to the IP vendor: fast data encryption because it is symmetric encryption, and freedom to use any symmetric scheme they choose.

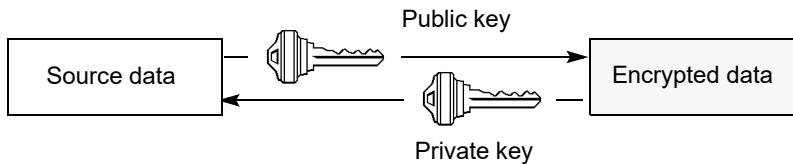
Symmetric Encryption/Decryption with One Key



Data Key Encryption - Step 2

Next, the IP vendor encrypts the data key used to encode the IP block, and generates a *key block*. For this operation, the vendor uses RSA asymmetric encryption and the public key provided by Synopsys.

Asymmetric Encryption/Decryption with Public and Private Keys



Asymmetric encryption offers the following advantages:

- Although asymmetric encryption is compute-intensive, the data key itself is small, so this is not time-intensive.
- The IP vendor can use public keys from different vendors to encrypt the same block for different EDA vendors. This capability ensures that IP consistency is maintained, because there is no need for multiple copies.
- Only the public key from the downstream vendor needs to be passed to the IP vendor.

Bundling of Encrypted Data Block and Data Key - Step 3

The IP vendor bundles the encrypted data block with the key block into one file for handoff to the EDA vendor. Note that this methodology allows the IP vendor to create just one version of the IP which includes the key blocks for all the downstream vendors it supports; for example, a synthesis tool and a simulation tool. Also, this approach eliminates the need to securely transmit the symmetric key, because this is included in the file. Security is maintained because both the key and the data are encrypted.

In the figure, this is the point at which the IP vendor hands off the IP to the synthesis tool.

Data Key Decryption - Step 4

Decryption is a two-stage process. The first step is to decrypt the symmetric data key from the IP vendor, which was encrypted using the asymmetric public key provided. To decode this key, use the private key counterpart to the public key and extract the data key.

Data Decryption - Step 5

The second step is to use the extracted data key to access the IP data. As the data key is the original symmetric key used to encode the IP, the process is quick. The synthesis tools can now synthesize the unencrypted IP.

After synthesis, the IP can be re-encrypted if the vendor has adopted one of the Synopsys methodologies.

Re-Encryption in the Synopsys FPGA IP Flow

Re-encryption of the synthesized IP for FPGA vendors downstream requires that the FPGA vendor supply Synopsys with a public key. When the input file includes a downstream key block, the re-encrypted data is accessible to the downstream tool. If such an agreement is not in place, the IP is treated as a black box. Accordingly, you can have an IP flow that outputs black boxes in the netlists, plaintext netlists, or encrypted netlists.

Working with Encrypted IP

The Synopsys FPGA IP encryption schemes available include:

- IEEE 1735-2014
-
- OpenIP

With either of these approaches, the IP vendor can encrypt and control distribution of the IP from their own website. The synthesis user will have access from the synthesis tool to the IP that the vendor makes available for download and evaluation within a synthesis design.

The following sections describe how to encrypt and package your IP for evaluation if you are an IP vendor, and how to access and evaluate available IP, if you are an end-user.

- [Encrypting Your IP](#), on page 529
- [Preparing the IP Package](#), on page 540

Encrypting Your IP

IP vendors can use either of the supported Synopsys FPGA IP schemes to provide IP for synthesis users to evaluate and use. Both schemes uses a two-stage encryption process:

- First, encrypt your IP files using a symmetric encryption algorithm and your own session or data key to create an encrypted data block.
- Next, encrypt the session key for the encrypted data block using an asymmetric algorithm and the Synopsys public key. The Synplify tool currently supports RSA encryption.

Synopsys provides scripts to simplify this process. See the following procedures for details on script usage.

Preparing and Encrypting Your IP

To prepare and encrypt your IP, do the following:

1. Gather your RTL files.

You only encrypt the RTL. You can encrypt any number of Verilog and VHDL (or mixed) RTL files to form your encrypted IP, and each file can be encrypted in its entirety.

2. Determine your file setup for each IP.

- Create a single set of files for the IP (for use with all supported FPGAs) if your IP has no vendor-specific or vendor-optimized content and if the output method is supported by all intended consumers (blackbox or plaintext).
- Create multiple versions of your protected IP if you have specific FPGA vendors or specific FPGA vendor families, if you are using FPGA device-family specific RTL like architecture-specific instantiations, or if you optimized your RTL or constraints for use with a specific FPGA vendor device family or FPGA vendor.
- Encrypt the files with the appropriate encryption script as described in one of the following subsections: [Encrypting IP with the encryptP1735.pl Script, on page 530](#)
- [Encrypting IP with the encryptIP Script, on page 536](#).

3. Package your IP, as described in [Preparing the IP Package, on page 540](#).

4. Verify that your IP works with the synthesis tools by going through the procedure that the user would use.

- Start the synthesis tool and load the IP with the Import IP->Import IP Package command. You can load your IP into an existing Synplify project.
- Run synthesis.

Encrypting IP with the encryptP1735.pl Script

The encryptP1735.pl script supports the IEEE 1735-2014 standard with limited interoperability. The encryptP1735.pl script accepts inputs from three sources: command line arguments, the RTL input file, and a file containing the public keys. The RTL input file, depending on the use model, may contain encryption attributes.

A `keys.txt` file, which contains the public key for consumption by Synopsys FPGA tools, is included with the script. Add other public keys to this file when the IP is to be consumed by additional EDA tools.

The following procedure shows you how to encrypt your data with the `encryptP1735.pl` script. This script automates the two-stage encryption process described in the Synopsys FPGA IP scheme (*The Synopsys FPGA IP Encryption Flow*, on page 524). The `encryptP1735.pl` script:

- First encrypts your IP files using a symmetric encryption algorithm and your own session or data key to create an encrypted data block.
- Next encrypts the session key for the encrypted data block using an asymmetric algorithm and the Synopsys public key. The tool currently supports RSA encryption.

The `encryptP1735.pl` script is located in the `installDir/lib` directory and requires the installation of Perl on your machine. The following examples show typical script applications. For more information on the script and the command line arguments, see *encryptP1735*, on page 57 in the Command Reference Manual. The `encryptP1735` encryption script supports the following three use models for encrypting RTL files:

- Full-File Use Model
- Partial File with All Pragmas Use Model
- Partial File with Minimal Pragmas Use Model

Full-File Use Model

With a full-file use model, the RTL contains no encryption-related pragmas, and the entire RTL file is encrypted by `encryptP1735.pl` to create the decryption envelope. This use model is intended to be used with complete RTL files that do not require the addition of encryption attributes; the `encryptP1735.pl` script automatically adds these attributes to create the decryption envelope.

To illustrate the full-file use model, consider a single, Verilog file (`tb_encrypt.v`) to be encrypted without pragmas. This file contains a single module named `secret`.

```
module secret (a, b, clk);  
  input a, clk;  
  output b;  
  reg b=0;
```

```
always @(posedge clk) begin
    b = a;
end
endmodule
```

With no encryption-related pragmas in the RTL file, a decryption envelope is created with the command:

```
perl encryptP1735.pl -list mylist -log encryptP1735.log
```

In the above command, the list file (mylist) contains the single Verilog file `tb_encrypt.v`. The command uses the default `keys.txt` file from the directory *installLocation/lib* as the public keys file to create the decryption envelope file `tb_encrypt.vp`. The resulting messages are written in the `encryptP1735.log` file.

Partial File with All Pragmas Use Model

With the partial file with all pragmas use model, the RTL contains all of the encryption-related pragmas which are used by the `encryptP1735.pl` script with the default `keys.txt` file to create the decryption envelope.

When there are overlapping pragmas in the RTL and the `keys.txt` file, the RTL pragma takes precedence over the corresponding pragma in the `keys.txt` file. For example, if the `data_method` pragma contains `des-cbc` in the RTL and `aes128-cbc` in the `keys.txt` file, the following pragma is copied to the decryption envelope:

```
data_method="des-cbc"
```

Verilog Example

To illustrate the partial file with all pragmas use model, consider a single, Verilog file (`tb_encrypt.v`) to be encrypted. This file contains a module named `secret` and all the encryption-related pragmas with the exception of the `key_public_key` in the RTL itself.

```
module secret (a, b, clk);
input a, clk;
output b;

`pragma protect version=1
`pragma protect encoding=(enctype="base64")
`pragma protect author="author-a", author_info="author-a-details"
`pragma protect encrypt_agent="encryptP1735.pl", encrypt_agent_info="Synplify encryption scripts"
`pragma protect key_keyowner="Synopsys",key_keyname="SYNP05_001", key_method="rsa", key_block
`pragma protect data_keyowner="ip-vendor-a",data_keyname="fpga-ip", data_method="des-cbc"
`pragma protect begin
reg b=0;
```

```
always @(posedge clk) begin
    b = a;
end
`pragma protect end
endmodule
```

With the encryption-related pragmas in the RTL file, a decryption envelope is created with the command:

```
perl encryptP1735.pl -list mylist
```

In the above command, the list file (mylist) contains the single Verilog file `tb_encrypt.v`. The command uses the default `keys.txt` file from the directory *installLocation/lib* as the public keys file to create the decryption envelope file `tb_encrypt.vp`. Any messages from the run are not output to a log file.

VHDL Example

To encrypt a partial VHDL file with an all pragmas use model, consider the single, VHDL file (`tb_encrypt.vhd`). The file contains a single entity/architecture pair named `secret` with all the encryption-related pragmas with the exception of the `key_public_key` in the RTL itself.

Note: VHDL formatted pragmas do not include a “pragma” string preceding the keyword “protect.”

```
library IEEE;
use IEEE.std_logic_1164.all;

entity secret is
    port (clk : in std_logic;
          a : in std_logic;
          b : out std_logic);
end entity;

architecture rtl of secret is

    `protect version=1
    `protect author="author-a", author_info="author-a-details"
    `protect encrypt_agent="encryptP1735.pl", encrypt_agent_info="Synplify encryption scripts"
    `protect encoding=(enctype="base64")
    `protect key_keyowner="Synopsys", key_keyname="SYNP05_001", key_method="rsa", key_block
    `protect data_keyowner="ip-vendor-a", data_keyname="fpga-ip", data_method="des-cbc"
    `protect begin

    signal b_reg: std_logic;
    begin
        process (clk) is
        begin
            if rising_edge(clk) then
                b_reg <= a;
            end if;
        end process;
        b <= b_reg;

    `protect end
```

```
end architecture;
```

With the encryption-related pragmas in the RTL file, a decryption envelope is created with the command:

```
perl encryptP1735.pl -list mylist
```

In the above command, the list file (mylist) contains the single VHDL file `tb_encrypt.vhd`. The command uses the default `keys.txt` file from the directory *installLocation/lib* as the public keys file to create the decryption envelope file `tb_encrypt.vhdp`. Any messages from the run are not output to a log file.

Partial File with Minimal Pragmas Use Model

With the partial file with minimal pragmas use model, the RTL contains only the begin and end encryption-related pragmas to indicate the start and end points of the encryption. This use model is helpful when automatic insertion is used to reduce the amount of manually inserted pragmas. After defining the start- and end-point pragmas, the `encryptP1735.pl` script, using the default `keys.txt` file, creates the decryption envelope. To illustrate this use model, consider a single, Verilog file (`tb_encrypt.v`) to be encrypted with only begin and end pragmas. This file contains a single module named `secret`.

```
module secret (a, b, clk);
  input a, clk;
  output b;
  `pragma protect begin
  reg b=0;

  always @(posedge clk) begin
    b = a;
  end
  `pragma protect end
endmodule
```

With the start and end pragmas in the RTL file, a decryption envelope is created with the command:

```
perl encryptP1735.pl -list mylist
```

In the above command, the list file (mylist) contains the single Verilog file `tb_encrypt.v`. The command uses the default `keys.txt` file from the directory *installLocation/lib* as the public keys file to create the decryption envelope file `tb_encrypt.vp`. The absence of a specified log file (-log option) results in no messages being written to the log file.

Encrypting Multiple RTL Files

The examples in the previous sections included a single Verilog or VHDL file. Multiple files can be similarly encrypted by adding their file names to the specified list file. Running the script produces encryption envelopes for each listed file. To create a list file for multiple files, add the name of each file to be encrypted on a separate line.

Public Keys Repository File

The encryptP1735.pl encryption script requires public keys from the file specified by the `-public_keys` (or `-pk`) option. This file includes public keys for each of the tools that require a key block in the encrypted file.

```
// Use verilog pragma syntax in this file

`pragma protect version=1
`pragma protect author="default"
`pragma protect author_info="default"

`pragma protect key_keyowner="Synopsys", key_keyname="SYNP05_001", key_method="rsa"
`pragma protect key_public_key

MIIBIjANBgkqhkiG9w0BAQEFAAOCAQ8AMIIBCgKCAQEAYbsQaMidichZyh14wbXn
UpP8lK+jJY5oLpGqDfSW5PMXBVp0Wfd1d32onXEpRkwxEJLlK4RgS43d0FG2ZQ1l
irdimRKNnUtPxsrJzbMr74MQkwmG/X7SEe/1EqwK9Uk77cMEncLycI5yX4f/K9Q9
WS5nLD+Nh6BL7kwR0vSevfePC1fkOaluC7b7Mwb1mcqCLBBRP9/eF0wUIoxVRzjA
+pJvORwhYtZEhnwvTb1BJsnynet1LfDi/D5WZoikTP/0KBiP87QHMSuVByDMA7J7
g6sxKB92hx2Dpv1ojds1Y5ywjxFxOAA93nFjmLsJq3i/P01v5TmtnCyx3Wkryw4B
eQIDAQAB

// Add additional public keys below this line
// Add additional public keys above this line

`pragma protect data_keyowner="default-ip-vendor"
`pragma protect data_keyname="default-ip-key"
`pragma protect data_method="aes128-cbc"

// End of file
```

For the partial file with all pragmas use model, the following pragma attribute values must match the corresponding values in the key-block section of the encryption envelope:

```
`pragma protect key_keyowner="Synopsys", key_keyname="SYNP05_001", key_method="rsa"
```

Note: This encryptP1735.pl script is compatible only with versions I-2013.09 and later of the Synopsys FPGA synthesis tool.

Encrypting IP with the encryptIP Script

The following procedure shows you how to encrypt your data with the encryptIP (OpenIP) script. The encryptIP script automates the two-stage encryption process proposed in the Synopsys FPGA IP scheme ([The Synopsys FPGA IP Encryption Flow](#), on page 524).

- First, it encrypts your IP files using a symmetric encryption algorithm and your own session or data key. This creates an encrypted data block.
 - Next, it encrypts the session key for the encrypted data block using an asymmetric algorithm and the Synopsys public key. The Synplify tool currently supports RSA encryption.
1. Install the encryptIP Perl script.
 - Download the encryptIP Perl script.
 - Install Perl on your machine. You cannot run the script if you do not have Perl installed.
 2. Make sure that the encryptIP script specifies the decryption key and the matching key length:
 - Specify the symmetric data decryption key with the `-k` option. Optionally, you can also specify a symmetric encryption key in hexadecimal format with the `-kx` option.
 - Make sure you specify the right key length for the encryption algorithm with the `-c` option. For example, TEST1234 becomes a 64-bit key, so you specify the des-cbc algorithm.

See [Perl encryptIP Script Syntax](#), on page 55 in the *Reference Manual* for full details of the encryptip syntax.

3. Make sure you specify the appropriate output method (`-om`) when you run the script.

This is important because the output method (`-om`) determines what is encrypted to the user. When the example above is synthesized, the user can view the output netlist because the output method specified is plain-text, which means that the synthesis output netlist includes the IP netlist in an unencrypted and readable form. See [Specifying the Script Output Method](#), on page 537 for more information.

The script encrypts the IP with the standard symmetric encryption algorithm you specified, and produces a `data_block`. The data key used for

encrypting the HDL is then encrypted with an asymmetric algorithm and the Synopsys public key, and produces a `key_block`. The `data_block` and the `key_block` are combined with the appropriate pragmas for the flow being used, and the script creates an encrypted HDL file. For a detailed figure, see [Encryption and Decryption, on page 525](#).

All other output files from synthesis, including `srm`, `srd`, and `vm` files, are encrypted using the same encryption method specified for the input to synthesis. Output constraints are not encrypted.

4. Run the `encryptIP` script on each RTL file you want to encrypt.

The following example encrypts the Verilog `plain_ip.v` file into an encrypted file called `protected_ip.v`, using AES128-cbc encryption. The session key is `MY_AES_SAMPLEKEY`. See [Perl encryptIP Script Syntax, on page 55](#) in the *Reference Manual* for details about the syntax and required parameters.

```
perl encryptIP -in plain_ip.v -out protected_ip.v -c aes128-cbc  
-k MY_AES_SAMPLEKEY -bd 16OCT2007 -om plaintext -v
```

5. Check the encrypted RTL file to make sure that there is only one key block present.

Specifying the Script Output Method

You can control access to the IP by setting the appropriate output method. You specify the output method using the `-om` parameter, as described in [Perl encryptIP Script Syntax, on page 55](#) or [Perl encryptP1735 Script Syntax, on page 58](#) in the *Reference Manual*.

The output method mainly affects the output netlist. The following are guidelines for setting the output method for the `encryptIP` script, and detail the effects of different settings:

1. When using the `encryptIP` script, set `-om` to `persistent_key` in the following cases:
 - If you are working with a Lattice non-CPLD technology
 - If you have an agreement in place with Synopsys and want the output netlist to be encrypted
2. Set `-om` to `plaintext` in the following cases:

- If you want to allow the IP to be incorporated in a logic synthesis design

Setting the output method to `plaintext` allows the tool to synthesize, run gate-level simulations, place and route, and implement an FPGA (that includes the IP) on a board.

- If you want the IP to be freely optimized by the synthesis tools

Although IP cores are already optimized, the synthesis tools can effect additional optimizations based on the design context in which it will be used. When the synthesis tool is allowed to optimize the IP, it can prune away IP logic that is unused or unnecessary in the current design context. Or take the case where the output of an instantiated IP core is timing-critical because it drives hundreds of user loads. If the synthesis tool can freely optimize, it can replicate sources within the core and fix the problem.

3. To let the IP be incorporated in a logic synthesis design, set `-om` to `plaintext` or `blackbox`.

Setting the output method to `plaintext` allows the tool to synthesize, run gate-level simulations, place and route, and implement an FPGA (that includes the IP) on a board. Setting the output method to `blackbox` does not allow the tool to run gate-level simulations or place and route the IP, because it only uses the port and connectivity information.

4. If you have set `-om` to `plaintext` and you want to specify individual cores as white boxes, set the `syn_macro` directive to 1 on the view for the IP.

Note that you must set this on the view, not the instance. When this is set, the tool treats the IP as a white box and only uses the timing and connection information from the IP. The synthesis tool maintains the IP boundary and only trims unused logic inside the IP.

5. During synthesis, the IP contents appear as a black box in the RTL and Physical Analyst views, irrespective of the output method selected. When the output method is set to `plaintext`, you can push down into the IP from the Technology view.
6. After synthesis, the output method affects the results in the following ways:
 - Output constraints for an IP are in the standard Synopsys format and are not encrypted.

- The output method affects the contents of the output netlist and its format. This table summarizes the encryptIP or encryptP1735 behavior with different output methods.

Method (-om)	Output Netlist After Synthesis
blackbox	The output netlist contains the IP interface only, and no IP contents. It only includes IP ports and connections. The IPs are treated as black boxes, and there are no nets or instances shown inside the IP. This applies to all the netlist formats generated for different vendors, whether it is HDL (vm or vhm), EDIF (edf or edn), or vqm.
plaintext	The output netlist contains your unencrypted synthesized IP, which is completely readable (nothing is encrypted).
persistent_key (encryptIP only)	The output netlist includes encrypted versions of the IP. For Lattice designs, the tool generates one output netlist (EDIF or HDL) that includes the encrypted IP blocks. The netlist is readable, except for the IP block sections, which are encrypted.

Preparing the IP Package

Do the following to package your IP and make it accessible from the synthesis tools:

1. Collect the files for the package.
 - Encrypt the files you need, as described in [Encrypting Your IP, on page 529](#).
 - Make sure your package includes the files listed in [IP Package File List, on page 541](#).
 - Structure the files as described in [Suggested Directory Structure, on page 541](#).
2. If your IP package is intended for synthesis only, without subsystem assembly, create a compressed package for download, using one of these methods:
 - Create a compressed tarball (.tar.gz), which is a tar archive compressed with the gzip utility, using one of these commands:

```
tar cf -fileList | gzip -c > compressed-tarball
gtar -cf compressed-tarball fileList
```

Preserve the directory structure when you run gzip.

- Create a zip file (zip) by running WinZip. WinZip archives and preserves your directory hierarchy.
3. Post the packaged IP on your website for downloading.

The user generally untars or unzips the IP package into a top-level directory after downloading it. The synthesis tools can then read the contents of the directory.
 4. Supply Synopsys with the following:
 - The URL for the download package.
 - Vendor and advertising information you wish to display on the Synopsys website. See [Supplying Vendor Information, on page 541](#) for details.

IP Package File List

Your IP package should contain the following files:

Files	Description
ipinfo.txt	Text file that lists the name of the IP, the version, restrictions for use, support contact information, and an email alias to request a licence for the full RTL for your IP.
Documentation, preferably a PDF	Documents the IP, and includes detailed information about usage restrictions like vendor, device family, etc.
Readme	An optional text file that contains instructions on use of the IP for assembly and/or synthesis, and hints on how to use it correctly.
Encrypted HDL or EDIF	Protected RTL for the IP, created using the Synopsys encryptIP script. See the documentation for details.
SDC constraints	Unencrypted design constraints for the IP.

Suggested Directory Structure

Follow these recommendations when you structure the IP package:

- Always use relative paths to reference a file.
- Always preserve directory structure when you run `gzip`.
- You can place IP-XACT `.xml` files in the top-level directory or in a common subdirectory. You can have multiple files or a single file for the same component or variants of a component. However, it is preferred that you keep all IP-XACT components that are in one library at the same directory level, even if it is many levels deep in the directory hierarchy.

Supplying Vendor Information

To make your IP accessible for downloads and evaluation from the Synopsys synthesis tools, you must supply Synopsys with some vendor information as well as information for each of the cores or IPs to be used.

1. Supply Synopsys with the following general information to advertise your company and IP on the Synopsys website:

IP vendor name and logo	Your vendor name and logo for display.
Optional IP description	Short paragraph describing the IP and key features.
Email alias	Synopsys sends leads to this alias when evaluation cores are requested on the Synopsys IP website.
Website URL	Unique URL for accessing IP. After the user has filled out lead information on the website, the Synopsys tool directs the user to this URL to download the IP. The lead form on your website can be pre-filled by prior arrangement with Synopsys Marketing.

2. Supply Synopsys with the following information about each core or IP to be used:

IP name	Name of the IP.
IP short description	Sentence describing the IP, which is displayed in the summary view on the Synopsys website.
IP paragraph description	More detailed description of the IP, covering functional description and compatibility with other cores or peripherals.
Notes about usage	Any other information, like licensing requirements
Core datasheet (HTML or PDF)	Information about the characteristics, features, functions, and interfaces.
Supported FPGA vendors and devices	List of the targeted vendors and devices that the core supports.
IP-XACT compatibility information	List of the IP-XACT version number supported, the IP-XACT VLVN, and the IP-XACT VLVNs of all the bus definitions required for the core, along with a link to download each of these bus definitions.

Using Hyper Source

Hyper source is a useful feature that lets you prototype ASIC designs that use one or more FPGAs. You can also use it to validate and debug the RTL for IP designs. See the following for details:

- [Using Hyper Source for Prototyping](#), on page 543
- [Using Hyper Source for IP Designs](#), on page 543
- [Threading Signals Through the Design Hierarchy of an IP](#), on page 544

Using Hyper Source for Prototyping

For prototyping, use Hyper Source to efficiently thread nets across multiple modules to the top-level design to support Time Domain Multiplexing (TDM).

You can also use it to easily replace an ASIC RAM with an FPGA RAM. Follow these guidelines to replace an ASIC RAM with an FPGA RAM:

1. Change the RTL for the RAM instantiation.
2. Add an extra clock signal to all the module interfaces.

Hyper source reduces the number of modified RTL modules to two, one for the RAM and one for the top level.

Using Hyper Source for IP Designs

For IP designs, Hyper Source is useful for validating and debugging the RTL without directly modifying it. After the RTL has been fully tested with complete QoR results, use Hyper Source to debug, as described in the following cases:

- Add some instrumentation logic that is not part of the original design, such as a cache profiler that counts cache misses or bus monitor that might count statistics about bus contention. The cache or bus might be buried deep inside the RTL; accessing the cache or the bus means ports might need to be added through several levels of hierarchy in the RTL. The instrumentation logic can be included anywhere in the design, so you can use hyper source and hyper connect to easily thread the necessary connections during synthesis.

- Insert other hyper sourcing inside the IP to probe, monitor, and verify correct operation of known signals within the IP.

Threading Signals Through the Design Hierarchy of an IP

Use this mechanism to thread a signal through the design hierarchy of a user IP. This signal can be threaded to a top-level port or signal. This works even if the Verilog or VHDL is compiled separately. The tool automatically adds ports and signals between the source and the connection. Otherwise, these connections must be manually added to the RTL code.

The following procedure describes a method for using hyper source, using the example HDL shown in [Hyper Source Example, on page 545](#).

1. Define how to connect to the signal source. The following apply to this example:
 - Signal `syn_hyper_source (in1)` module defines the source, with a width of 1.
 - The tag name "tag_name" is the global name for the hyper source.
2. Define how to access the hyper source which drives the local signal or port. The following apply to this example:
 - Signal `syn_hyper_connect (out1)` module defines the connection. The signal width of 1 matches the source.
 - Tag name can be the global name or the instance path to the hyper source.
3. In this hierarchical design, note the following about hyper source:
 - Applies to the module `lower_module`.
 - Signal `syn_hyper_source my_source(din)` module is defined for the source with a width of 8.
 - The tag name of "probe_sig" must match the name used in the hyper connect block to thread the signal properly.
4. In this hierarchical design, note the following about the hyper connect:
 - Applies to the top-level module `top`, but can be any level of hierarchy.
 - Signal `syn_hyper_connect connect_block (probe)` module is defined for the connection with a width of 8.

- Tag name of "probe_sig" must match the name used in the hyper source block to thread the signal properly.

5. After you run synthesis, the following message appears in the log file:

```
Available hyper_sources - for debug and ip models
HyperSrc label sub2_module.sub1_module.lower_module.probe_sig

Making connections to hyper_source modules
@N: : hyper_example.v(54) | Connected syn_hyper_connect hstdm_training_done_connect, label probe_sig
Finished RTL optimizations (Time elapsed 0h:00m:01s; Memory used current: 122MB peak: 129MB)
```

Hyper Source Example

Verilog Example

```
/* Connect to a signal you want to export example : in1*/
module syn_hyper_source(in1) /*synthesis syn_black_box=1
syn noprun=1 */;
parameter w = 1;
parameter tag = "tag_name"; /* global name of hyper_source */
input [w-1:0] in1;
endmodule

/* Use to access hyper_source and drive a local signal or port example
:out1 */
module syn_hyper_connect(out1) /* synthesis syn_black_box=1
syn noprun=1 */;
parameter w = 1; /* width must match source */
parameter tag = "tag_name"; /* global name or instance path to
hyper_source */
parameter dflt = 0;
parameter mustconnect = 1'b1;
output [w-1:0] out1;
endmodule

/* Example hierarchical design which uses hyper_source */
module lower_module (clk, dout, din1, din2, we);
output reg [7:0] dout;
input clk, we;
input [7:0] din1, din2;
wire [7:0] din;

syn_hyper_source my_source(din);
defparam my_source.tag = "probe_sig"; /* to thread the signal this
tag name must match to name used in the hyper connect block */
defparam my_source.w = 8;

always @(posedge clk)
if (we)
dout <= din;
assign din = din1 & din2;
endmodule

module sub1_module (clk, dout, din1, din2, we);
output [7:0] dout;
input clk, we;
input [7:0] din1, din2;
```

```

lower_module lower_module (clk, dout, din1, din2, we);
endmodule

module sub2 module (clk, dout, din1, din2, we);
output [7:0] dout;
input clk, we;
input [7:0] din1, din2;
sub1 module sub1_module (clk, dout, din1, din2, we);
endmodule

module top (clk, dout, din1, din2, we, probe);
output[7:0] dout;
output [7:0] probe;
input clk, we;
input [7:0] din1, din2;

syn_hyper_connect connect block(probe);
defparam connect_block.tag = "probe_sig"; /* to thread the signal
this tag name must match to name used in the hyper connect block */
defparam connect_block.w = 8;

sub2_module sub2_module (clk, dout, din1, din2, we);
endmodule

```

VHDL Example

```

library ieee ;
use ieee.std_logic_1164.all;
use ieee.numeric_std.all;

entity lower_module is

    port (
        clk: in std_logic;
        we: in std_logic;
        din1: in std_logic_vector (7 downto 0);
        din2: in std_logic_vector (7 downto 0);
        dout: out std_logic_vector(7 downto 0)
    );

end entity lower_module;

architecture behav1 of lower_module is
    signal din: std_logic_vector(7 downto 0);

    component syn_hyper_source
        generic (
            w: integer := 8;
            tag: string := "tag_name");

        port (in1 : in std_logic_vector (w-1 downto 0));
    end component;

begin

    process (clk,we) begin

```

```

        if (we='1') then
            if (clk'event and clk = '1') then
                dout<= din;
            end if;
        end if;
    end process;

    din <= din1 and din2;

my_source : syn_hyper_source
generic map (w => 8,tag => "tag1") port map (din); -- SYN_HYPER_SOURCE
module (This is source module)

end behav1;

=====
library ieee;
use ieee.std_logic_1164.all;

entity subl_module is
    port(
        clk: in std_logic;
        we: in std_logic;
        din1 : in std_logic_vector(7 downto 0);
        din2 : in std_logic_vector(7 downto 0);
        dout: out std_logic_vector(7 downto 0)
    );

end entity subl_module;

architecture behav of subl_module is
    component lower_module is
    port(
        clk: in std_logic;
        we: in std_logic;
        din1: in std_logic_vector (7 downto 0);
        din2: in std_logic_vector (7 downto 0);
        dout: out std_logic_vector(7 downto 0)
    );
    end component;

begin
    lmodule: lower_module
    port map ( clk,we,din1,din2,dout);
end behav;

=====
library ieee;
use ieee.std_logic_1164.all;

```

```

entity sub2_module is
    port(
        clk: in std_logic;
        we: in std_logic;
        din1: in std_logic_vector(7 downto 0);
        din2: in std_logic_vector(7 downto 0);
        dout: out std_logic_vector(7 downto 0)
    );

```

```

end entity sub2_module;

```

```

architecture behav of sub2_module is
    component sub1_module is
    port(

```

```

        clk: in std_logic;
        we: in std_logic;
        din1: in std_logic_vector (7 downto 0);
        din2: in std_logic_vector (7 downto 0);
        dout: out std_logic_vector(7 downto 0)
    );

```

```

end component;

```

```

begin
    smodule: sub1 module
    port map ( clk,we,din1,din2,dout);
end behav;

```

```

=====

```

```

library ieee ;
use ieee.std_logic_1164.all;
use ieee.numeric_std.all;

```

```

entity top is
    port (
        clk: in std_logic;
        we: in std_logic;
        din1: in std_logic_vector (7 downto 0);
        din2: in std_logic_vector (7 downto 0);
        dout: out std_logic_vector(7 downto 0);
        probe: out std_logic_vector(7 downto 0)
    );

```

```

end entity top;

```

```

architecture behav2 of top is

```

```

    component syn_hyper_connect
        generic (
            w: integer := 8;
            tag: string := "tag_name"
        );

```

```

port (out1: out std_logic_vector (w-1 downto 0));
end component;

component sub2_module
port(
    clk: in std_logic;
    we: in std_logic;
    din1: in std_logic_vector (7 downto 0);
    din2: in std_logic_vector (7 downto 0);
    dout: out std_logic_vector(7 downto 0)
);

end component;

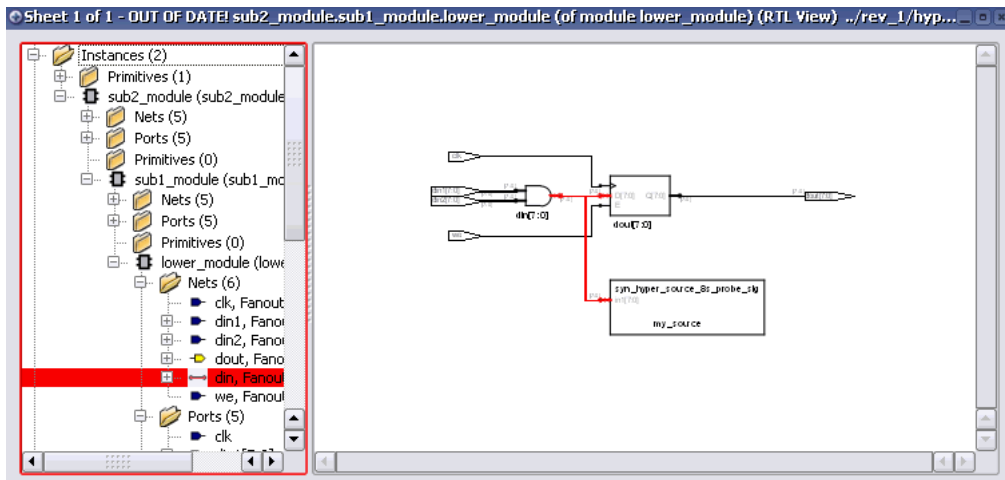
begin
my_connect: syn_hyper_connect generic map (w => 8, tag => "tag1") port
map (probe);
smodule: sub2_module

port map ( clk,we,din1,din2,dout);
end behav2;

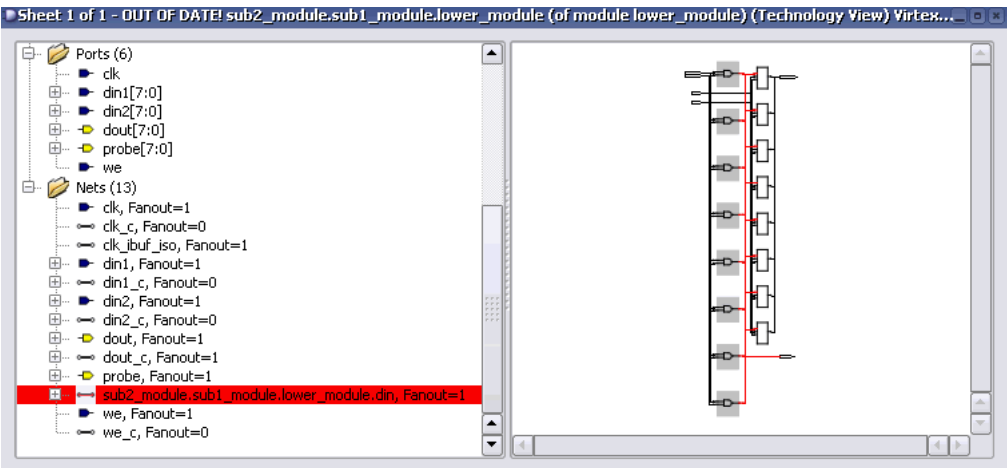
```

The following figures show how the hyper source signal automatically gets connected through the hierarchy of the IP in the HDL Analyst views.

RTL View



Technology View



Working with Lattice IP

The Lattice IP encryption scheme uses the flow described in [Encryption and Decryption, on page 525](#). The following procedure details how to incorporate Lattice encrypted IP in your design.

1. Create a synthesis project.
2. Obtain the IP from Lattice, and add the encrypted file to your project.

The Lattice IP is encrypted using the `persistent_key` method described in [Specifying the Script Output Method, on page 537](#).

3. Synthesize your design.

The tool automatically decrypts the protected IP and synthesizes it with the other unencrypted files in the design. During synthesis, the IP is treated as a black box and you cannot view its contents in HDL Analyst views.

After synthesis, the tool generates one output netlist. The IP is re-encrypted and included in this netlist. The rest of the netlist is not encrypted and can be read. The IP is not included in any simulation netlists that are written out; it is treated as a black box.

See [Encryption and Decryption, on page 525](#) for a flow diagram of this process.

CHAPTER 13

Optimizing Processes for Productivity

This chapter covers topics that can help the advanced user improve productivity and interoperability with other tools. It includes the following:

- [Using Batch Mode](#), on page 558
- [Working with Tcl Scripts and Commands](#), on page 564
- [Automating Flows with synhooks.tcl](#), on page 571

Using Batch Mode

Batch mode is a command-line mode in which you run scripts from the command line. You might want to set up multiple synthesis runs with a batch script. You can run in batch mode if you have a floating license, but not with a node-locked license.

Batch scripts are in Tcl format. For more information about Tcl syntax and commands, see [Working with Tcl Scripts and Commands](#), on page 564.

This section describes the following operations:

- [Running Batch Mode on a Project File](#), on page 558
- [Running Batch Mode with a Tcl Script](#), on page 560
- [Queuing Licenses](#), on page 561

Running Batch Mode on a Project File

Use this procedure to run batch mode if you already have a project file set up. You can also run batch mode from a Tcl script, as described in [Running Batch Mode with a Tcl Script](#), on page 560.

1. Make sure you have a project file (`prj`) set up with the implementation options. For more information about creating this Tcl file, see [Creating a Tcl Synthesis Script](#), on page 566.
2. From a command prompt, go to the directory where the project files are located, and type one of the following, depending on which product you are using:

```
synplify_pro -batch project_file_name.prj
```

The software runs synthesis in batch mode. Use absolute path names or a variable instead of a relative path name.

If the `-tclcmd` switch is used, synthesis will not automatically run. To make synthesis run, `project -run` must be added:

```
synplify_pro -batch myproj.prj -tclcmd "project -run"
```

The `-tclcmd` switch specifies the tcl commands to be executed before the synthesis starts. To run a constraint check before synthesis:

```
synplify_pro -batch myproj.prj -tclcmd "project -run constraint_-\ncheck"
```

The `-tclcmd` switch also allows the synthesis results path to be changed.

```
synplify_pro -batch "D:/tests/myproj.prj" -tclcmd "set_option\n-result_file \"./impl1/test.edif\"; project -run"
```

The software returns the following codes after the batch run:

- 0 - OK
 - 2 - logical error
 - 3 - startup failure
 - 4 - licensing failure
 - 5 - batch not available
 - 6 - duplicate-user error
 - 7 - project-load error
 - 8 - command-line error
 - 9 - Tcl-script error
 - 20 - graphic-resource error
 - 21 - Tcl-initialization error
 - 22 - job-configuration error
 - 23 - parts error
 - 24 - product-configuration error
 - 25 - multiple top levels
3. If there are errors in the source files, check the standard output for messages. On Linux systems, this is generally the monitor; on Windows systems, it is the `stdout.log` file.
 4. After synthesis, check the `resultFile.srr` log file for error messages about the run.

Running Batch Mode with a Tcl Script

The following procedure shows you how to create a Tcl batch script for running synthesis. If you already have a project file set up, use the procedure described in [Running Batch Mode on a Project File, on page 558](#).

1. Create a Tcl batch script. See [Creating a Tcl Synthesis Script, on page 566](#) for details.
2. Save the file with a `tcl` extension to the directory that contains your source files and other project files.
3. From a command prompt, go to the directory with the files and type one of the following as appropriate:

```
synplify_pro -batch Tcl_script.tcl
```

The software runs synthesis in batch mode. The synthesis (compilation and mapping) status results and errors are written to the log file *result-File.srr* for each implementation. The synthesis tool also reports success and failure return codes.

4. Check for errors.
 - For source file or Tcl script errors, check the standard output for messages. On Linux systems, this is generally the monitor in addition to the `stdout.log` file; on Windows systems, it is the `stdout.log` file.
 - For synthesis run errors, check the *resultFile.srr* log file. The software uses the following error codes:
 - 0 - OK
 - 2 - logical error
 - 3 - startup failure
 - 4 - licensing failure
 - 5 - batch not available
 - 6 - duplicate-user error
 - 7 - project-load error
 - 8 - command-line error
 - 9 - Tcl-script error
 - 20 - graphic-resource error
 - 21 - Tcl-initialization error
 - 22 - job-configuration error
 - 23 - parts error
 - 24 - product-configuration error
 - 25 - multiple top levels

Queuing Licenses

A common problem when running in batch mode is that the run fails because all of the available licenses are in use. License queuing allows a batch run to wait for the next available license when a license is on the server but not immediately available.

You can specify either blocking or non-blocking queuing. With blocking-style queuing, the tool waits until a license becomes available; with non-blocking-style queuing, the tool waits the specified length of time for a license to become available.

You can also queue DesignWare IP licenses, so that they can be used as they become available.

For details, see the following:

- [Queuing Considerations](#), on page 561
- [Queuing Licenses](#), on page 561

Queuing Considerations

Consider these points when using queuing:

- A blocking-style queuing is used; license checkout does not exit until a license becomes available.
- There is no maximum wait time; once initiated, the tool can wait indefinitely for a license.
- If the server shuts down while the tool is waiting, a checkout failure is reported.
- When two licenses are required, queuing waits only until the first license becomes available (and not the second) to avoid holding a license unnecessarily.

Queuing Licenses

The following procedure describes how to specify blocking-style or non-blocking style queuing for synthesis licenses. You can specify the licensed features for queuing in an environment variable or directly in batch mode.

1. Specify the list of licensed features you want to queue, using either of the following methods:

- Set the `toolName_LICENSE_TYPE` environment variable to the features you want. For example:

```
SYNPLIFYPRO_LICENSE_TYPE=synplifypro:synplifypro_lattice
```

- Specify a list of features to wait for using the `-batch`, `-license_wait` and `-license_type` options. For example:

```
synplify_pro -batch -license_wait -license_type  
synplifypro:synplifypro_lattice myProject.prj
```

See [synplify_pro](#), on page 143 in the *Command Reference* for syntax details.

2. To enable blocking-style queuing, do one of the following:

- Set environment variable `toolName_LICENSE_WAIT=1` (`toolName` is the name of the FPGA synthesis tool).
- In batch mode, include a `-license_wait` command-line argument, as shown in the following examples:

```
synplify_pro -batch -license_wait Tcl_script.tcl
```

With blocking-style queuing enabled, the tool waits until the requested license becomes available. It generates the following message in the `stdout.log` or the Tcl window:

```
Waiting for license: toolName
```

3. To enable non-blocking-style queuing, do either of the following:

- Set environment variable `toolName_LICENSE_WAIT=waitTime` (`toolName` is the name of the FPGA synthesis tool and `waitTime` is the maximum wait time in seconds). For example:

```
SYNPLIFYPRO_LICENSE_WAIT=180
```

The `waitTime` value determines the maximum wait time, in seconds:

WaitTime Value	Queuing Behavior
Undefined or 0	Queuing off
1	Queuing on; wait indefinitely
>1	Queuing on; wait up to the specified number of seconds

- Include a `-license_wait waitTime` command-line argument when launching batch mode as shown in the following examples:

```
synplify_pro -batch -license_wait waitTime Tcl_script.tcl
```

When non-blocking-style queuing is enabled, the tool waits up to the maximum time limit specified for the license to become available. The tool generates the following message in `stdout.log` or the Tcl window:

```
Waiting up to n seconds for license: toolName
```

Working with Tcl Scripts and Commands

The software uses extensions to the popular Tcl (Tool Command Language) scripting language to control synthesis and for constraint files. See the following for more information:

- [Using Tcl Commands and Scripts](#), on page 564
- [Generating a Job Script](#), on page 565
- [Setting Number of Parallel Jobs](#), on page 565
- [Creating a Tcl Synthesis Script](#), on page 566
- [Using Tcl Variables to Try Different Clock Frequencies](#), on page 568
- [Using Tcl Variables to Try Several Target Technologies](#), on page 569
- [Running Bottom-up Synthesis with a Script](#), on page 570

You can also use synhooks Tcl scripts, as described in [Automating Flows with synhooks.tcl](#), on page 571.

Using Tcl Commands and Scripts

1. To get help on Tcl syntax, do any of the following:
 - Refer to the online help (Help->Tcl Help) for general information about Tcl syntax.
 - Refer to the *Command Reference Manual* for information about the synthesis commands.
 - Enter help * in the Tcl window for a list of all the Tcl synthesis commands.
 - Enter help *commandName* in the Tcl window to see the syntax for an individual command.
2. To run a Tcl script, do the following:
 - Create a Tcl script. Refer to [Generating a Job Script](#), on page 565 and [Creating a Tcl Synthesis Script](#), on page 566.
 - Run the Tcl script by either entering source *Tcl_scriptfile* in the Tcl script window, or by selecting File->Run Tcl Script, selecting the Tcl file, and clicking Open.

The software runs the selected script by executing each command in sequence. For more information about Tcl scripts, refer to the following sections.

Generating a Job Script

You can record Tcl commands from the interface and use it to generate job scripts.

1. In the Tcl script window, enter recording `-file logfile` to write out a Tcl log file.
2. Work through a synthesis session.

The software saves the commands from this session into a Tcl file that you can use as a job script or as a starting point for creating other Tcl files.

For the command syntax, see [recording, on page 101](#) in the Command Reference Manual.

Setting Number of Parallel Jobs

You can set the maximum number of parallel jobs by setting a variable in the `ini` file, by defining a Tcl variable or specifying the maximum number in the GUI.

1. To set the maximum number of parallel jobs in the `ini` file, do the following:
 - Open the `ini` file for the synthesis tool. For example, `synplify_pro.ini`.
 - Add the `MaxParallelJobs` variable to the `ini` file, as follows:

```
[JobSetting]
MaxParallelJobs=<n>
```

The tool uses the `MaxParallelJobs` value from the `ini` file as the default for both the UI (Project->Options) and batch mode. This value remains in effect until you reset it in the `ini` file or from the GUI, as described in the next step. To locate this configuration and initialization file (`ini`), see [Input Files, on page 144](#).

2. To set or change the maximum number of parallel jobs from the GUI, do the following:
 - Select Project->Options->Configure Parallel or Compile Point Process.
 - Set the value you want in the Maximum number of parallel synthesis jobs field, and click OK. This field shows the current `ini` value, but you can reset it, and it will remain in effect until you change it again. The value you set is saved to the `ini` file.
3. To set a Tcl variable for the maximum number of parallel jobs, do the following:
 - Determine where you are going to define the variable. You can do this in the project file, or a Tcl file, or you can type it in the Tcl window. If you specify it in a Tcl file, you must source the file. If you specify it in the Tcl window, the tool does not save the value and it will be lost when you end the current session.
 - Specify the `max_parallel_jobs` variable with the `set_option` Tcl command:

```
set_option -max_parallel_jobs value
```

The tool applies the `max_parallel_jobs` *value* specified to all project files and their respective implementations. This is a global option. The maximum number of parallel jobs remains in effect until you specify a new value. This new value takes effect immediately, going forward. However, when you set this option from the Tcl command window, the `max_parallel_jobs` value is not saved and will be lost when you exit the application.

Creating a Tcl Synthesis Script

Tcl scripts are text files with a `tcl` extension. You can use the graphic user interface to help you create a Tcl script. Interactive commands that you use actually execute Tcl commands, which are displayed in the Tcl window as they are run. You can copy the command text and paste it into a text file that you build to run as a Tcl script. For example:

```
add_file prep2.v
set_option -technology LIFCL
set_option -part LIFCL_40
set_option -package CSBGA289C

project -run
```

The following procedure covers general guidelines for creating a synthesis script.

1. Use a text file editor or select File->New, click the Tcl Script option, and type a name for your Tcl script.
2. Start the script by specifying the project with the `project -new` command. For an existing project, use `project -load project.prj`.
3. Add files using the `add_file` command. The files are added to their appropriate directories based on their file name extensions (see [add_file](#), on page 28). Make sure the top-level file is last in the file list:

```
add_file statemach.vhd
add_file rotate.vhd
add_file memory.vhd
add_file top_level.vhd
add_file design.fdc
```

For information on constraints and vendor-specific attributes, see [Using a Text Editor for Constraint Files \(Legacy\)](#), on page 49 for details about constraint files.

4. Set the design synthesis controls and the output:
 - Use the `set_option` command for setting implementation options and vendor-specific controls as needed. See the appropriate vendor chapter in the *Command Reference Manual* for details.
 - Set the output file information with `project -result_file` and `project -log_file`.
5. Set the file and run options:
 - Save the project with a `project -save` command
 - Run the project with a `project -run` command
 - Open the RTL and Technology views:

```
open_file -rtl_view
open_file -technology_view
```

6. Check the syntax.
 - Check case (Tcl commands are case-sensitive).
 - Start all comments with a hash mark (#).
 - Always use a forward slash (/) in directory and pathnames, even on the Windows platform.

Using Tcl Variables to Try Different Clock Frequencies

To create a single script for multiple synthesis runs with different clock frequencies, you need to create a Tcl variable for the different settings you want to try. For example, you might want to try different target technologies.

1. To create a variable, use this syntax:

```
set variable_name {  
    first_option_to_try  
    second_option_to_try  
    ...}
```

2. Create a foreach loop that runs through each option in the list, using the appropriate Tcl commands. The following example shows a variable set up to synthesize a design with different frequencies. It also creates a separate log file for each run.

```
set try_freq {  
    85.0  
    90.0  
    92.0  
    95.0  
    97.0  
    100.0  
}  
  
foreach frequency $try_freq {  
    set_option -frequency $frequency  
    project -log_file $frequency.srr  
    project -run}
```

Set of frequencies to try

Foreach loop

Tcl commands that set the frequency, create separate log files for each run, and run synthesis

The following code shows the complete script:

```
project -load design.prj  
set try_these {  
    20.0  
    24.0  
    28.0  
    32.0  
    36.0  
    40.0  
}
```

```
foreach frequency $try_these {  
    set_option -frequency $frequency  
    project -log_file $frequency.srr  
    project -run  
    open_file -edit_file $frequency.srr  
}
```

Using Tcl Variables to Try Several Target Technologies

This technique used here to run multiple synthesis implementations with different target technologies is similar to the one described in [Using Tcl Variables to Try Different Clock Frequencies, on page 568](#). As in that section, you use a variable to define the target technologies you want to try.

1. Create a variable called `try_these` with a list of the technologies.

```
set try_these {  
  
    LIFCL LFD2NX # list of technologies  
}
```

2. Add a `foreach` loop that creates a new implementation for each technology and opens the RTL view for each implementation.

```
foreach technology $try_these {  
    impl -add  
    set_option -technology $technology  
    project -run -fg  
    open_file -rtl_view  
}
```

The following code example shows the script:

```
# Open a new project, set frequency, and add files.
project -new
set_option -frequency 33.3
add_file -verilog D:/test/simpletest/prep2_2.v

# Create the Tcl variable to try different target technologies.
set try_these
    LIFCL LFD2NX # list of technologies
}

# Loop through synthesis for each target technology.
foreach technology $try_these {
    impl -add
    set_option -technology $technology
    project -run -fg
    open_file -rtl_view
}
```

Running Bottom-up Synthesis with a Script

To run bottom-up synthesis, you create Tcl scripts for individual logic blocks, and a script for the top level that reads the other Tcl scripts.

1. Create a Tcl script for each logic block. The Tcl script must synthesize the block. See [Creating a Tcl Synthesis Script, on page 566](#) for details.
2. Create a top-level script that reads the block scripts. Create the script with the with the `project -new` command.
3. Add the top-level data:
 - Add source and constraint files with the `add_file` command.
 - Set the top-level options with the `set_option` command.
 - Set the output file information with `project -result_file` and `project -log_file`.
 - Save the project with a `project -save` command.
 - Run the project with a `project -run` command.
4. Save the top-level script, and then run it using this syntax:

source *block_script.tcl*

When you run this command, the entire design is synthesized, beginning with the lower-level logic blocks specified in the sourced files, and then the top level.

Automating Flows with synhooks.tcl

This procedure provides the advanced user with callbacks that let you customize your design flow or integrate with other products. For example, you might use the callbacks to send yourself email when a job is done (see [Automating Message Filtering with a Tcl Script, on page 223](#)), or to automatically copy files to another location after mapping. You can use the callback functions to integrate with a version control system, or generate the files needed to run verification. The procedure is based on a file called `synhooks.tcl`, which contains the Tcl callbacks.

1. Copy the `synhooks.tcl` file from the `installDirectory/examples` directory to a new location.

You must copy the file to a new location so that it does not get overwritten by subsequent product installations and you can maintain your customizations from version to version. For example, copy it to `C:/work/synhooks.tcl`.

2. Define an environment variable called `SYN_TCL_HOOKS`, and point it to the absolute path location of the `synhooks.tcl` file. For example:

```
$SYN_TCL_HOOKS=/remote/rell/projects/MyProj/synhooks.tcl
```

3. Open the `synhooks.tcl` file in a text editor, and edit the file so that the commands reflect what you want to do. The default file contains examples of the callbacks, which provide you with hooks at various points of the design process.
 - Customize the file by deleting the ones you do not need and by adding your customized code to the callbacks you want to use. The following table summarizes the various design phases where you can use the callbacks and lists the corresponding functions. For details of the syntax, refer to [Tcl Script Examples, on page 280](#) in the *Reference Manual*.

Design Phase	Tcl Callback Function
Project Setup Callbacks	
Settings defaults for projects	proc syn_on_set_project_template
Creating projects	proc syn_on_new_project
Opening projects	proc syn_on_open_project
Closing projects	proc syn_on_close_project
Application Callbacks	
Starting the application after opening a project	proc syn_on_start_application
Exiting the application	proc syn_on_exit_application
Run Callbacks	
Starting a run. See Example: proc syn_on_start_run , on page 573.	proc syn_on_start_run
Ending a run	proc syn_on_end_run
Key Assignment Callbacks	
Setting an operation for Ctrl-F8. See Example: proc syn_on_press_ctrl_f8 , on page 573.	proc syn_on_press_ctrl_f8
Setting an operation for Ctrl-F9	proc syn_on_press_ctrl_f9
Setting an operation for Ctrl-F11	proc syn_on_press_ctrl_f11

- Save the file.

As you synthesize your design, the software automatically executes the function callbacks you defined at the appropriate points in the design flow.

Example: proc syn_on_start_run

The following code example gets selected files from the project browser at the start of a run:

```
proc syn_on_start_run {compile c:/work/prep2.prj rev_1} {  
    set sel_files [get_selected_files -browser]  
    while {[expr [llength $sel_files] > 0]} {  
        set file_name [lindex $sel_files 0]  
        puts $file_name  
        set sel_files [lrange $sel_files 1 end]  
    }  
}
```

Example: proc syn_on_press_ctrl_f8

The following code example gets all the selected files from the project browser and project directory when the Ctrl-F8 key combination is pressed:

```
proc syn_on_press_ctrl_f8 {} {  
    set sel_files [get_selected_files]  
    while {[expr [llength $sel_files] > 0]} {  
        set file_name [lindex $sel_files 0]  
        puts $file_name  
        set sel_files [lrange $sel_files 1 end]  
    }  
}
```


CHAPTER 14

Improving Runtime

The following sections describe how to use multiprocessing to run parallel synthesis jobs and improve runtime. See [Multiprocessing With Compile Points](#), on page 576.

Multiprocessing With Compile Points

This procedure describes how to run multiprocessing on a design with compile points. For information about defining compile points, see [Chapter 11, *Working with Compile Points*](#).

To run compile points with multiprocessing:

1. If required, specify additional license types to use for multiprocessing, as described in [Using Different License Types for Multiprocessing, on page 578](#).

The tool runs four jobs in parallel per license, so additional licenses increase the number of jobs that can be run in parallel. The actual number of licenses used depends on certain factors. See [Specifying Licenses for Multiprocessing, on page 577](#) for an explanation.

2. Select Options->Configure Parallel or Compile Point Process and set the maximum number of jobs to run in parallel.

See [Setting Maximum Parallel Jobs, on page 576](#) for other ways to set this value.

3. Synthesize the design as usual.

The synthesis software runs multiple, independent compile point jobs simultaneously, providing additional runtime improvements for the logical compile point synthesis flows.

Setting Maximum Parallel Jobs

You can set maximum number of parallel jobs in the following ways:

- [Setting the MaxParallelJobs Variable in the ini File, on page 577](#)
- [Setting the max_parallel_jobs Tcl Option](#)

Setting the MaxParallelJobs Variable in the ini File

The maximum number of parallel jobs is set in the product `ini` file. The following commands are set in the *product.ini* file (for example, *synplify_pro.ini*):

```
[JobSetting]

MaxParallelJobs=<n>
```

The `MaxParallelJobs` value is used by the UI as well as in batch mode. This value is effective until you specify a new value. To change the number of parallel jobs you can run, use the Options->Configure Parallel or Compile Point Process command from the Project view menu. On the Configure Parallel or Compile Point Process dialog box, in the Maximum number of parallel synthesis jobs field you will see the current `ini` value. You can specify a new `MaxParallelJobs` value which is effective until you change it again. Once you click OK, the new value is saved in the `ini` file.

Setting the max_parallel_jobs Tcl Option

You can also manually set an override value for the maximum number of parallel jobs. To do this, use the Tcl command:

```
set_option -max_parallel_jobs numberJobs
```

You can choose to:

- Source the Tcl file containing this option.
- Add this option to the Project file.
- Set this option from the Tcl command window.

This `max_parallel_jobs` value is applied to all project files and their respective implementations. This is a global option. The maximum number of parallel jobs remains in effect until you specify a new value. This new value takes affect immediately going forward. However, when you set this option from the Tcl command window, the `max_parallel_jobs` value is not saved and will be lost when you exit the application.

Specifying Licenses for Multiprocessing

When you decide to run parallel synthesis jobs, additional licenses may be required for the compile point jobs. By default, four parallel jobs use one license. For example, if you set the Maximum number of parallel synthesis jobs to

12, the synthesis tool consumes one license to run 4 compile point jobs and can utilize the two additional licenses to run 8 more parallel jobs if they are available for your computing environment. Licenses are released as jobs complete, and then consumed by new jobs which need to run.

The actual number of licenses utilized depends on the following factors:

1. Synthesis software scheme for the compile point requirements used to determine the maximum number of parallel jobs or licenses a particular design tries to use.
2. Value set on the Configure Parallel or Compile Point Process dialog box.
3. Number of licenses actually available. You can use Help->Preferred License Selection to check the number of available license. If you need to increase the number of available licenses, you can specify multiple license types. For more information, see [Using Different License Types for Multiprocessing](#), on page 578.

Factors 1 and 3 can change during a single synthesis run. The number of jobs equals the number of licenses; which then equates to the lowest value of these three factors.

Using Different License Types for Multiprocessing

You can specify multiple license types to increase the total number of licenses available for multiprocessing. Use either of these methods to specify the license types:

- Use the `-licensetype` command line option when you execute your tool.

For example, suppose you have two `synplifypro` licenses, two `synplifypro_allvendor` licenses, and three `synplifypro_lattice` licenses. Type the following at the command line:

```
synplify_pro.exe -licensetype  
"synplifypro:synplifypro_allvendor:synplifypro_lattice"
```

- Use one of the following environment variables specified with the license type:

- SYNPLIFYPRO_LICENSE_TYPE

```
setenv SYNPLIFYPRO_LICENSE_TYPE=  
"synplifypro:synplifypro_allvendor:synplifypro_lattice"
```

Multiprocessing can access any of these license types for additional licenses.

CHAPTER 15

Clock Conversion

ASICs are often implemented in FPGAs for prototyping. The inherent differences between ASICs and FPGAs can make this conversion difficult. One particular source of problems can be attributed to the complex clocking circuitry of FPGAs that often includes a large number of gated and internally generated clocks. The Synopsys FPGA synthesis tools provide two features, *Gated Clock Conversion* and *Generated Clock Conversion*, to address the complex clocking schemes.

These features move the generated-clock and gated-clock logic from the clock pin of a sequential element to its enable pin. Relocating these clocks allows the sequential elements to be tied directly to the skew-free source clock and also reduces the number of clock sources in the design which frees up routing resources and expedites placement and routing. Dedicated FPGA clock trees are routed to every sequential device on the die and are designed with low skew to avoid hold-time violations. Using these global clock trees allows the programmable routing resources of the FPGA to be used primarily for logic interconnect and simplifies static timing analysis because checks for hold-time violations based on minimum delays are unnecessary.

For details, refer to the following topics:

- [Working with Gated Clocks](#), on page 582
- [Optimizing Generated Clocks](#), on page 613

Working with Gated Clocks

This section first describes the gated-clock solution. The information is organized into the following sections:

- [Obstacles to Conversion](#), on page 584
- [Prerequisites for Gated Clock Conversion](#), on page 585
- [Defining Clocks Properly](#), on page 587
- [Synthesizing a Gated-Clock Design](#), on page 592
- [Accessing the Clock Conversion Report](#), on page 593
- [Analyzing the Clock Conversion Report](#), on page 594
- [Interpreting Gated Clock Error Messages](#), on page 596
- [Disabling Individual Gated Clock Conversions](#), on page 600
- [Integrated Clock Gating Cells](#), on page 600
- [Using Gated Clocks for Black Boxes](#), on page 605
- [OR Gates Driving Latches](#), on page 606
- [Obstructions to Optimization](#), on page 607
- [Unsupported Constructs](#), on page 608
- [Other Potential Workarounds to Solve Clock-Conversion Issues](#), on page 609
- [Restrictions on Using Gated Clocks](#), on page 609

The clock conversion solution separates the gating from the clock inputs, and combines individual clock trees on the dedicated FPGA global clock trees. The software logically separates the gating from the clock and routes the gating to the clock enables on the sequential devices, using the programmable routing resources of the FPGA. The software separates a clock net going through an AND, NAND, OR, or NOR gate by doing one of the following:

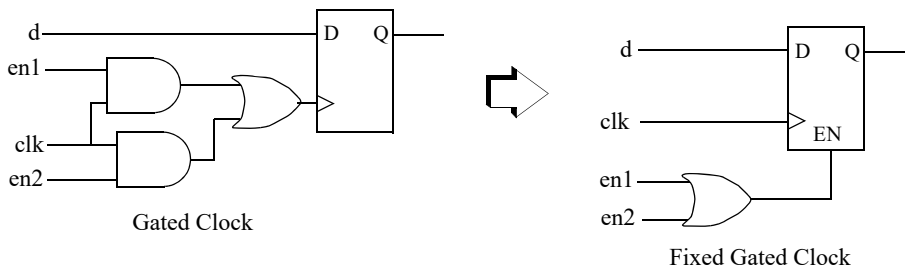
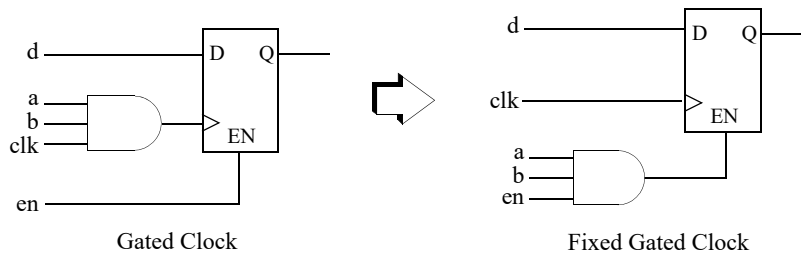
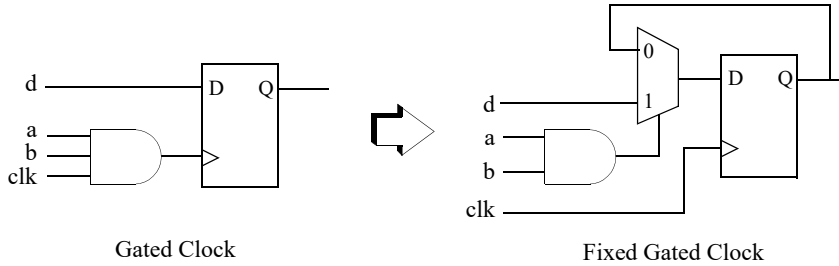
- Inserting a multiplexer in front of the input pin of the synchronous element and connecting the clock net directly to the clock pin
- Moving the gating from the clock input pin to the dedicated enable pin, when this pin is available.

The ungated or base clock is routed to the clock inputs of the sequential devices using the global FPGA clock resources. Typically, many gated clocks are derived from the same base clock, so separating the gating from the clock allows a single global clock tree to be used for all gated clocks that reference that base clock.

It is not always necessary to convert all clocks from an ASIC design – fitting the design into the FPGA device and meeting timing are the ultimate goals. However, some situations require unconverted clock structures to be modified such as:

- Clock structures with a large number of loads resulting in congestion issues
- Clock structures resulting in a large clock skew between the starting and ending clocks on a path that does not meet timing

The following figure illustrates several cases of how gated clocks are converted.¹



Obstacles to Conversion

The following is a summary of issues to look for that can prevent a clock structure from being converted. More details about each condition are described later.

- Are the clocks defined properly?
- Is the Clock Conversion option enabled?
- Does the clock structure contain instantiated cells?

- Is something in the clock logic blocking optimization?
- Are there any unsupported constructs?

Prerequisites for Gated Clock Conversion

For a gated clock to be converted successfully, the design must meet these requirements:

Condition	Description
Combinational logic only	The gated clock logic must consist only of combinational logic. A derived clock that is the output of a register is not converted.
Single base clock	Identify only one input to the combinational logic for the gated clock as a base clock. To identify a net as a clock, specify a period or frequency constraint for either the gate or the clock in the constraint file. This example defines the clk input as the base clock: <pre>create_clock -name {clk} -freq 10.000 -clockgroup default_clkgroup</pre>
Supported primitives	The sequential primitive clocked by the gated clock must be a supported object. The tools support gated-clock conversion for most sequential primitives. Black-box modules driven by gated clocks can be converted if special synthesis directives are used to define the clock and clock enable inputs to the black box. See Using Gated Clocks for Black Boxes , on page 605.
Correct logic format	See Correct Logic Format , on page 585 for an example of the correct logic format.

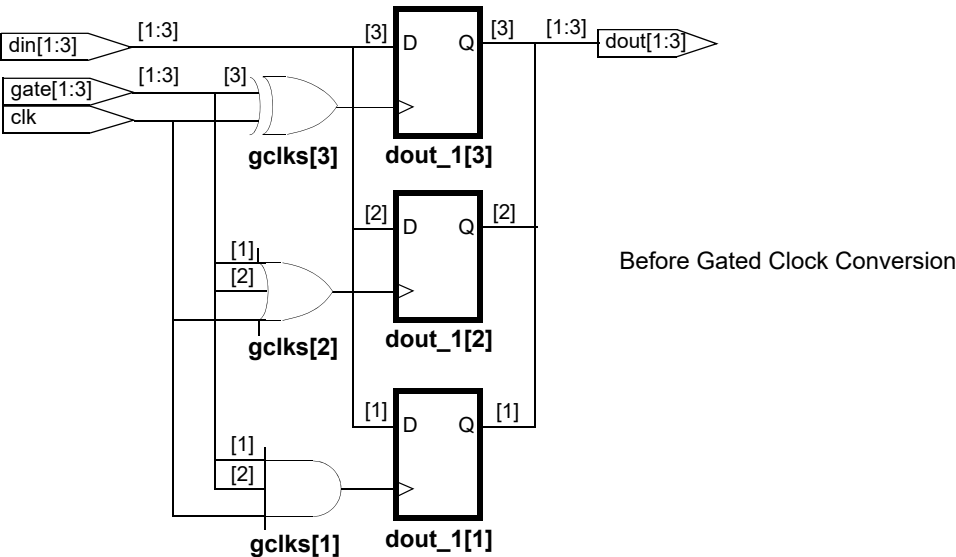
Correct Logic Format

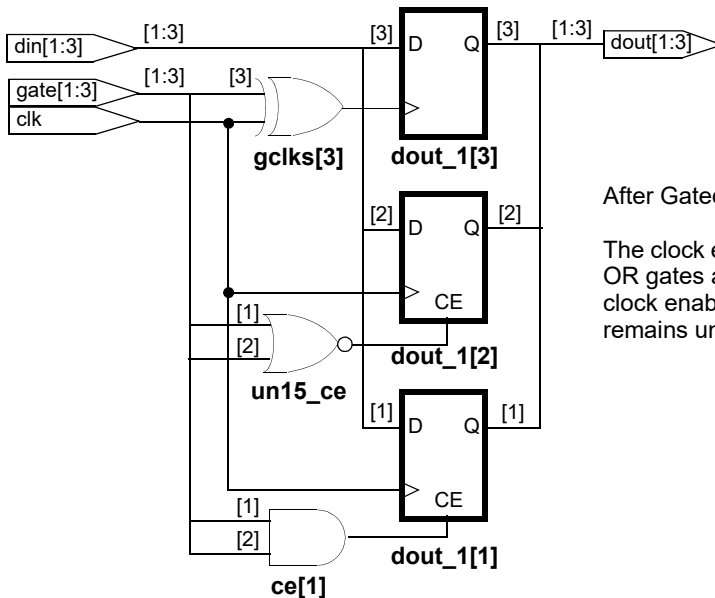
Specifically, the combinational logic for the gated clock must satisfy the following two conditions to have the correct format:

- For at least one set of gating input values, the value output for the gated clock must be constant and not change as the base clock changes.
- For at least one value of the base clock, changes in the gating input must not change the value output for the gated clock.

The correct logic format requirements are illustrated with the simple gates shown in the following figures. When the software synthesizes a design with the Gated Clocks option enabled, clock enables for the AND gate and OR gate are converted, but the exclusive-OR gate shown in the second figure is not converted. The following table explains.

AND gate gclks[1]	If either gate[1] or gate[2] is 0, then gclks[1] is 0, independent of the value of clk which satisfies the first condition. Also, if clk is 0, then gclks[1] is 0, independent of the values of gate[1] and gate[2] which satisfies the second condition. Because gclks[1] satisfies both conditions, it is successfully converted to the clock-enable format.
OR gate gclks[2]	If either gate[1] or gate[2] is 1, then gclks[2] is 1 independent of the value of clk which satisfies the first condition. Also, if clk is 1, then gclks[2] is 1 independent of the value of gate[1] or gate[2] which satisfies the second condition. Because gclks[2] satisfies both conditions, it is successfully converted to the clock-enable format.
Exclusive-OR gate gclks[3]	Irrespective of the value of gate[3], gclks[3] continues to toggle. The exclusive-OR function causes gclks[3] to fail both conditions which prevents gclks[3] from being converted.





After Gated Clock Conversion

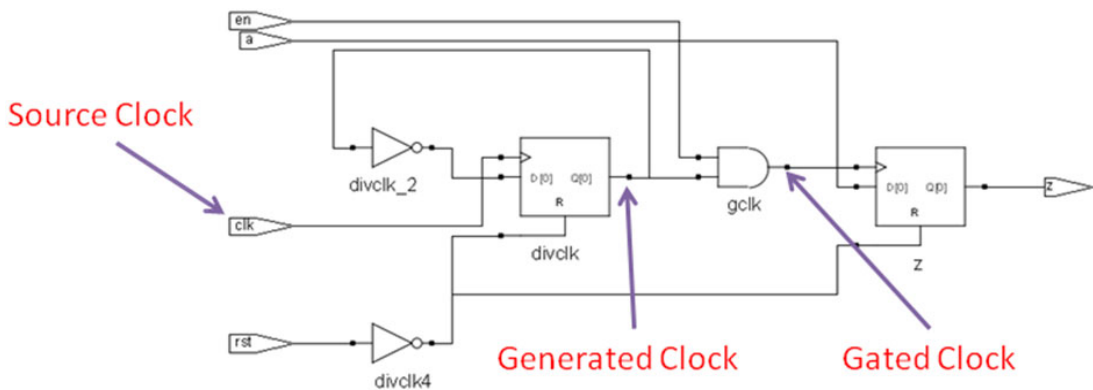
The clock enables for the AND and OR gates are converted, but the clock enable for the exclusive OR remains unchanged.

Defining Clocks Properly

The most common problems preventing clock conversion from occurring automatically are related to improperly defined clocks. When defining gated and generated clocks:

- Gated clocks should be defined at the nodes to be connected to the clock pins of the sequential cells.
- Use `create_generated_clock` constraints to define the relationship between generated clocks and their sources.
- If a clock circuit contains both a clock generator and a gating element, a `create_generated_clock` constraint is needed. Without this constraint, the tool does not know the relationship between the gated signal and the source clock.

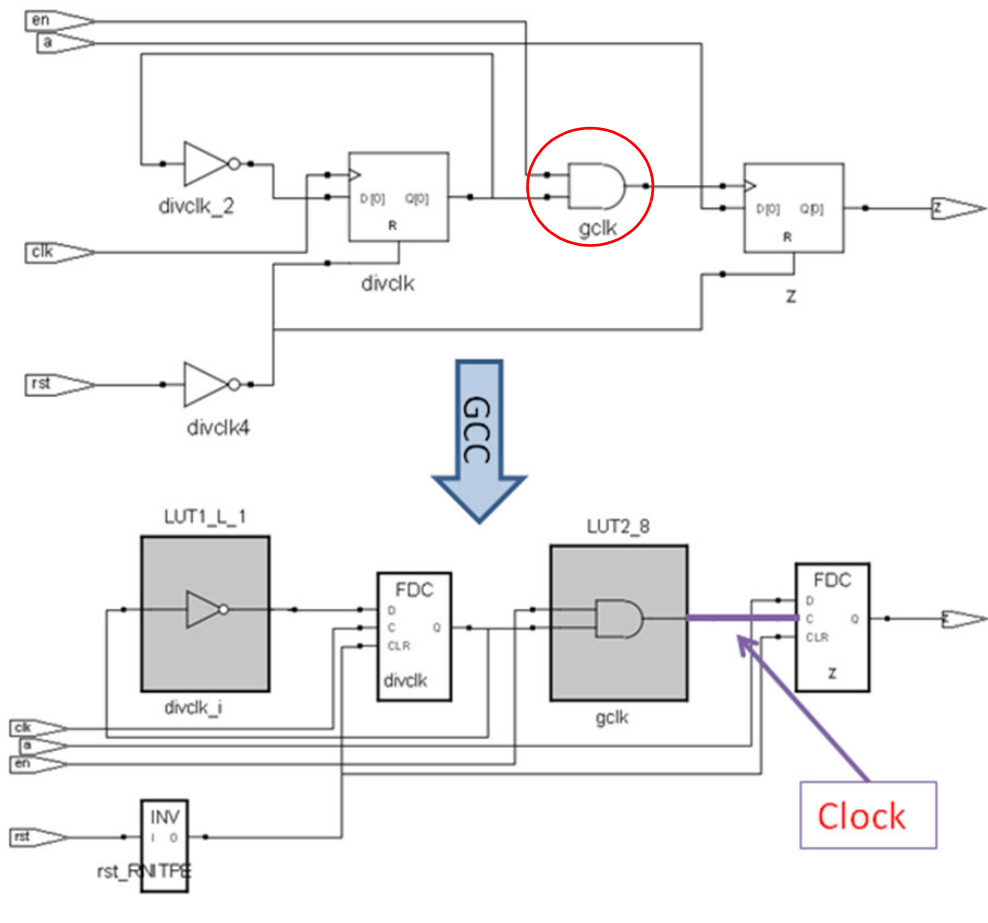
Defining Clocks Example



Using the above example, the first constraint applied is:

```
create_clock -name clk [get_ports clk] -period 10
```

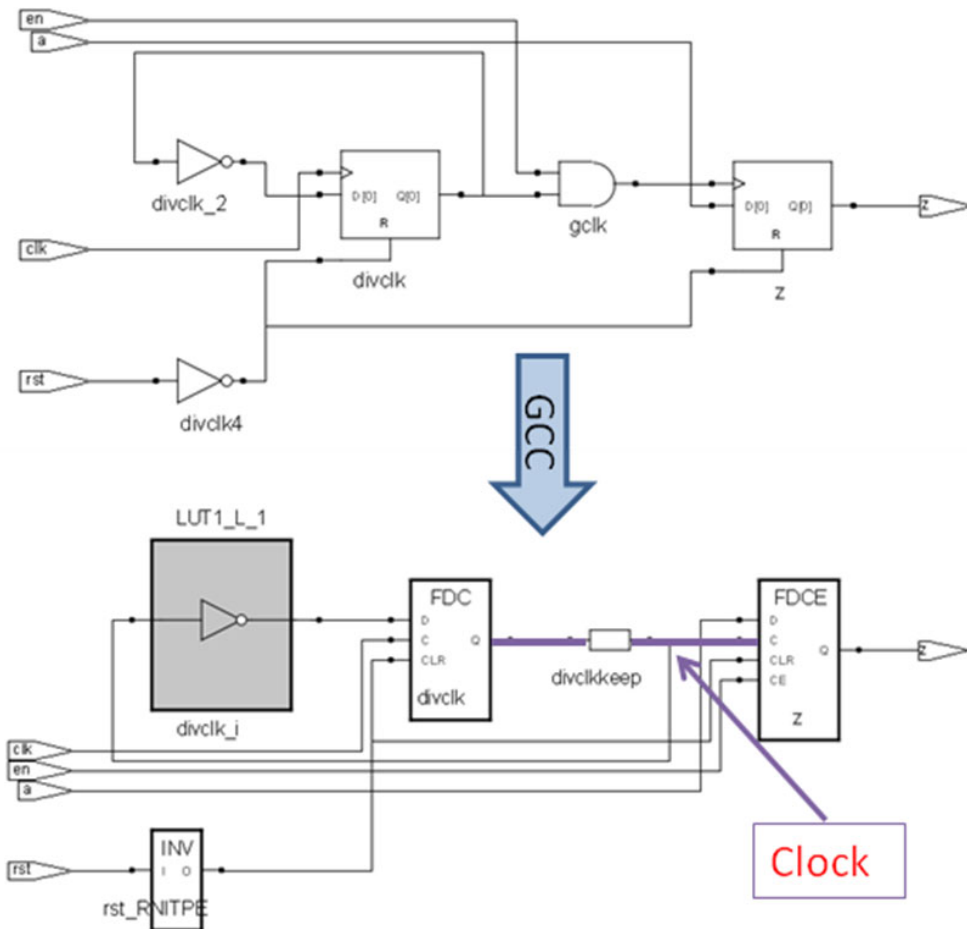
In the above constraint, `get_ports` identifies `clk` as a port. With only the source clock defined, the tool is unable to determine which input of the AND gate is the clock and which input is the enable. Accordingly, no conversion is performed as shown in the following schematic diagram.



Continuing with the example, a second constraint is added:

```
create_clock -name clk [get_ports clk] -period 10
create_clock -name divclk [get_nets {divclk}] -period 20
```

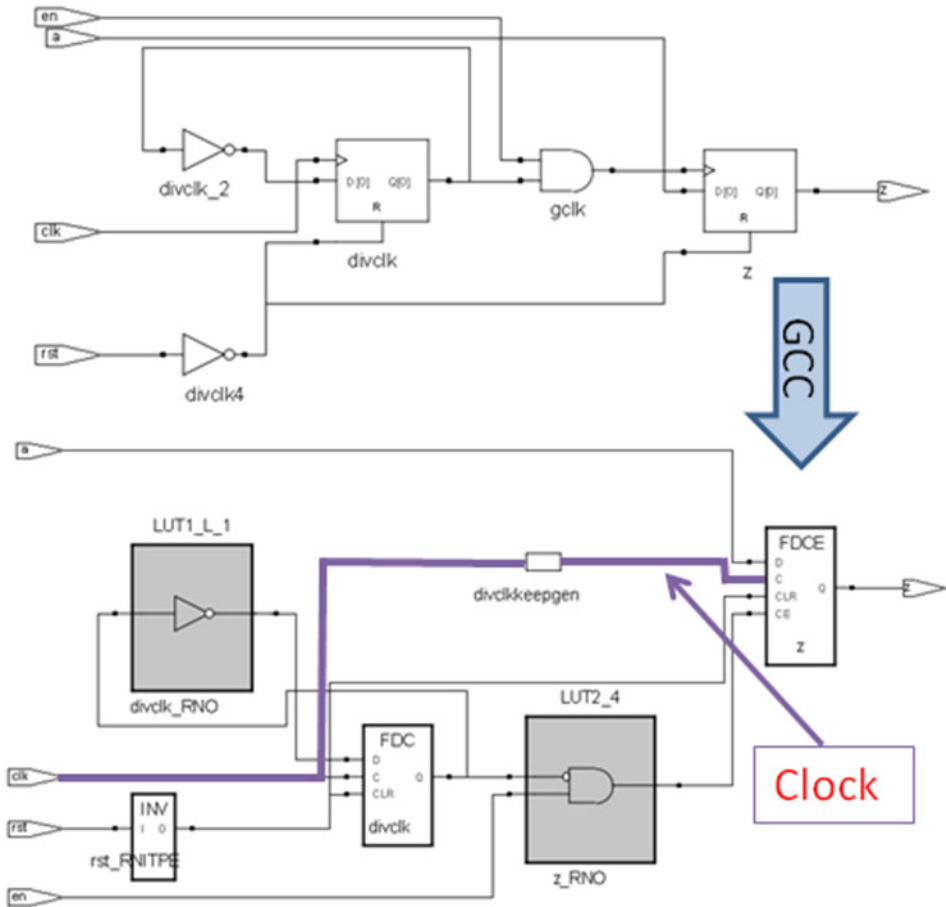
In the second constraint, `get_nets` identifies `divclk` as a net. With independent clock constraints defining the source and generated clocks, the tool now knows which input of the AND gate is the clock, but still does not know the relationship between the source and generated clocks. In this case, the clock gate and data register 'Z' are converted to an enable flip-flop (FDCE) which is connected to the generated clock as shown in the next schematic.



For the final example, the second clock constraint is replaced with a generated clock constraint.

```
create_clock -name clk [get_ports clk] -period 10
create_generated_clock -name divclk [get_nets {n:divclk}]
  -source [get_ports clk] -divide_by 2
```

The tool knows which input of the AND gate is the clock and the relationship between the source and generated clocks. For this case, the entire clock circuit is converted to an enable on the 'Z' register, and the 'Z' register clock pin is connected directly to the source clock as shown in the final schematic.



Internal Clocks

Applying internal clock constraints to output pins of inferred combinational logic (for example, an AND gate) is not supported. Apply clock constraints only to nets driven by logical output pins.

Only One Clock per Clock Pin Fan-in Cone

Every fan-in cone of the clock pin of a sequential device should have one defined clock. As long as one clock is defined, the clock can be traced forward through [unate](#) logic to the clock pins.

When no clock is defined:

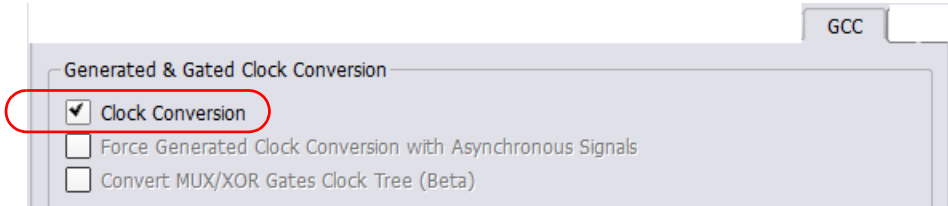
- The clock source is derived by tracing the clock pin back to the first non-trivial (multiple input) gate
- Because the tool does not know which input of a multiple-input cell to trace, a clock is inferred at the output of that cell
- Since inferred clocks were not defined by the user explicitly or implicitly, no conversion occurs

When more than one clock is defined, the tool does not know which clock to convert, and no conversion occurs.

Synthesizing a Gated-Clock Design

To synthesize a gated-clock design:

1. Make sure that the gated clocks have the correct logic format and satisfy the prerequisites for conversion. See [Prerequisites for Gated Clock Conversion, on page 585](#) for details.
2. If the gated clock drives a black box, specify the clock and the associated clock enable signal with using `syn_force_seq_prim`, `syn_isclock`, and `syn_gatedclk_en` directives. See [Using Gated Clocks for Black Boxes, on page 605](#) for details.
3. Make sure that the clock net has a constraint specified in the constraint file for the current implementation. If you do not specify an explicit constraint on the clock net or if you set a global frequency constraint, enabling clock conversion as described in the next step will not have any effect.
4. Enable the Clock Conversion option.
 - Select Project->Implementation Options.
 - On the GCC tab, click the Clock Conversion check box.



5. Synthesize the design. For gated clocks, the option converts qualified flip-flops, counters, latches, synchronous memories, and instantiated technology primitives. The software logically separates the gating from the clock and routes the gating to the clock enables on the sequential devices, using the programmable routing resources of the FPGA. The ungated base clock is routed to the clock inputs of the sequential devices using the global clock resources. Because many gated clocks are normally derived from the same base clock, separating the gating from the clock allows a single global clock tree to be used for all gated clocks that reference the same base clock.

See [Restrictions on Using Gated Clocks, on page 609](#) for additional information.

6. Check the results in the START OF CLOCK OPTIMIZATION REPORT section of the log file. See [Analyzing the Clock Conversion Report, on page 594](#) for an example of this report.

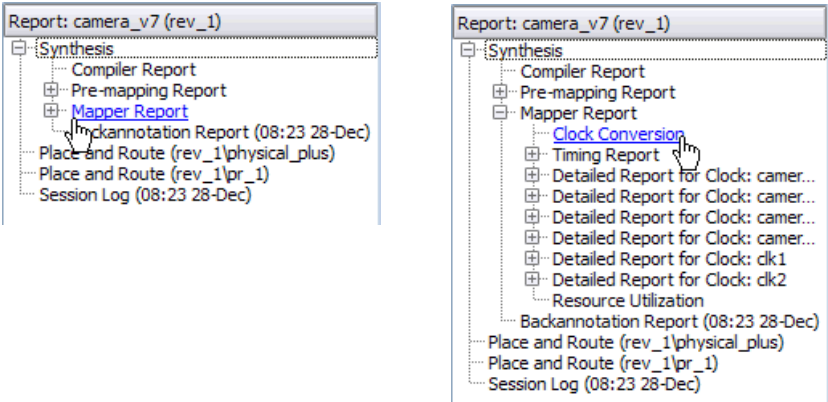
Accessing the Clock Conversion Report

The clock conversion report can be accessed from the log file or from the Project Status tab.

Log File Access

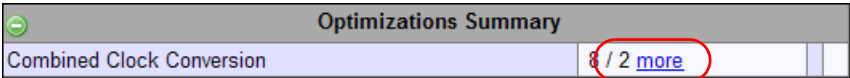
To access the clock conversion report directly in the log file:

1. Open the log file by clicking the View Log button.
2. In the browser, expand the Mapper Report entry by clicking the adjacent '+' sign and click on the Clock Conversion link to go directly to the clock conversion report. Note that the browser is only available when the View log file in HTML option is enabled.



Project Status Tab Access

To access the clock conversion report directly from the Project Status tab, click the more link in the Optimizations Summary section at the bottom of the display.



Analyzing the Clock Conversion Report

When clock conversion is enabled, a typical report resembles the following:

```
#### START OF CLOCK OPTIMIZATION REPORT ####
```

```
8 non-gated/non-generated clock tree(s) driving 434 clock pin(s) of sequential element(s)
2 gated/generated clock tree(s) driving 16 clock pin(s) of sequential element(s)
10 instances converted, 16 sequential instances remain driven by gated/generated clocks
```

3-line Summary

Non-Gated/Non-Generated Clocks				
Clock Tree ID	Driving Element	Drive Element Type	Fanout	Sample Instance
ClockId0003	wb_clk_i	port	215	camera_wishbone_if.dest_addr_cnt[31]
ClockId0004	bufg_inst2	BUFG	163	control_synchronization.hsyc_active_hi_sync
ClockId0005	ibufgds_inst	ISUBFG		control_synchronization.cam_int_sync
ClockId0006	bufg_inst3	BUFG		control_synchronization.cam_int_sync
ClockId0007	bufg_inst1	BUFG		control_synchronization.cam_int_sync
ClockId0008	clk1	port	2	reg_file_u1.ff_out[7]
ClockId0009	clk2	port	2	clk_top_inst.clk_tree2_c1.ff_clk_u1.clk_out
ClockId0010	clk3	port	1	clk_top_inst.reg_ff_u4.ff_out
Gated/Generated Clocks				
Clock Tree ID	Driving Element			Sample Instance
ClockId0001	clk_top_inst.clk_tree2_c1.com			clk_top_inst.reg_file_u2.ff_out[6]
ClockId0002	clk_top_inst.clk_tree3_c2.q			clk_top_inst.reg_file_u3.ff_out[7]

Non-gated/non-generated Clock Tree Table

Gated/generated Clock Tree Table

```
#### END OF CLOCK OPTIMIZATION REPORT ####
```

3-line Summary

The 3-line summary provides a quick snapshot of the design.

- The first line reports the number of *clean* clock trees (8 in the above report) and the number of driven clock pins (434).
- The next line reports the number of gated/generated clock trees (2 in the above report) and the number of driven clock pins (16).
- The last line reports the number of instances converted (10 in the above report) and the number of remaining instances driven by gated/generated clocks (16).

The following find command can be used to group the converted instances:

```
find -hier -inst * -filter @syn_gc_converted==1
```

Non-gated/non-generated Clocks Table

The non-gated/non-generated clocks section provides details of the *clean* clocks in the design and also includes any clock trees that never included gating logic. This section reports the following:

- Clock Tree ID – created for all instances in the clock tree that lead to the sample instance. Clicking on an ID opens a filtered technology view of the source clock.

- Driving Element – identifies the source of the driving clock.
- Drive Element Type – identifies the type of drive element such as a port or BUFG.
- Fanout – lists the clock fanout.
- Sample Instance – an instance within the clock tree.

The following find command can be used to identify all instances in the clock path:

```
find -hier -inst * -filter @syn_sample_clock_path==CKIDxxxx
```

If a clock is defined within the cone-of-logic for the gated clock, it is reported in this table.

Gated/generated Clocks Table

The gated/generated clocks section lists a sample failure within each clock tree in the design with an explanation of why the conversion failed (see the following section [Interpreting Gated Clock Error Messages, on page 596](#)). The gated/generated clocks section reports the following:

- Clock Tree ID – created for all instances in the clock tree that lead to the failed sample instance. Clicking on an ID opens a filtered technology view of the source clock.
- Driving Element – identifies the source of the driving clock.
- Drive Element Type – identifies the type of drive element such as a LUT.
- Fanout – lists the clock fanout.
- Sample Instance – an instance within the clock tree.

In the table, failures with common gating-control signals are only reported once. The software traverses back from the clock pin of the sample instance until it reaches the driving element at the point of the failure.

Interpreting Gated Clock Error Messages

The following table describes the gated clock conversion error messages reported in the Gated/Generated Clocks section of the clock conversion report.

Error Message	Explanation
Asynchronous set/reset mismatch prevents generated clock conversion	Unshared asynchronous signals have been detected between a FF-derived clock circuit and its sequential load.
Can't determine input clock driver	Self-explanatory. Specify the driver.
Clock conversion disabled	Gated clocks have been detected, but clock conversion is not enabled.
Clock propagation blocked by fixed hierarchy	Clock property has been blocked by a fixed hierarchy which prevents clock conversion from propagating upstream.
Clock propagation blocked by hard hierarchy	Clock property has been blocked by a hard hierarchy which prevents clock conversion from propagating upstream.
Clock propagation blocked by locked hierarchy	Clock property has been blocked by a locked hierarchy which prevents clock conversion from propagating upstream.
Clock propagation blocked by syn_keep	Clock property has been blocked by a syn_keep property which prevents clock conversion from propagating upstream.
Clock source is constant	Self-explanatory.
Combinational loop in clock network	Self-explanatory. Fix the combination loop.
FF-derived clock conversion disabled	FF-derived clocks have been detected, but clock conversion is not enabled.
Gating structure creates improper gating logic	Self-explanatory. Analyze and fix logic.
Illegal instance on clock path	Self-explanatory. Analyze and fix logic.
Inferred clock from port	Self-explanatory notification.
Input clock depends on output	Self-explanatory.
Latch gated by OR originally on clock tree	Self-explanatory.

Error Message	Explanation
Multiple clock inputs on sequential instance	Self-explanatory. Analyze and fix logic.
Multiple clocks on generating sequential element	Self-explanatory. Analyze and fix logic.
Multiple clocks on instance	Multiple clocks found feeding an instance of a clock tree.
Need declared clock or clock from port to derive clock from ff	Self-explanatory. Specify the clock.
No clocks found on inputs	No clocks found feeding an instance of a clock tree.
No generated or derived clock directive on output of sequential instance	Self-explanatory. Specify the clock.
No hierarchical driver	Self-explanatory. Specify the driver.
Signal from port	Self-explanatory.
Unconverted clock gate	Self-explanatory.
Unable to determine clock driver on net	Self-explanatory. Specify the clock driver.
Unable to determine clock input on sequential instance	Self-explanatory. Analyze and fix logic.
Unable to follow clock across hierarchy	Self-explanatory. Analyze and fix logic.
Unable to use latch as gated clock generator	Self-explanatory. Analyze and fix logic.

Checking the Log file

Many clock constraint problems can be identified by reviewing the Clock Summary table in the pre-mapping report section and the Performance Summary section of the log file shown below:

- Clock conversion is attempted on declared, generated, and derived clocks, and is never performed on inferred clocks or on the system clock.

- Declared, generated, and derived clocks can still have issues such as unsupported structures in their clock logic or incorrect constraints.
- Inferred clocks and the system clock are created when no clocks are defined in the fan-in logic of the clock pin of a sequential cell.

```
Performance Summary
*****
Worst slack in design: 8.300
```

Starting Clock	Slack	Clock Type	Clock Group
camera_open_source CLKOUT0_derived_clock_CLKIN1	997.947	derived	Inferred_clkgroup_1
camera_open_source CLKOUT1_derived_clock_CLKIN1	997.919	derived	Inferred_clkgroup_1
camera_open_source camera_clock_pad_i	998.073	inferred	Inferred_clkgroup_1
camera_open_source wb_clk_i	996.467	inferred	Inferred_clkgroup_0
clk1	8.447	declared	default_clkgroup
clk2	8.300	declared	default_clkgroup
clk3	NA	declared	default_clkgroup
en3_0	NA	declared	default_clkgroup
System	NA	system	system_clkgroup

Following compilation, the software issues warnings about inferred clocks or the system clock, that include the clock name and one of the registers driven by that clock to help the user locate the issue in the HDL Analyst. The following are examples of these warnings:

- @W: MT529: Found inferred clock *clkName* which controls *numClkPins* sequential elements including *instanceName*. This clock has no specified timing constraint which may prevent conversion of gated or generated clocks and may adversely impact design performance.
- @W: MT531: Found signal identified as System clock which controls *numClkPins* sequential elements including *instanceName*. Using this clock, which has no specified timing constraint, can prevent conversion of gated or generated clocks and can adversely impact design performance.

Disabling Individual Gated Clock Conversions

By default, enabling gated-clock conversion applies globally to a design. For critical paths, individual clocked elements can be expressly excluded from the gated-clock conversion by adding a `syn_keep` directive directly to the input clock net. The `syn_keep` attribute can be applied in the source code or in a constraint file. The following example shows a `syn_keep` attribute applied to the `gclk` net in a source file.

```
module and_gate (clk, en, a, z);
  input clk, en, a;
  output reg z;
  wire gclk /* synthesis syn_keep = 1 */;
  assign gclk = en & clk;

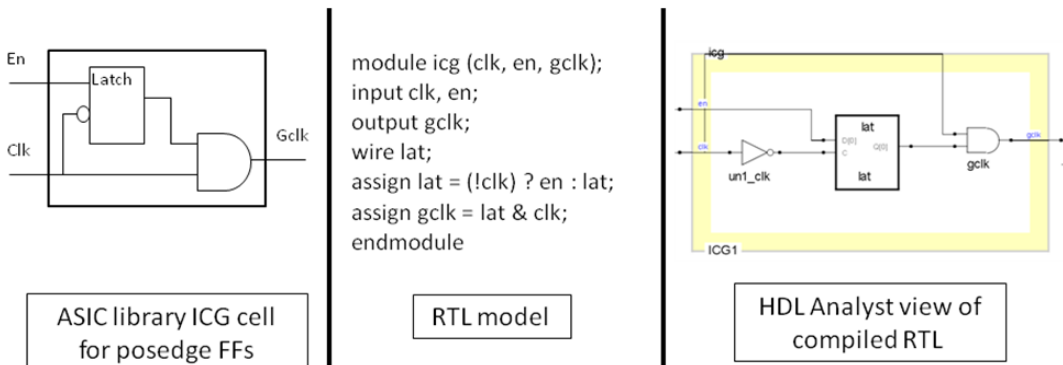
  always @(posedge gclk) z <= a;
endmodule
```

The corresponding entry in a constraint file is:

```
define_attribute {n:gclk} {syn_keep} {1}
```

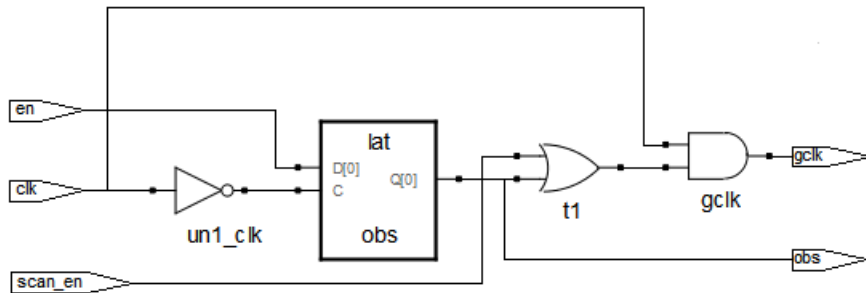
Integrated Clock Gating Cells

The Synopsys synthesis tools do not read in ASIC cell libraries. To support conversion of Integrated Clock Gating (ICG) cells, these cells must be modeled in RTL.



If there are ASIC-only pins on an ICG cell, such as a scan-enable input or an observation output, the pins must be removed to allow the clock conversion to occur.

The following example shows an ICG cell with ASIC-only pins and the corresponding Verilog source code.

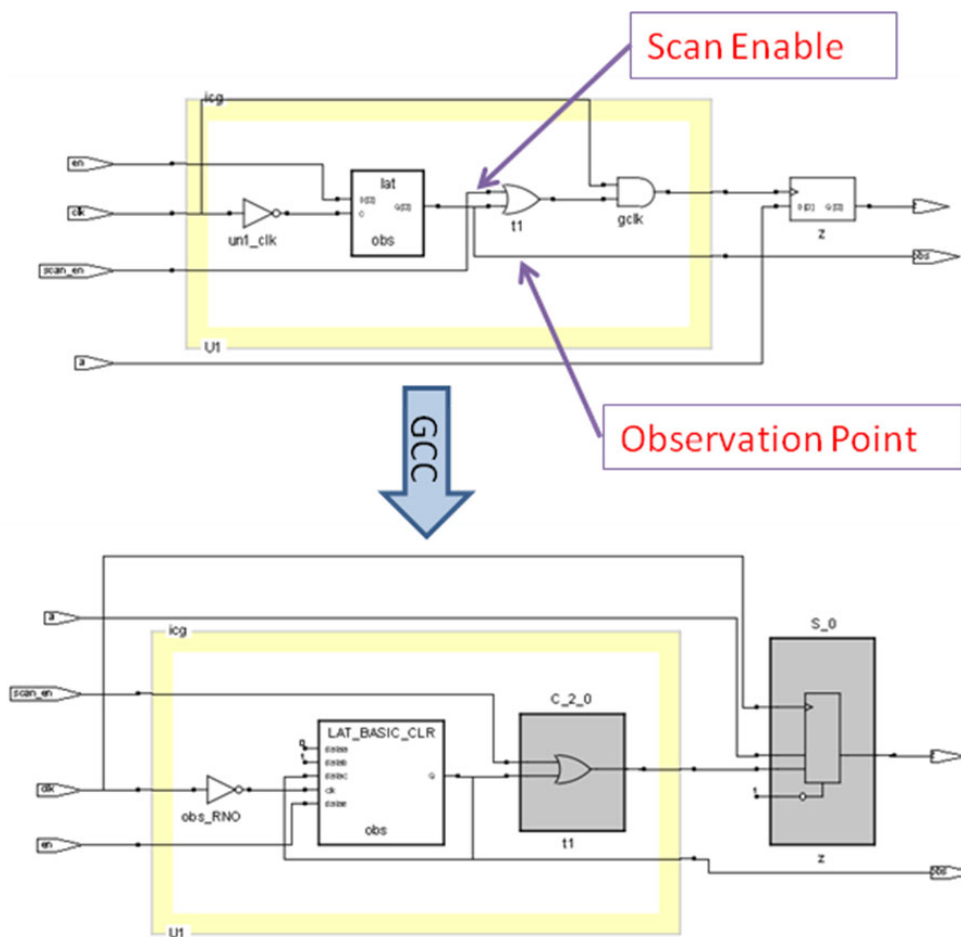


```

module icg (clk, en, scan_en, obs, gclk);
input clk, en, scan_en;
output obs, gclk;
wire t0, t1;
assign t0 = (!clk) ? en : t0;
assign obs = t0;
assign t1 = t0 | scan_en;
assign gclk = t1 & clk;
endmodule

```

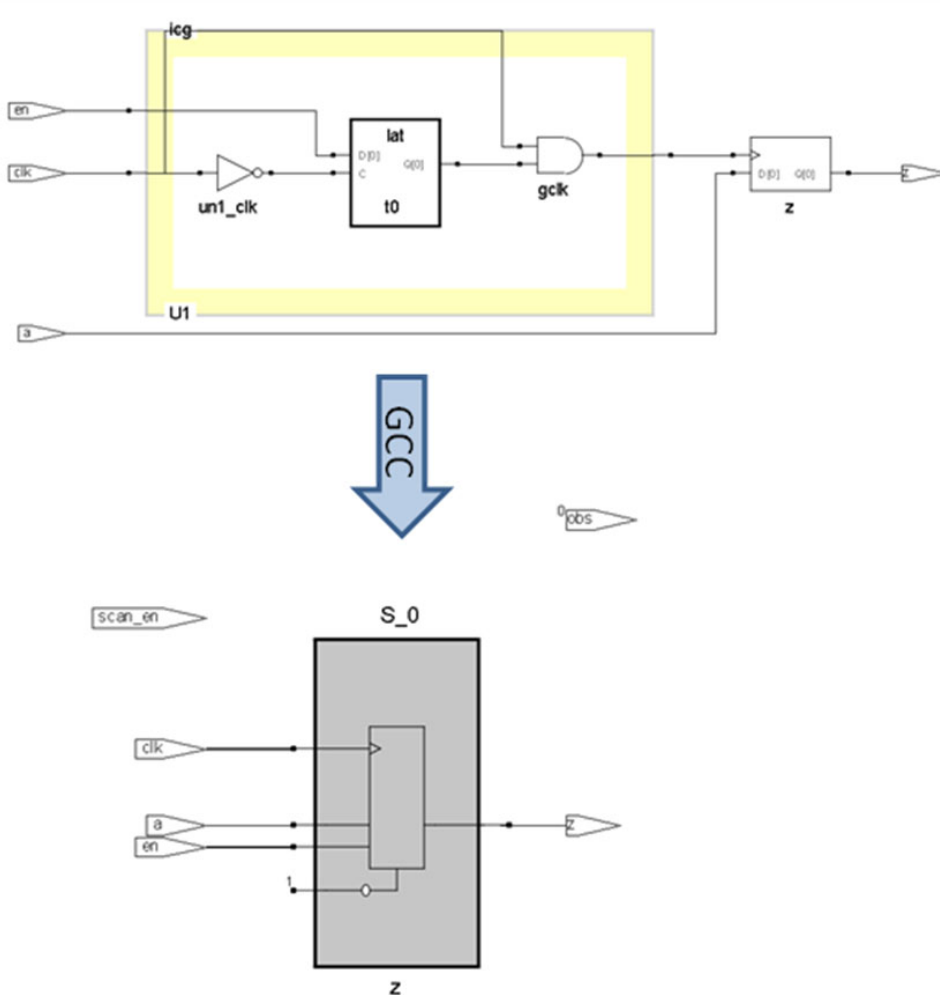
The presence of the scan control and observation point signals forces the tool to preserve intermediate signals in the clock-gating circuit that prevents it from being optimized. In this case, the data register is connected directly to the source clock, but the latch and OR gate for the ICG model remain as part of the enable logic as shown in the following schematic.



The following code simplifies the RTL model of the complex IGC cell and allows the desired gated-clock conversion to occur.

```
module icg (clk, en, scan_en, obs, gclk);
  input clk, en, scan_en;
  output obs, gclk;
  wire t0, t1;
  assign t0 = (!clk) ? en : t0;
  assign obs = 1'b0 /* t0 */;
  assign t1 = t0 | 1'b0 /* scan_en */;
  assign gclk = t1 & clk;
endmodule
```

In the modified code, the `scan_en` input, which is generally not used in an FPGA implementation, is replaced with a constant '0'. The `obs` observation point output, which normally is not required in an FPGA implementation, is also tied to a constant '0'. These changes remove the intermediate points as shown in the following schematic. Note that if no other changes are made to the design, this model still “fits” because the ports are unchanged. Also, the '0' tied to the `obs` output propagates to simplify downstream logic.

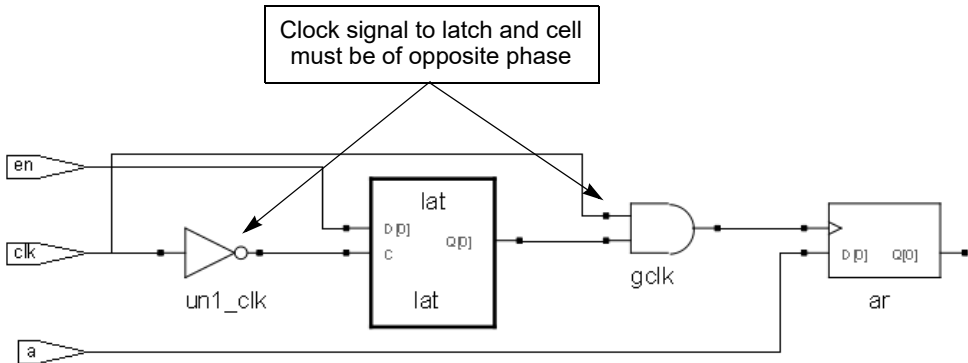


Incorrect Phase

When there is an error in the RTL model of an ICG cell such that an incorrect phase of the clock is connected to the latch:

- Only the AND gate is converted leaving the latch driving the enable pin of the data flip-flops.

- A warning is generated stating “Active phase of latch *latchName* prevents its use as a stability latch for clock gate *clockGateName*.”



Using Gated Clocks for Black Boxes

To convert gated clocks that drive black boxes, you must identify the clock and clock enable signal inputs to the black box using the `syn_force_seq_prim`, `syn_isclock`, and `syn_gatedclk_clock_en` directives. Refer to the *Attributes Reference Manual* for information about these directives. You assume responsibility for the functionality of the black-box content.

The following are Verilog and VHDL examples of a black box with the required directives specified.

Verilog

```
module bbe (ena, clk, data_in, data_out)
/* synthesis syn_black_box */
/* synthesis syn_force_seq_prim="clk" */
;
input clk
/* synthesis syn_isclock = 1 */
/* synthesis syn_gatedclk_clock_en="ena" */;
input data_in, ena;
output data_out;
endmodule
```

VHDL

```

library synplify;
use synplify.attributes.all;

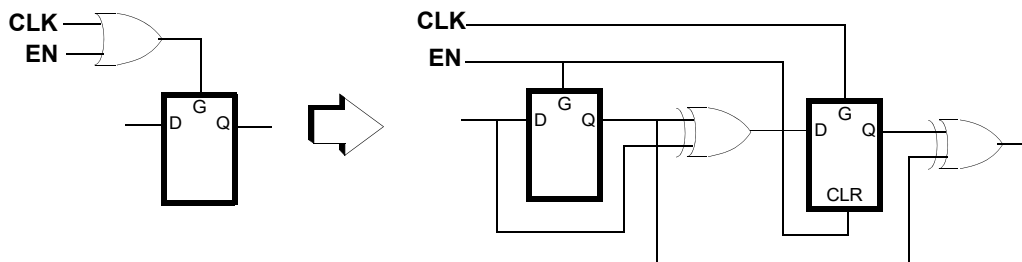
entity bbe is
    port (clk : in std_logic;
          en : in std_logic;
          data_in : in std_logic;
          data_out : out std_logic);
    attribute syn_isclock : boolean;
    attribute syn_isclock of clk : signal is true;
    attribute syn_gatedclk_clock_en : string;
    attribute syn_gatedclk_clock_en of clk : signal is "en";
end bbe;

architecture behave of bbe is
    attribute syn_black_box : boolean;
    attribute syn_force_seq_prim : string;
    attribute syn_black_box of behave : architecture is true;
    attribute syn_force_seq_prim of behave : architecture is "clk";
begin
end behave;

```

OR Gates Driving Latches

To prevent problems in timing analysis when a latch is being driven by the OR of a clock and an enable, gated-clock conversion is implemented using the logic shown in the following diagram.



As illustrated in the above diagram, the enable (EN) and clock (CLK) feed separate latches, and XOR gates are used to condition the inputs and outputs. The conversion masks the combinational path from the timing engine and duplicates the behavior of the original circuit. Conversion only occurs if the original latch is not driving a clock network.

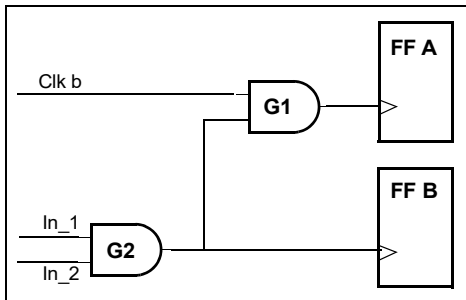
Obstructions to Optimization

The following optimization obstructions in the clock structure prevent conversion:

Inferred Clocks can Prevent Clock Conversions

Gated Clock Conversions

Beginning with the K-2015.09-SP1 release, the presence of inferred clocks can prevent the conversion of gated clocks (as the example illustrates below) or generated clocks. As a result, rerunning a design from a previous release may perform fewer conversions than previously reported.



In the example, `clk_b` is a declared clock and `In_1` and `In_2` are not declared as clocks. Previously, FF A could be a candidate for gated clock conversion. Now, this conversion cannot occur since there is also an inferred clock driving AND gate G1 (that is, the output of the AND gate G2 fed by `In_1` and `In_2` drives the clock of FF B).

“Keep_buffers” in the Clock Structure

Keep buffers are components that are added to the RTL view netlist to preserve specific nets. If a net in the clock circuitry is preserved by a keep buffer, it cannot be optimized away and can prevent conversion of the associated clock circuit. Keep buffers are the result of `syn_keep` attributes and constraints placed on nets such as false path, multi-cycle path, and other clock constraints. The following Tcl `find` command can be used to locate the keep buffers:

```
find -hier -view keepbuf
```

“Hard” Hierarchy Defined on a Clock Logic Block

A “hard” hierarchy attribute on a block containing a clock structure prevents conversion by forcing the interface of this block to remain unchanged. More specifically, if a hard hierarchy is between the clock logic and the data registers, the conversion cannot occur because the new signals for the enables cannot be added to the block interface. However, if all the clock logic is contained within the hard hierarchy, the gated clock conversion can occur. The following Tcl find command can be used to locate the hard hierarchies:

```
find -view * -filter (@syn_hier==hard)
```

Loads on Intermediate Nodes in the Clock Logic

Intermediate nodes in the clock logic should not have extra loads. For conversion to occur when extra loads are present, the RTL must be modified to separate the loads that drive clock pins of data registers from the other loads.

“Disable Sequential Optimizations” Project Option Prevents Latches from Being Removed

For conversion to occur, the Disable Sequential Optimizations option cannot be enabled.

Unsupported Constructs

The following are examples of constructs that prevent conversion from occurring.

- If the clock circuitry contains logic such that asynchronous set/reset signals can produce an active edge on the gated-clock line (without any change in the source clock), the circuit is not converted. Any active edges on the gated-clock output must be controlled by the source clock.
- By default, conversion only occurs if the registers in the generated-clock logic have the same asynchronous set/reset signals as the data registers that they control.
- If the wrong phase of the clock is connected to a latch in a gated-clock circuit, the following warning is reported:

```
Active phase of latch latchName prevents its use as a  
stability latch for clock gate clockGateName
```

Other Potential Workarounds to Solve Clock-Conversion Issues

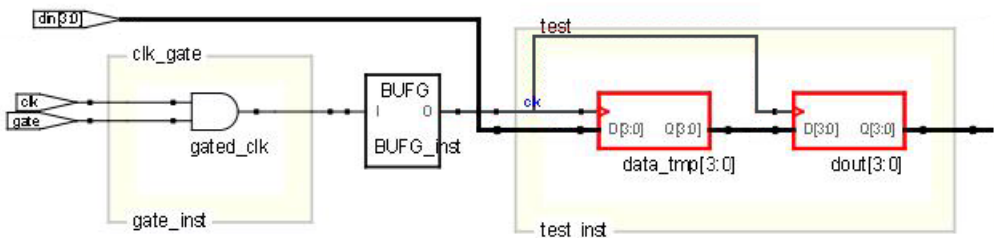
If the previous sections do not help resolve your issue, there are a few more things you can try:

- Use a `syn_direct_enable` attribute on one of the gating signals to force the conversion.
- Recode RTL to fit one of the cases allowed for automatic conversion.

Restrictions on Using Gated Clocks

The Clock Conversion option has the following restrictions for gated clocks:

- A `syn_keep` attribute attached to a net, which should preserve the net during optimization, is not honored by the Clock Conversion option.
- A global buffer, when instantiated on a gated clock, blocks gated-clock conversion from being performed (the buffer is viewed as the clock source by the downstream circuitry). In the circuit below, the `clk` input to `test_inst` remains gated.



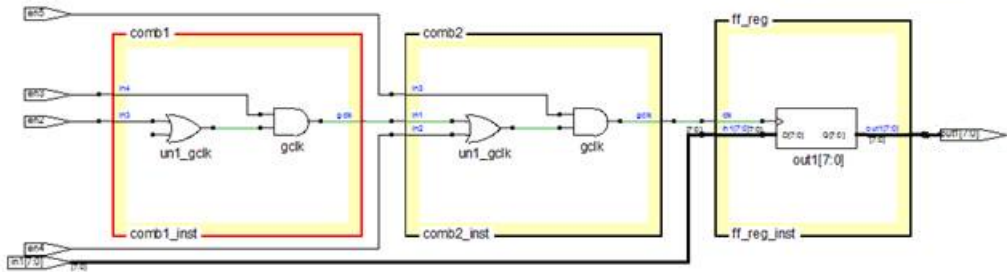
Similarly, defining a clock constraint at the output of a gated clock prevents conversion from continuing upstream.

Issues with Gated/Generated Clock Reporting

Several gated/generated clock reporting issues have been identified:

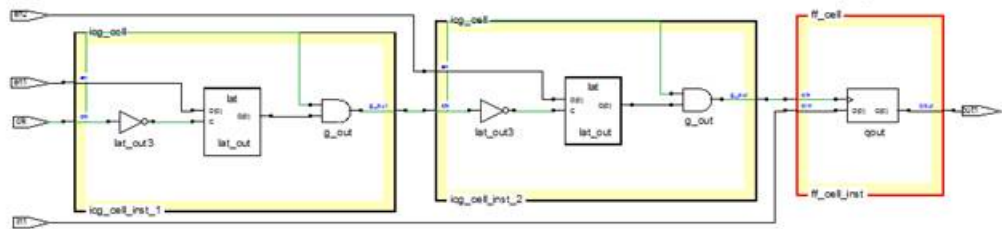
- When both inputs for XOR gates are defined as clocks, the software incorrectly reports the explanation as “No clocks found on input”. The correct explanation is “Improper clock gating structure.” For HAPs compatible devices, the correct explanation is “Multiple clocks found on inputs.”

- When both inputs for MUXs are defined as clocks, the software incorrectly reports “Multiple clocks on instances”. The correct explanation is “Improper clock gating structure”.
- Gated clocks might not be converted for logic from a hierarchy that uses `syn_hier=hard/fixed`. In the following example, the `comb1` hierarchy has this attribute. Although the combinational logic within the `comb2` hierarchy is outside the scope of `syn_hier=hard`, the enable logic of register, `out1` does not get converted.

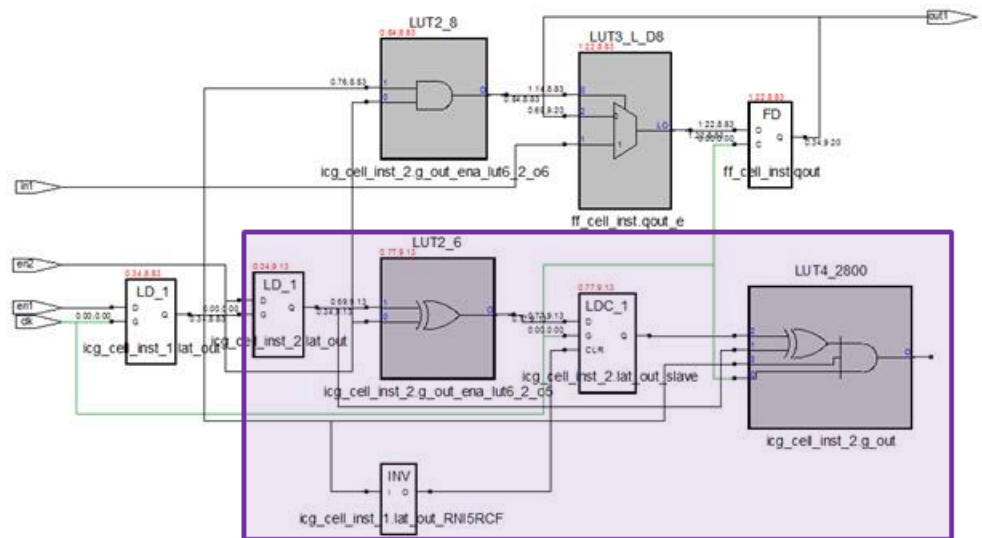


- For cascaded ICGs driving posedge flip-flops that use either compile points or the `syn_hier` attribute, the tool might produce a circuit with unused logic that does not get pruned away. See the shaded logic in the following figure. Depending on how the compile point or `syn_hier` boundaries are defined, the tool can report unused latches as unconverted, with the explanation “Unable to use latch as gated clock generator.”

RTL:



Post Mapping:



- The explanation “Unconverted clock gate” message indicates a gated/generated clock case that is not documented.

Clock Conversion for Multiple Input Clocks to OR Gate Latch

If multiple clocks are defined as inputs to an OR gate driving a latch, then clock conversion does not occur and the following message is generated: Multiple clocks on instance. If this message occurs, make sure that only one clock is input to the OR gate driving the latch.

Clock Conversions for the Traditional Flow

Clock conversions are handled in a traditional (non-distributed) synthesis flow as follows:

- Clock conversion runs at the `pre_map` stage, on compiler primitives.
- The names of referenced objects and the numeric values in the Clock Optimization report are defined with the conditions below:
 - Driving Element/Drive Element Type references compiler objects (e.g. MUX).
 - Clock trees with `n` number of instances and with the same inputs will be reported `n` times.
- ICG latch removal is performed in conjunction with clock conversion.
 - The ICG Latch Removal Summary is listed just before the Clock Optimization Report at the `pre-map` stage.
- ICGs that drive latches are supported and will be converted to registers.

When the `syn_hier=hard` attribute is applied to hierarchies that “cut” a gating cone of logic, the software does not prevent clock conversion to occur.

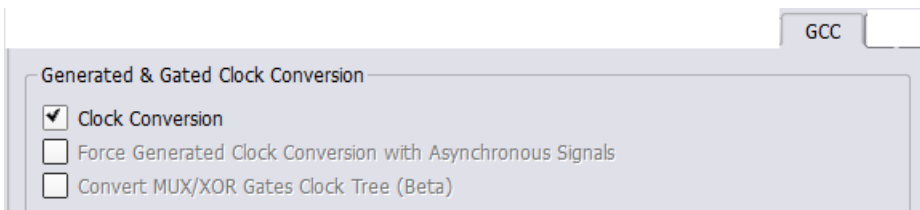
Optimizing Generated Clocks

When the Clock Conversion option is enabled, the generated-clock logic is replaced during synthesis with logic that uses the initial clock with an enable. With generated-clock optimization, the original circuit functionality is preserved while performance is improved by reducing clock skew. For more information, see the following topics:

- [Enabling Generated-Clock Optimization](#), on page 613
- [Conditions for Generated-Clock Optimization](#), on page 613
- [Generated-Clock Optimization Examples](#), on page 614

Enabling Generated-Clock Optimization

Generated-clock optimization is enabled by checking Clock Conversion on the GCC tab of the Implementation Options dialog box.



Conditions for Generated-Clock Optimization

To perform generated-clock optimization, the following conditions must be met:

1. The combinational logic must be driven by flip-flops.
2. The input flip-flops cannot have an active set or reset (an active-low reset must be tied high, or an active-high set must be tied low). Similar rules apply to all the input flip-flops in the cone.
3. All input flip-flops must be driven by the same edge of the same clock.

With generated-clock optimization, you do not have to specify a primary clock.

Generated-Clock Optimization Examples

Example 1: Generated Clock from Combinational Logic

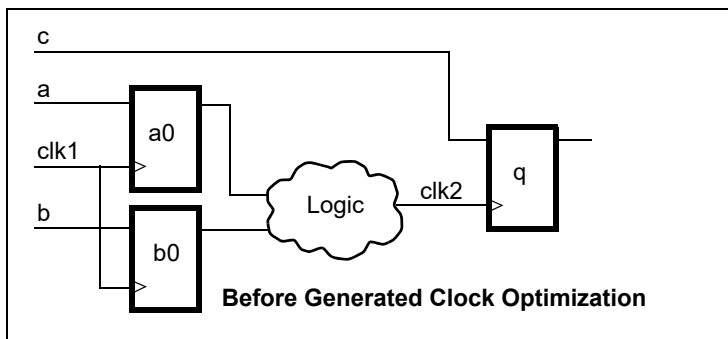
The following code segment illustrates generated-clock optimization:

```
module gen_clk(clk1,a,b,c,q);
  input clk1, a, b, c;
  output q;
  reg ao,bo,q;
  wire en;

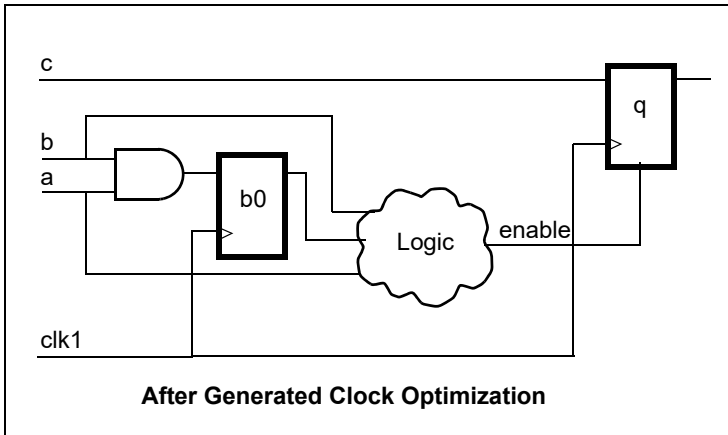
  always @(posedge clk1)
    begin
      ao <= a;
      bo <= b;
    end
  assign en = ao & bo;

  always @(posedge en)
    begin
      q <= c;
    end
endmodule
```

With generated-clock optimization disabled (Clock Conversion unchecked), the circuit in the following figure shows a flip-flop (q) driven by a generated clock that originates from the combinational logic driven by flip-flops a0 and b0 which, in turn, are driven by the initial clock (clk1).

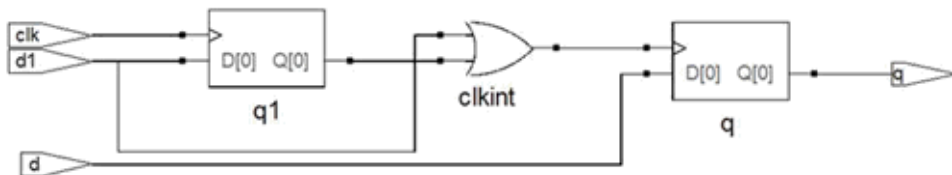


When generated-clock optimization is enabled (Clock Conversion checked), flip-flop *q* is replaced with an enable flip-flop. This flip-flop is clocked by the initial clock (*clk1*) and is enabled by combinational logic based on the *a* and *b* inputs as shown in the following figure.



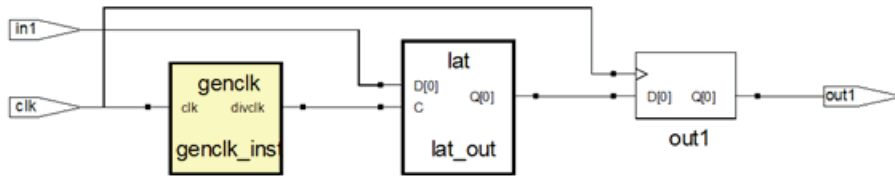
Example 2: Generated Clock Followed by a Gate Clock

A design can have a register that is clocked by a generated clock, then followed by a gated clock. If the generated-clock constraint also includes the enable, the Clock Conversion option may optimize away the register and replace it with a constant value. When this occurs, the software converts the generated and gated-clock logic into a clock and an enable; the enable may never go active.

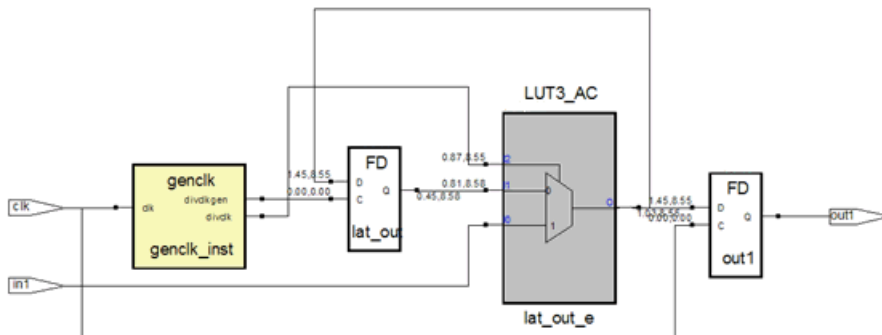


Example 3: Datapath Latch Conversion

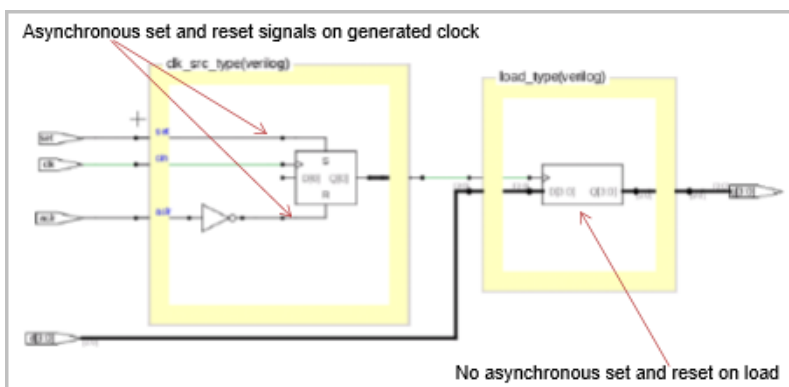
This example shows a latch on the datapath that is clocked by a generated clock.



GCC converts the circuit to a flip-flop and multiplexer as shown in the following figure.



In this example, there are asynchronous set and reset signals on the clock generated from the flip-flop, but the load does not have asynchronous set and reset signals. By default, the tool does not run GCC in this scenario.



CHAPTER 16

Optimizing for Lattice Designs

This chapter covers techniques for optimizing your design for Lattice. The information in this chapter is intended to be used together with the information in [Chapter 9, *Inferring High-Level Objects*](#).

This chapter describes [Optimizing Lattice Designs](#), on page 618.

For additional information about working with Lattice designs, see [Passing Information to the P&R Tools](#), on page 632 and [Generating Vendor-Specific Output](#), on page 633.

Optimizing Lattice Designs

This section describes the following techniques for working with Lattice designs:

- [Instantiating Lattice Macros](#), on page 618
- [Using Lattice GSR Resources](#), on page 619
- [Inferring Lattice PIC Latches](#), on page 620
- [Controlling I/O Insertion in Lattice Designs](#), on page 627
- [Forward-Annotating Lattice Constraints](#), on page 628

Instantiating Lattice Macros

You can instantiate Lattice macros that are predefined in the Lattice libraries that come with the tool, in the *installDirectory/lib* directory.

1. To use a Verilog macro library, add the appropriate library to your project, making sure that it is the first file in the source files list.

The Verilog macro libraries are under the *installDirectory/lib* directory: Add the library appropriate to the technology and language (v or vhd) you are using

LIFCL device	installDirectory/lib/lucent/lifcl
LFD2NX device	installDirectory/lib/lucent/lfd2nx
LFMX05 device	installDirectory/lib/lucent/lfmxo5
LFPCNX device	installDirectory/lib/lucent/lfcpx
iCE40UP device	installDirectory/lib/generic/ice40up

2. To use a VHDL macro library, add the appropriate library and use clauses to your VHDL source code at the beginning of the design units that instantiates the macros.

You only need the VHDL macro libraries for simulation, but it is good practice to add them to the code. The library names may vary, depending on the map file name, which is often user-defined. The simulator uses the map file names to point to a library.

LIFCL device	<code>library lifcl;</code> <code>use lifcl.components.all</code>
LFD2NX device	<code>library lfd2nx;</code> <code>use lfd2nx.components.all</code>
LFMX05 device	<code>library lfmxo5;</code> <code>use lfmxo5.components.all</code>
LFCPNX device	<code>library lfcpnx;</code> <code>use lfcpnx.components.all</code>
ICE40UP device	<code>library ice40up;</code> <code>use ice40up.components.all</code>

3. Instantiate the macros from the library as described in [Instantiating Black Boxes and I/Os in Verilog, on page 384](#) and [Instantiating Black Boxes and I/Os in VHDL, on page 386](#).

Using Lattice GSR Resources

The following procedure describes how to use GSR (global set/reset) resources and check resource usage. The GSR resource is a prerouted signal that connects to the reset input of every flip-flop, regardless of any other defined reset signals.

1. You can control the use of GSR resources as follows:

- To improve routability and performance, use the dedicated GSR resource. Select Project ->Implementation Options and enable the Force GSR Usage option on the Device tab.

When you set this option, the synthesis software creates a GSR instance to access the resource. It uses the GSR resource for reset signals, instead of general routing. All registers are reset. when the GSR is activated, even if some flip-flops do not have a reset.

- If a global set/reset does not correctly initialize the design, turn off the option. Select Project ->Implementation Options and disable the Force GSR Usage option on the Device tab. When this option is off, the software does not use the GSR resource unless all flip-flops have resets, and all resets use the same signal.

2. To optimize area, set the Resource Sharing option, as described in [Sharing Resources, on page 434](#).

3. To check resource usage, do the following:
 - Synthesize the design.
 - Select View Log and check the Resource Usage section.

Inferring Lattice PIC Latches

The following procedure shows you how to control the inference of programmable I/O cells (PICs) in Lattice designs.

1. For the software to automatically infer PICs, make sure of the following:
 - The latch must be at the input port.
 - The latch must be directly driven by the input FPGA pad.
 - The design has one of the supported input control schemes: no clear or reset controls, and asynchronous clear or asynchronous resets.

After synthesis, the tool implements the following primitives for the PICs:

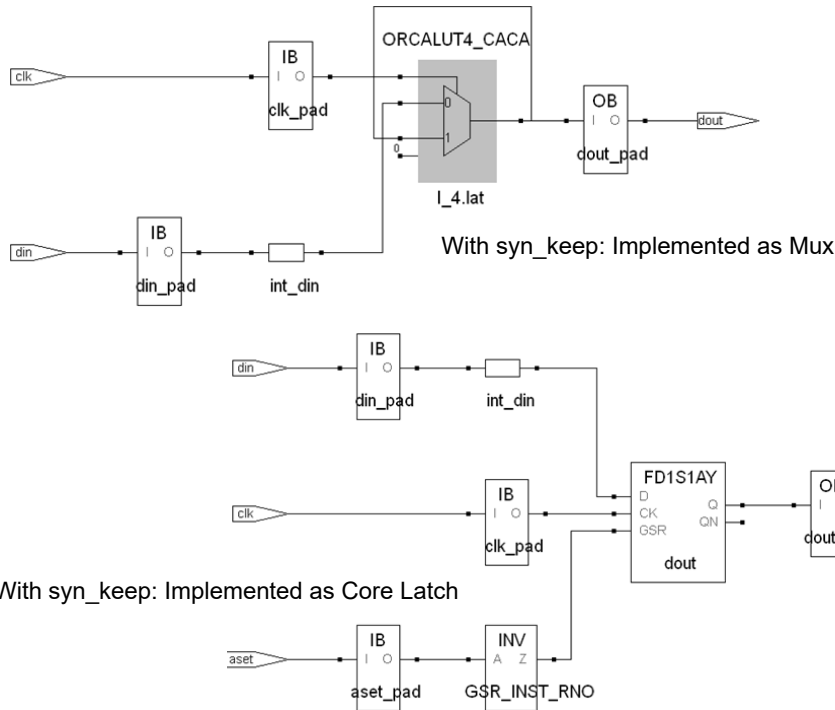
IF1S1D	Latches with asynchronous clear Latches with GSR used for clear

IF1S1B	Latches with asynchronous reset Latches with GSR used for reset.

See [Examples of PIC Latches, on page 621](#) for examples of inferred PIC latches.

2. If you do not want to infer a PIC, set `syn_keep` on the input data net for the latch.

After synthesis, the tool implements the latch as either a core latch with the LATCH primitive or as a mux, depending on the Lattice technology you selected. The following figure shows an input latch with no reset or clear implemented as a mux and a core latch in different technologies.



Examples of PIC Latches

The following examples show how the tool infers PICs from the code:

- [Positive Level Data Latch with No Resets or Clears](#), on page 622
- [Negative Level Data Latch with No Resets or Clears](#), on page 623
- [Positive Level Data Latch with Asynchronous Reset](#), on page 624
- [Positive Level Data Latch with Asynchronous Clear](#), on page 626

Positive Level Data Latch with No Resets or Clears

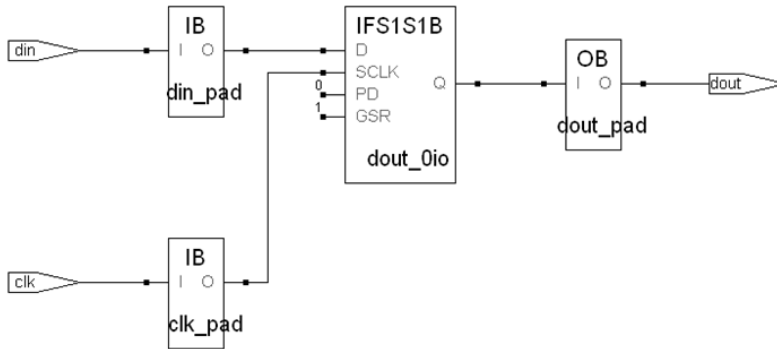
```
VHDL    library ieee;
        use ieee.std_logic_1164.all;

        entity inlatch is port
          (clk : in std_logic;
           din : in std_logic;
           dout: out std_logic);
        end entity inlatch;

        architecture bhve of inlatch is
        begin
          process (clk,din)
          begin
            if clk='1' then
              dout <= din;
            end if;
          end process;
        end bhve;
```

```
Verilog module inlatch (clk,din,dout);
         input clk;  input din;  output dout;
         reg dout;
         always @(clk)
         begin
           if(clk)
             dout <= din;
         end
         endmodule
```

With this code, the tool implements the IFS1S1B latch primitive in the Technology view:



Negative Level Data Latch with No Resets or Clears

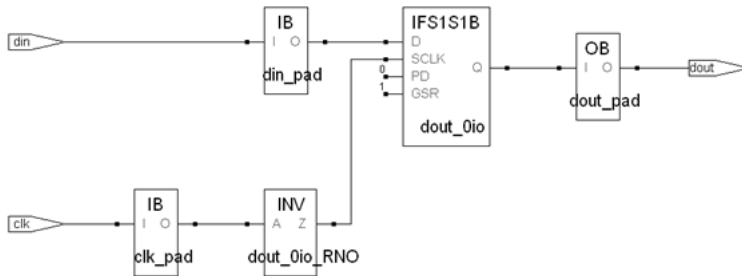
```
VHDL
library ieee;
use ieee.std_logic_1164.all;

entity inlatch is port
  (clk : in std_logic;
   din : in std_logic;
   dout: out std_logic);
end entity inlatch;

architecture bhve of inlatch is
begin
  process(clk,din)
  begin
    if clk='0' then
      dout <= din;
    end if;
  end process;
end bhve;
```

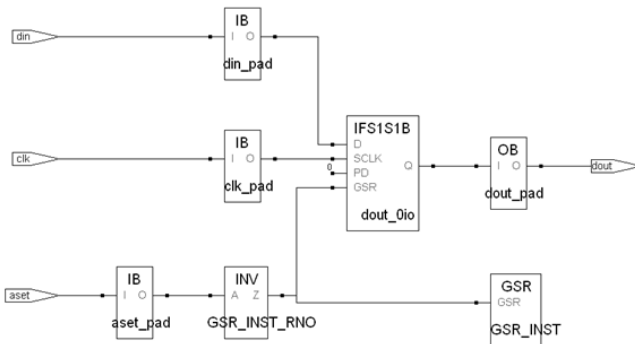
```
Verilog
module inlatch (clk,din,dout);
input clk; input din; output dout;
reg dout;
always @(clk)
begin
  if(!clk)
    dout <= din;
end
endmodule
```

With this code, the tool implements the IFS1S1B latch primitive in the Technology view:



Positive Level Data Latch with Asynchronous Reset

The tool infers the IFS1S1B latch primitive in the Technology view with the code shown below:



```
VHDL    library ieee;
        use ieee.std_logic_1164.all;
        entity inlatch is port
        (clk : in std_logic; aset: in std_logic;
         din : in std_logic; dout: out std_logic);
        end entity inlatch;
        architecture bhve of inlatch is
        begin
            process (clk,din,aset)
            begin
                if aset ='1' then
                    dout <='1';
                elsif clk='1' then
                    dout <= din;
                end if;
            end process;
        end bhve;
```

```
Verilog module inlatch(clk,din,aset,dout);
        input clk; input  din; input aset; output dout;
        reg  dout;
        always @(clk or aset)
        begin
            if(aset)
                dout <= 1'b1;
            else if (clk)
                dout <= din;
        end
        endmodule
```

Positive Level Data Latch with Asynchronous Clear

```
VHDL    library ieee;
        use ieee.std_logic_1164.all;

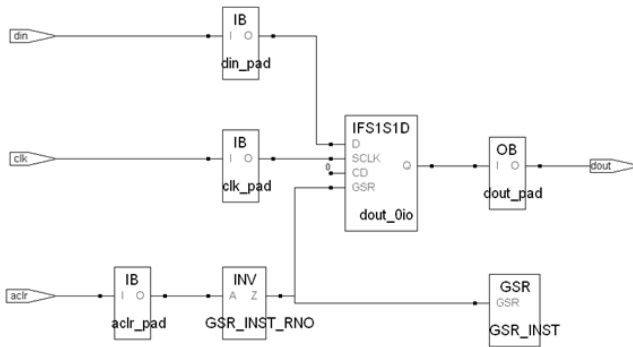
        entity inlatch is port
        (clk : in std_logic; aclr: in std_logic;
        din : in std_logic; dout: out std_logic); }
        end entity inlatch;

        architecture bhve of inlatch is
        begin
            process(clk,din,aclr)
            begin
                if aclr ='1' then
                    dout <='0';
                elsif clk='1' then
                    dout <= din;
                end if;
            end process;
        end bhve;
```

```
Verilog module inlatch(clk,din,aclr,dout);
        input  clk; input  din; input aclr; output dout;
        reg  dout;

        always @(clk or aclr)
        begin
            if(aclr)
                dout <= 1'b0;
            else if (clk)
                dout <= din;
        end
        endmodule
```

With this code, the tool infers the IFS1S1D primitive in the Technology view:



Controlling I/O Insertion in Lattice Designs

You can control I/O insertion globally, or on a port-by-port basis.

1. To control the insertion of I/O pads at the top level of the design, use the Disable I/O Insertion option as follows:

- Select Project->Implementation Options and click the Device panel.
- If you do not want to insert any I/O pads in the design, enable Disable I/O Insertion

Do this if you want to check the area your blocks of logic take up, before you synthesize an entire FPGA. If you disable automatic I/O insertion, you do not get *any* I/O pads in your design, unless you manually instantiate them.

- If you want to insert I/O pads, disable the Disable I/O Insertion option.

When this option is set, the software inserts I/O pads for inputs, outputs, and bidirectionals in the output netlist. Once inserted, you can override the I/O pad inserted by directly instantiating another I/O pad.

2. To force I/O pads to be inserted for input ports that do not drive logic, follow the steps below.
 - To force I/O pad insertion at the module level, set the `syn_force_pads` attribute on the module. Set the attribute value to 1. To disable I/O pad insertion at the module level, set the `syn_force_pads` attribute for the module to 0.

- To force I/O pad insertion on an individual port, set the `syn_force_pads` attribute on the port with a value to 1. To disable I/O insertion for a port, set the attribute on the port with a value of 0.

Enable this attribute to preserve user-instantiated pads, insert pads on unconnected ports, insert bi-directional pads on bi-directional ports instead of converting them to input ports, or insert output pads on unconnected outputs.

If you do not set the `syn_force_pads` attribute, the synthesis design optimizes any unconnected I/O buffers away.

Forward-Annotating Lattice Constraints

You can forward-annotate multicycle and false path constraints to Lattice place-and-route tool by following the procedure below. For additional information about forward-annotation, see [Generating Constraint Files for Forward Annotation \(Legacy\)](#), on page 50.

1. To forward-annotate a from, to, or through multicycle constraint, open the SCOPE spreadsheet and do either of the following:
 - Click the Multi-Cycle Paths tab. Depending on the type of constraint you want to set, select or type the instance name under the To, From or Through column. Next, set the number of clock cycles under the Cycles column.

When you set this constraint, the software runs timing-driven synthesis and then forward-annotates the constraint.

- Click the Other tab. In the Command column, type `define_multicycle_path`. In the Arguments column, type `-from` and the source port or register name, and `-to` and the destination port or register name. For example: `-from in0_int -to output 2`.

When you set this constraint from the Other tab, the software forward-annotates the constraint, but does not run timing-driven synthesis using this constraint.

2. To forward-annotate a false path constraint, open the SCOPE spreadsheet and do either of the following:
 - Click the False Paths panel. Depending on the type of constraint you want to set, select or type the instance name under the To, From or Through column. When you set this constraint, the software runs timing-driven synthesis and then forward-annotates the constraint.
 - Click the Other tab. In the Command column, type `define_false_path`. In the Arguments column, type `-from` and the source port or register name, and `-to` and the destination port or register name. For example:

Command	Arguments
<code>set_false_path</code>	<code>-from in1_int -to output</code>
<code>set_false_path</code>	<code>-from in* -to out*</code>

When you set this constraint from the Other tab, the software forward-annotates the constraint, but does not run timing-driven synthesis using this constraint.

3. Select Project->Implementation Options and enable the Write Vendor Constraint File option on the Implementation Results tab.
4. Run your design. The synthesis tool creates the `$DESIGN_synplify.ldc` file in the same directory as your results files.
5. Start the Lattice Radiant place-and-route tool and run the Map stage (after importing the `$DESIGN_synplify.ldc` file).
6. Run the PAR and BIT stages in Radiant.

CHAPTER 17

Working with Synthesis Output

This chapter covers techniques for optimizing your design for various vendors. The information in this chapter is intended to be used together with the information in [Chapter 9, *Inferring High-Level Objects*](#).

This chapter describes the following:

- [Passing Information to the P&R Tools](#), on page 632
- [Generating Vendor-Specific Output](#), on page 633
- [Invoking Third-Party Vendor Tools](#), on page 634

Passing Information to the P&R Tools

The following procedures show you how to pass information to the place-and-route tool; this information generally has no impact on synthesis. Typically, you use attributes to pass this information to the place-and-route tools. This section describes the following:

- [Specifying Pin Locations](#), on page 632
- [Generating Vendor-Specific Output](#), on page 633

Specifying Pin Locations

In certain technologies you can specify pin locations that are forward-annotated to the corresponding place-and-route tool. The following procedure shows you how to specify the appropriate attributes.

1. Start with a Lattice design
2. Add the appropriate attribute to the port. For a bus, list all the bus pins, separated by commas.
 - To add the attribute from the SCOPE interface, click the **Attributes** tab and specify the appropriate attribute and value.
 - To add the attribute in the source files, use the appropriate attribute and syntax. See the *Attribute Reference Manual* for syntax details.

Family	Attribute and Value
Lattice	<code>loc {pin_number}</code>

Generating Vendor-Specific Output

The following topics describe generating vendor-specific output in the synthesis tools. See [Targeting Output to Your Vendor](#), on page 633.

Targeting Output to Your Vendor

You can generate output targeted to your vendor.

1. To specify the output, click the Implementation Options button.
2. Click the Implementation Results tab, and check the output files you need.

The following table summarizes the outputs to set for the different vendors, and shows the P&R tools for which the output is intended.

Vendor	Output Netlist	P&R Tool
Lattice	VM (.vm)	Radiant

3. To generate mapped Verilog/VHDL netlists and constraint files, check the appropriate boxes and click OK.

See [Specifying Result Options](#), on page 78 for details about setting the option. For more information about constraint file output formats and how constraints get forward-annotated, see [Generating Constraint Files for Forward Annotation \(Legacy\)](#), on page 50.

Invoking Third-Party Vendor Tools

You can invoke third-party tools from within the Synopsys FPGA synthesis products, and configure the locations and common arguments for the tools. This capability lets you modify source files or libraries or debug problems from within the third-party tool, without leaving the synthesis environment.

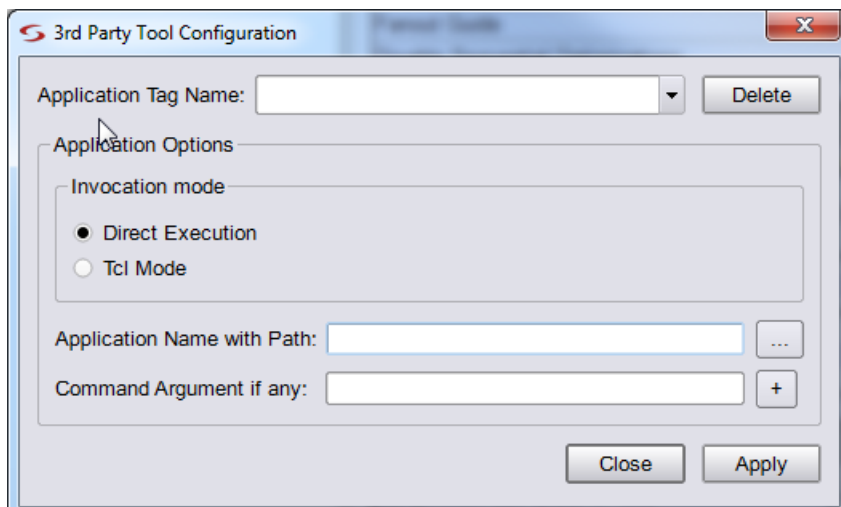
You can invoke pre-configured tools or add your own. The process consists of two steps:

- [Configuring Tool Tags](#), on page 634
- [Invoking a Third-Party Tool](#), on page 635

Configuring Tool Tags

A tool tag is a configuration definition for a tool you want to invoke from the synthesis interface. You define a tool tag to set up the third-party tool you want to use. The synthesis software has some popular applications already configured for you to use when the synthesis software starts up. The following procedure shows you how to define your own tool tags, or add command arguments. Use this to specify other tools, other versions of a tool, or to run a tool with different arguments.

1. Select Options->Configure 3rd Party Tool Options from the Project view.



2. Define the application tag information for the tool you want to invoke.
 - Specify the application you want to invoke in Application Tag Name.
 - Specify how you want to invoke the application tool. If you want to run the tool directly from the UI, select Direct Execution. If your application is a Tcl procedure, select TCL Mode.
 - Specify the location of the application executable or Tcl procedure name in Application Name with Path or Tcl Procedure Name.
 - Specify any command arguments you want in the Command Argument if any field. You can use this to define a new tool tag or to add arguments to a tool tag that is already defined.

For a list of predefined command arguments, click the + button and select them from the list. Otherwise, type the command arguments. For other internal Synopsys tools, you must select \$SynCode from the Command Argument if any field.

- Click Apply.
- Click Close.

The tool saves these settings in the FPGA synthesis tool .ini file and retrieves them for subsequent invocations. For information about invoking a third-party tool, see [Invoking a Third-Party Tool, on page 635](#), next.

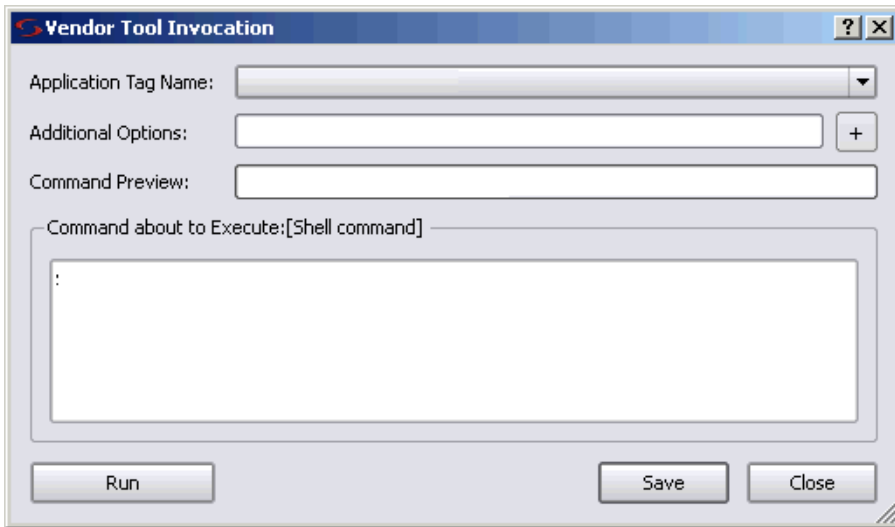
Invoking a Third-Party Tool

You can define tool tags globally and then use these tool tags to run the third-party tool from the Project view for the specified tool tag only. Some common tool tags are pre-configured and are read when the application starts up. You can add or modify existing tool tags or define your own Tcl procedures to invoke within the FPGA synthesis tools.

1. Define a tool tag for your application, as described in [Configuring Tool Tags, on page 634](#).
2. Right-click in the Project view on a file or folder which is configured to run the vendor tool, and select Launch Tools->Run Vendor Tool from the popup menu.

This dialog box automatically displays tool tag information associated with the file or folder. If no tool tag information is specified, look for the parent hierarchy and edit or change it, if necessary.

3. To associate a file or folder with a particular third-party tool, do the following:
 - Select the file or folder in the Project view. If you select a folder, the third-party tool is associated with all the files in the folder. If you associate a tool with a file, this setting overrides the folder setting.
 - Right-click a file or folder and select Launch Tools->Run Vendor Tool from the popup menu.



- In the Vendor Tool Invocation dialog box, select the application in Application Tag Name.
 - Include any additional options you want to use with this file when you invoke the vendor tool. You can set command arguments now, if you did not configure them earlier.
 - Verify the command string in the dialog box.
 - Click Save, and Close. The third-party tool is associated with the file or folder and appears in the Launch Tools menu.
4. To invoke an associated third-party tool for a file or folder, do the following:
 - Right-click the file or folder in the Project view.
 - Select Launch Tools-><Third-Party Tool> from the popup menu. The synthesis tool automatically runs the tool or Tcl procedure as specified.

5. To invoke the tool at the same time that you associate a third-party tool with a file or folder, or to add additional arguments on the fly, do the following:
 - Right-click a file or folder and select Launch Tools->Run Vendor Tool from the popup menu.
 - In the Vendor Tool Invocation dialog box, select the application in Application Tag Name.
 - Include any additional options you want to use with this file when you invoke the vendor tool. You can set command arguments now, if you did not configure them earlier.
 - Verify the command string in the dialog box.
 - Click Save. The tool and arguments you specified is associated with the file or folder and appears in the Launch Tools menu.

If you defined a new tool tag, the 3rd Party Tool Configuration dialog box appears. After saving the settings here, go back to the Vendor Tool Invocation dialog box. You are prompted to save this information to the project file before invoking the third-party tool.

- Click the Run button in the Vendor Tool Invocation dialog box. The synthesis tool launches the third-party tool or runs the Tcl procedure with the arguments you specified.

These settings are saved in the FPGA synthesis tool `ini` file, from where it can be retrieved for subsequent invocations.

CHAPTER 18

Running Post-Synthesis Operations

The following topics describe how to run post-synthesis operations, like place-and-route and verification, with compatible tools:

- [Running P&R Automatically after Synthesis](#), on page 640
- [Simulating with the VCS Tool](#), on page 643

Running P&R Automatically after Synthesis

You can run place-and-route automatically from within the tool or in batch mode for Lattice iCE devices.

For detailed procedures, see the following:

- [Integrating Synthesis and Place-and-Route in One Run](#), on page 640
- [Running Place and Route Using a Custom Script](#), on page 640
- [Releasing the Synthesis License During Place and Route](#), on page 641

Integrating Synthesis and Place-and-Route in One Run

You can run the place-and-route tool for your target technology automatically after synthesis.

1. Check the Release Notes and make sure that you are using the correct version of the P&R tool.
2. Set the PATH variable to point to the place-and-route tool.
3. To automatically run the P&R tool after synthesis completes, do the following:
 - Click the Add P&R Implementation button. In the dialog box, select the P&R implementation you want to run and enable Run Place & Route following synthesis.
 - Synthesize the design.

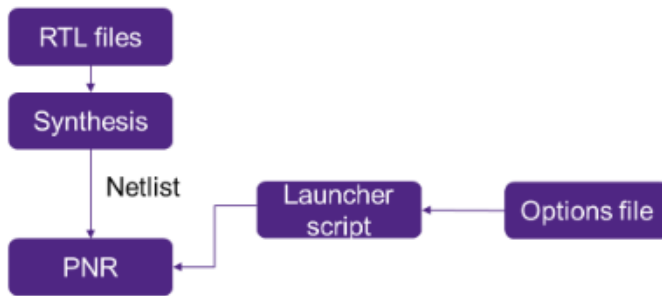
The tool automatically runs P&R after synthesis.

Running Place and Route Using a Custom Script

Ideally, after synthesis, the tool provides an *options file* (.tcl), along with the synthesis netlist output, to launch the integrated place and route (P&R) tool for your target technology.

For devices that do not have integrated P&R support, FPGA designers can provide a custom shell script to launch the P&R flow based on the device selected in the project options. The shell script provides the path to the P&R tool and identifies the output files (output netlist and constraints) after synthesis.

The flow is shown below:



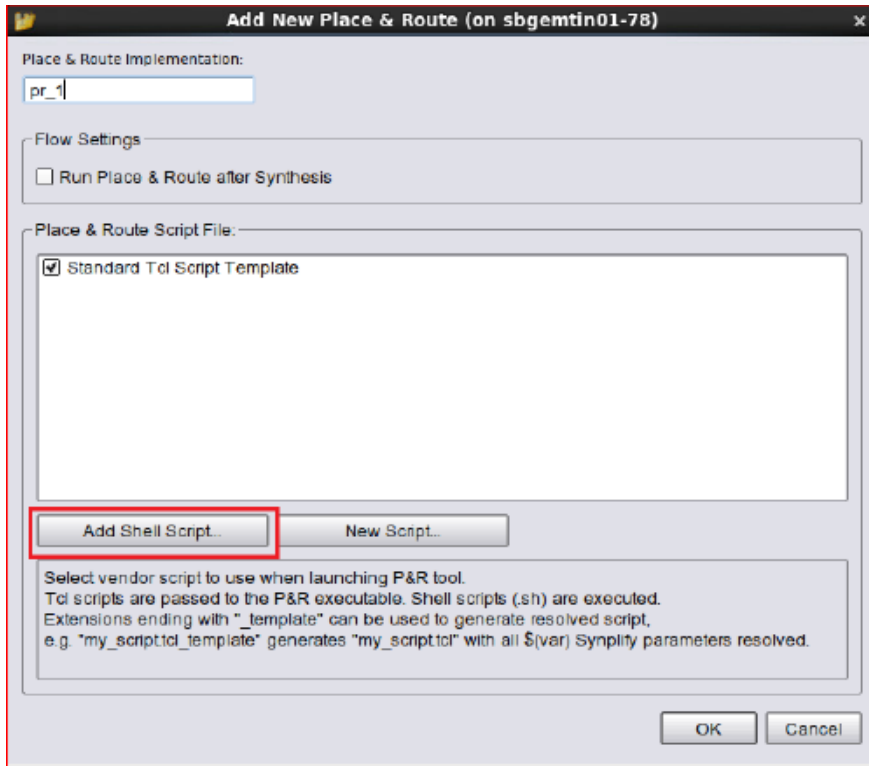
To launch the P&R tool using a custom P&R script, do the following:

1. Ensure that you have a custom script to use when launching the P&R tool.
2. Click the Add P&R Implementation button.
3. In the Add New Place & Route dialog box, from Flow Settings, select the Run Place & Route after synthesis.
4. Click Add Shell Script to add your custom script to the project.
5. Click OK.

The custom script is launched and the tool automatically runs P&R after synthesis.

Releasing the Synthesis License During Place and Route

When invoking a third-party place-and-route tool from the FPGA synthesis tool, you can choose to have place and route continue to run even after exiting the synthesis tool so that it does not consume an FPGA license. The software lets you release the license for the synthesis tool and run the place-and-route tool in batch mode.



To release the FPGA license, specify the following command:

```
toolName -batch -license_release
```

Where *toolName* can be the following keyword: `synplify_pro`.

In synthesis batch mode (`synbatch`), the `-license_release` option obtains all the synthesis licenses that are checked out for the session and checks them in immediately after the place-and-route job is launched.

When licenses are released, you see the following message is generated:

```
Exiting session due to -license_release option
```

Simulating with the VCS Tool

The Synopsys VCS® tool is a high-performance, high-capacity Verilog simulator that incorporates advanced, high-level abstraction verification technologies into a single, open, native platform. You can launch this simulation tool from the synthesis tools on Linux and Unix platforms by following the steps below. The VCS tool does not run under the Windows operating system.

1. Set up the tools.

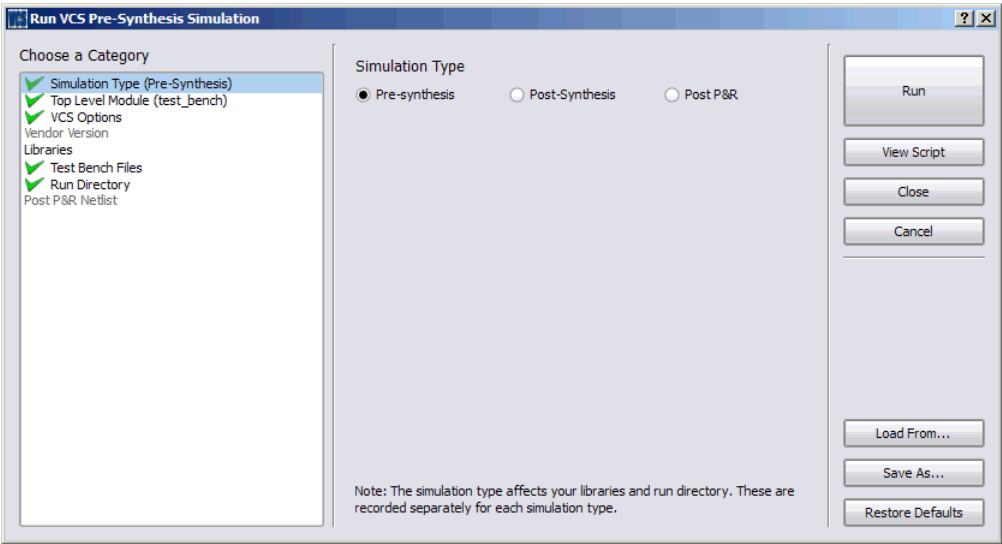
- Install the VCS software and set up the \$VCS_HOME environment variable to define the location of the software.
- Set up the place-and-route tool.
- In the synthesis software, either select Run->Configure and Launch VCS Simulator, or click the  icon.

If you did not set up the \$VCS_HOME environment variable, you are prompted to define it. The Run VCS Simulator dialog box opens. For descriptions of the options in this dialog box, see [Configure and Launch VCS Simulator Command](#), on page 429 of the *Reference Manual*.

2. Choose the category Simulation Type in the dialog box to configure the simulation options.

- Specify the kind of simulation you want to run.

RTL simulation	Enable Pre-Synthesis
Post-synthesis netlist simulation	Enable Post-Synthesis
Post-P&R netlist simulation	Enable Post P&R

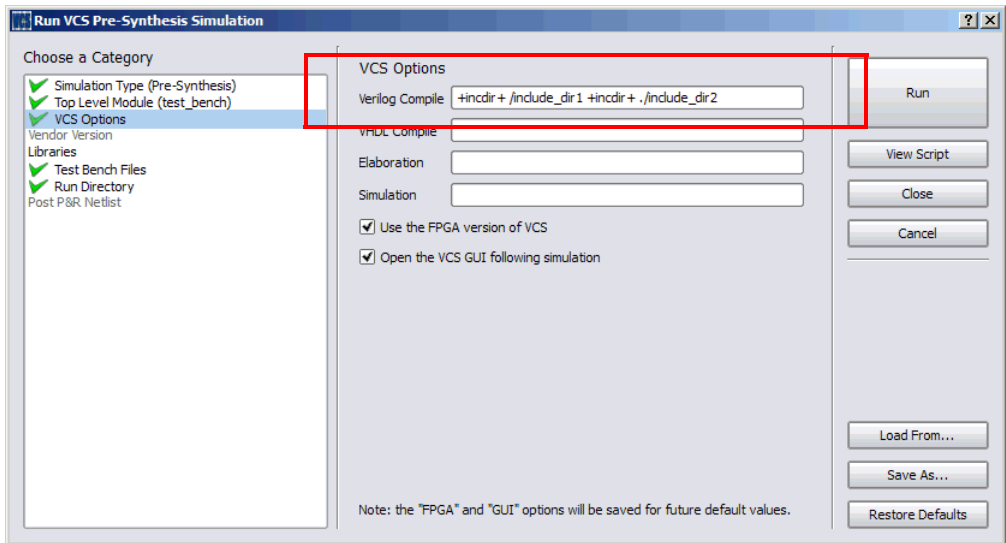


- Choose the category VCS Options in the dialog box to set options such as the following VCS commands.

To set...	Type the option in...
VLOGAN command options for compiling and analyzing Verilog, like the -q option	Verilog Compile
VHDLAN options for compiling and analyzing VHDL	VHDL Compile
VCS command options	Elaboration
SIMV command options, like -debug	Simulation

The options you set are written out as VCS commands in the script. If you leave the default settings the VCS tool uses the FPGA version of VCS and opens with the debugger (DVE) GUI and the waveform viewer. See the VCS documentation for details of command options.

3. If your project has Verilog files with ``include` statements, you must use the `+incdir+ fileName` argument when you specify the `vlogan` command. You enter the `+incdir+` in the Verilog Compile field in the VCS Options dialog box, as shown below:



Example Verilog File:

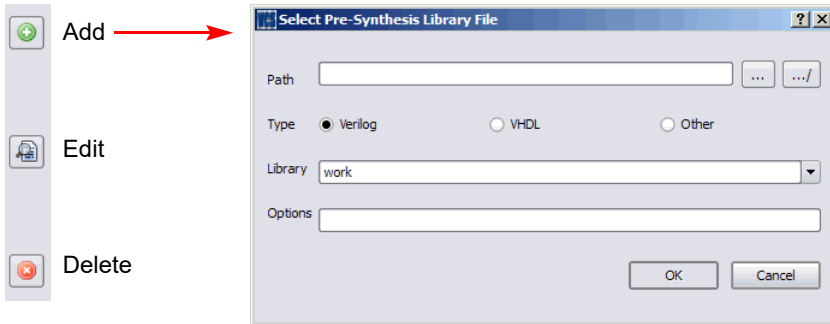
```
`include "component.v"
module Top (input a, output x);
...
endmodule
```

The syntax for the VCS commands must reflect the relative location of the Verilog files:

- If the Verilog files are in the same directory as the top.v file, specify:
 - vlogan -work work Top.v +incdir+ ./
- If the Verilog files are in the a directory above the top.v file, specify:
 - vlogan -work work Top.v +incdir+ ../include1 +incdir+ ../ include2
- If the Verilog files are in directories below and above the top.v file, specify:
 - vlogan -work work Top.v +incdir+ ./include_dir1 +incdir+../include_dir2

4. Specify the libraries and test bench files, if you are using them.

- To specify a library, click the green Add button, and specify the library in the dialog box that opens. Use the full path to the libraries. For pre-synthesis simulation, specifying libraries is optional.



- For post-synthesis and post-P&R synthesis, by default the dialog box displays the UNISIM and SIMPRIM libraries in the P&R tool path. You can add and delete libraries or edit them, using the buttons on the side. To restore the defaults, click the Verilog Defaults or VHDL Defaults button, according to the language you are using.
 - If you have test bench files, choose the category Test Bench Files in the dialog box to specify them. Use the buttons on the side to add, delete, or edit the files.
5. Specify the top-level module and run directory.
- Choose the category Top Level Module in the dialog box to specify the top-level module or modules for the simulation.
 - If necessary, choose the category Run Directory near the bottom of the dialog box to edit the default run directory listed in the field. The default location is in the implementation results directory.
6. Generate the VCS script.
- To view the script before generating it, click the View Script button on the top right of the dialog box. A window opens with the specified VCS commands and options.
 - To generate the VCS script, click Save As, or run VCS by clicking the Run button in the upper right. The tool generates the XML script in the directory specified.

7. To run VCS from the synthesis tool interface, do the following:
 - If you do not already have it open, open the Run VCS Simulator dialog box by clicking the  icon.
 - To use an existing script, click the Load From button on the lower right and select the script in the dialog box that opens. Then click Run in the Run VCS Simulator dialog box.
 - If you do not have an existing script, specify the VCS options, as described in the previous five steps. Click Run.

The tool invokes VCS from the synthesis interface, using the commands in the script.

Limitations

If Verilog include paths have been added to your project file, these paths are not automatically added to the VCS script. Add the Verilog include paths manually by using one of the following workarounds:

- From the Run VCS Simulator dialog box, add `+incdir+includePath` in the Verilog Compile options field.
- Modify the VCS script file, adding the `+incdir+includePath` to all or any relevant `vlogan` commands.

Index

Symbols

.adc file [373](#)
.ini file
 parallel jobs [565](#)
.ldc file [629](#)
.scf file [52](#)

Numerics

3rd party vendor tools
 invoking [634](#)

A

adc constraints [373](#)
adc file
 creating [373](#)
 object names [377](#)
adc file, using [371](#)
adders
 SYNCore [512](#)
Alt key
 column editing [35](#)
 mapping [334](#)
analysis design constraint file (.adc) [373](#)
analysis design constraints
 design scenarios [372](#)
analysis design constraints (adc) [371](#)
analysis design constraints (adc), using
 with sdc [373](#)
Analyst view
 traversing hierarchy [249](#)
archive utility
 using [117](#)
archiving projects [117](#)
area, optimizing [415](#)
asterisk wildcard
 Find command [265, 322](#)

attributes
 adding [86](#)
 adding in constraint files [51](#)
 adding in SCOPE [89](#)
 adding in Verilog [88](#)
 adding in VHDL [87](#)
 effects of retiming [423](#)
 for FSMs [395](#)
 handling properties [93](#)
 inferring RAM [399](#)
 syn_hier (on compile points) [477](#)
 VHDL package [87](#)

Attributes panel
 using SCOPE [139](#)
auto constraints, using [378](#)
AutoConstraint_design_name.sdc [381](#)

B

B.E.S.T [338](#)
backslash
 escaping dot wildcard in Find
 command [265, 322](#)
batch mode [558](#)
 using find and expand [164](#)
Behavior Extracting Synthesis
 Technology. *See* B.E.S.T
black boxes [384](#)
 adding constraints [388](#)
 adding constraints in SCOPE [391](#)
 adding constraints in Verilog [390](#)
 adding constraints in VHDL [389](#)
 gated clock attributes [605](#)
 instantiating in Verilog [384](#)
 instantiating in VHDL [386](#)
 passing VHDL boolean generics [46](#)
 passing VHDL integer generics [47](#)
 pin attributes [392](#)
 setting in hierarchical projects [109](#)
block RAM
 inferring [401](#)

- modes 398
- types 398
- blocking-style license queuing 561
- blocks
 - defining for hierarchical projects 95
- bookmarks
 - in source files 35
 - using in log files 206
- bottom-up design flow
 - compile point advantages 448
- browsers 309
- buffering
 - controlling 433
- byte-enable RAMs
 - SYNCore 500

C

- c_diff command, examples 170
- c_intersect command, examples 170
- c_list command
 - different from c_print 172
 - example 174
 - using 173
- c_print command
 - different from c_list 172
 - using 173
- c_syndiff command, examples 171
- c_union command, examples 170
- callback functions, customizing flow 571
- case sensitivity
 - Find command (Tcl) 157
- clock and path constraints
 - setting 140
- clock constraints
 - setting 140
 - setting (Legacy) 181
- clock conversion report
 - accessing 593
 - analyzing 594
- clock groups
 - effect on false path constraints 155
- clock trees 363
- clocks
 - implicit false path 155

- Clocks panel
 - using SCOPE 138
- collections
 - adding objects 169
 - concatenating 169
 - constraints 167
 - copying 173
 - creating from common objects 169
 - creating from other collections 167
 - creating in SCOPE 166
 - creating in Tcl 168
 - crossprobing objects 167
 - definition 165
 - diffing 169
 - highlighting in HDL Analyst views 172
 - listing objects 173
 - listing objects and properties 172
 - listing objects in a file 173
 - listing objects in columnar format 172
 - listing objects with c_list 172
 - special characters 171
 - Tcl window and SCOPE
 - comparison 165
 - using Tcl expand command 161
 - using Tcl find command 160
 - viewing 171
- Collections panel
 - using SCOPE 138
- column editing 35
- comments
 - source files 35
- compile point types
 - hard 453
 - locked 453
 - locked,partition 455
- compile points
 - advantages 448
 - analyzing results 478
 - automatic timing budgeting 459
 - child 451
 - constraint files 456
 - constraints for forward-annotation 467
 - constraints, internal 467
 - creating constraint file 476
 - defined 448
 - defining in constraint files 473
 - feature summary 455
 - incremental synthesis 480
 - manual compile point flow 469

- nested 451
 - optimization 463
 - order of synthesis 463
 - parent 451
 - preserving with syn_hier 477
 - resynthesis 464
 - setting constraints 476
 - setting type 475
 - syn_hier 477
 - synthesis process 463
 - synthesizing 469
 - types 452
 - using syn_allowed_resources attribute 478
 - Compile Points panel 139
 - compile-point synthesis
 - interface logic models 459
 - compile-point synthesis flow
 - defining compile points 473
 - setting constraints 476
 - compiler directives (Verilog)
 - specifying 82
 - constants
 - extracting from VHDL source code 84
 - constraint files
 - applying to a collection 167
 - compile point 456, 467
 - creating in a text editor 50
 - editing 147
 - effects of retiming 423
 - options 77
 - setting for compile points 476
 - vendor-specific 51
 - constraints
 - defining clocks (Legacy) 176
 - defining register delays (Legacy) 177
 - specifying through points 151
 - types 138
 - types (legacy) 178
 - using FDC template command 136
 - context
 - for object in filtered view 341
 - context help editor 30
 - SystemVerilog 30
 - counters
 - SYNCore 520
 - create_fdc_template
 - using 136
 - critical paths
 - delay 365
 - flat view 364
 - hierarchical view 364
 - negative slack on clock enables (Legacy) 188
 - slack time 365
 - using -route 417
 - viewing 363
 - crossprobing 330
 - and retiming 423
 - collection objects 167
 - filtering text objects for 335
 - from FSM viewer 336
 - from log file 206
 - from message viewer 220
 - from text files 333
 - Hierarchy Browser 266, 330
 - importance of encoding style 337
 - new HDL Analyst views 267
 - paths 334
 - RTL view 331
 - schematic views 266
 - Technology view 331
 - Text Editor view 331
 - text file example 334
 - to FSM Viewer 336
 - to place-and-route file 306
 - Verilog file 331
 - VHDL file 331
 - within RTL and Technology views 330
 - current level
 - expanding logic from net 345
 - expanding logic from pin 345
 - searching current level and below 319
 - custom folders
 - creating 64
 - hierarchy management 64
 - customization
 - callback functions 571
- ## D
- data block 532
 - data key 532
 - default enum encoding 84
 - define_attribute 93
 - Delay Paths panel

- using SCOPE 139
- design flow
 - customizing with callback functions 571
- design guidelines 414
- design hierarchy
 - viewing 274, 339
- design size
 - amount displayed on a sheet 306
- design views
 - moving between views 247, 305
- device options
 - See also* implementation options
- directives
 - adding 86
 - adding in Verilog 88
 - adding in VHDL 87
 - black box 389, 390
 - for FSMs 395
 - handling properties 93
 - specifying for Verilog compiler 82
 - syn_state_machine 441
 - syn_tco 390
 - adding black box constraints 389
 - syn_tpd 390
 - adding black box constraints 389
 - syn_tsu 390
 - adding black box constraints 389
- dissolving instances for flattening hierarchy 352
- dot wildcard
 - Find command 265, 322
- drivers
 - preserving duplicates with syn_keep 426
 - selecting 348
- dual-port RAMs
 - SYNCore parameters 497

E

- Editing window 33
- editor view
 - context help 30
- emacs text editor 38
- encoding styles
 - and crossprobing 337
- default VHDL 84
- FSM Compiler 439
- encryption flow. *See* ReadyIP, encryptIP
- encryptip output constraints 543
- encryptip output method
 - effect on output netlists 543
- encryptIP script
 - controlling output 542
 - encrypting IP 541
 - output methods 542
- encryptP1735 script
 - encrypting multiple files 540
 - location 536
 - public keys repository file 540
 - use models 536
- encryptP1735P script
 - encrypting IP 535
- environment variables
 - SYN_TCL_HOOKS 571
- error codes 559
- error messages
 - gated clock report 596
- errors
 - definition 33
 - filtering 219
 - sorting 219
 - source files 32
 - Verilog 32
 - VHDL 32
- expand
 - batch mode 164
- Expand command
 - connection logic 348
 - pin and net logic 281, 344
 - using 345
- expand command
 - different from Tcl search 326
- expand command (Tcl). *See* Tcl expand command
- Expand Inwards command
 - using 281, 345
- Expand Paths command
 - different from Isolate Paths 348
- Expand to Register/Port command
 - using 345
- expanding

- connections 348
- pin and net logic 281, 344
- F**
- false paths
 - defining between clocks (Legacy) 194
 - I/O paths 155
 - impact of clock group assignments 155
 - impact of clock group assignments (Legacy) 194
 - ports 155
 - ports (Legacy) 194
 - registers 155
 - registers (Legacy) 194
 - setting constraints 155
 - setting constraints (Legacy) 194
- fanouts
 - buffering vs replication 433
 - hard limits 432
 - soft global limit 431
 - soft module-level limit 432
 - using syn_keep for replication 427
 - using syn_maxfan 431
- feature comparison
 - FPGA tools 18
- FIFOs
 - compiling with SYNCORE 484
- files
 - .ldc 629
 - .prf file 221
 - .scf 52
 - filtered messages 223
 - fsm.info 440
 - log 203
 - message filter (prf) 221
 - output 633
 - rom.info 312
 - searching 114
 - statemachine.info 357
 - synhooks.tcl 571
 - Tcl 564
 - See also* Tcl commands
 - Tcl batch script 560
- Filter Schematic command,
 - using 279, 342
- Filter Schematic icon, using 279, 342
- filtering 279, 342
 - advantages over flattening 279, 342
 - using to restrict search 319
- Find command
 - 319
 - browsing with 318
 - hierarchical search 320
 - long names 318
 - message viewer 219
 - reading long names 321
 - search scope, effect of 322
 - search scope, setting 320
 - searching the mapped database 321
 - searching the output netlist 327
 - setting limit for results 321
 - using in RTL and Technology views 319
 - using wildcards 265, 322
 - wildcard examples 324
- find command
 - different from Tcl search 326
 - hierarchy 326
 - nuances and differences 326
- find command (Tcl)
 - See* Tcl find command
- Fix Gated Clocks option. *See* gated clocks
- Flatten Current Schematic command
 - transparent instances 350
 - using 350
- Flatten Schematic command
 - using 350
- flattening 289, 349
 - See also* dissolving
 - compared to filtering 279, 342
 - dissolving instances 290, 352
 - hidden instances 351
 - transparent instances 350
 - using syn_hier 429
 - using syn_netlist_hierarchy 429
- forward annotation
 - vendor-specific constraint files 51
- forward-annotation
 - compile point constraints 467
 - constraints 628
- FPGA Design Constraints Editor
 - using TCL View 145
- frequency
 - clocks (Legacy) 185
 - defining for non-clock signals (Legacy) 185

- internal clocks (Legacy) 185
 - setting global 76
- from constraints 150
- FSM Compiler
 - advantages 438
 - enabling 439
- FSM encoding
 - user-defined 397
 - using `syn_enum_encoding` 396
- FSM Explorer
 - when to use 438
- FSM view
 - crossprobing from source file 333
- FSM Viewer 355
 - crossprobing 336
- `fsm.info` file 440
- FSMs
 - See also* FSM Compiler, FSM Explorer
 - attributes and directives 395
 - defining in Verilog 393
 - defining in VHDL 394
 - definition 393
 - optimizing with FSM Compiler 438
 - properties 357
 - state encodings 357
 - transition diagram 355

G

- gated clocks
 - attributes for black boxes 605
 - conversion example 585
 - conversion requirements 585
 - defining (Legacy) 190
 - error messages 596
 - examples 583
 - procedure for fixing 592
 - restrictions 609
- gated-clock conversion
 - excluding elements 600
- Generated Clocks panel
 - using SCOPE 138
- generated-clock conversion 613
- generics
 - extracting from VHDL source code 84
 - passing boolean 46
 - passing integer 47

- global optimization options 74
- GSR resources 619

H

- HDL Analyst
 - See also* RTL view, Technology view
 - critical paths 363
 - crossprobing 266, 330
 - filtering schematics 279, 342
 - Push/Pop mode 312, 315
 - traversing hierarchy with mouse strokes 310
 - traversing hierarchy with Push/Pop mode 253, 312
 - using 274, 338
- HDL Analyst tool
 - deselecting objects 239, 303
 - selecting/deselecting objects 239, 303
- HDL Analyst views
 - highlighting collections 172
- HDL views, annotating timing information 361
- hidden instances
 - consequences of saving 340
 - flattening 351
 - restricting search by hiding 319
 - specifying 340
 - status in other views 340
- hierarchical design
 - expanding logic from nets 345
 - expanding logic from pins 281, 344
- hierarchical instances
 - dissolving 290, 352
 - hiding. *See* hidden instances, Hide Instances command
 - multiple sheets for internal logic 341
 - pin name display 343
 - viewing internal logic 340
- hierarchical objects
 - pushing into with mouse stroke 252, 311
 - traversing with Push/Pop mode 253, 312
- hierarchical projects
 - analyzing 112
 - configuring for synthesis 108
 - multiple implementations 104
 - synthesis options 108

- hierarchical search 319
- hierarchy
 - flattening 290, 349
 - traversing 249, 309
- hierarchy browser
 - clock trees 363
 - controlling display 306
 - crossprobing from 266, 330
 - defined 249, 309
 - finding objects 257, 317
 - traversing hierarchy 309
- hierarchy management (custom folders) 64
- hyper source
 - example 550
 - for IPs 548
 - for prototyping 548
 - IP design hierarchy 548
 - threading signals 549
- I**
- I/O insertion 436, 627
- I/O pads
 - specifying I/O standards 143
- I/O paths
 - false path constraint 155
- I/O standards
 - specifying 143
- I/O Standards panel
 - using SCOPE 139
- I/Os
 - auto-constraining 379
 - constraining 142
 - constraining (Legacy) 193
 - preserving 437, 628
 - Verilog black boxes 384
 - VHDL black boxes 386
- implementation options 72
 - device 72
 - global frequency 76
 - global optimization 74
 - part selection 72
 - specifying results 78
- implementations
 - copying 71
 - deleting 71
 - multiple. *See* multiple implementations.
 - overwriting 71
 - renaming 71
- incremental synthesis
 - compile points 480
 - locked, partition compile points 455
- initializing 408
- initializing RAM 408
- Input and output constraints
 - defining 142
- input constraints, setting 142
- input constraints, setting (Legacy) 193
- Inputs/Outputs panel
 - using SCOPE 139
- Instance Hierarchy tab 260
- instances
 - preserving with syn_noprune 426
 - properties 236, 298
 - properties of pins 236, 298
- integrated P&R 641
 - custom shell script 641
- ILM *See* interface logic models
- interface logic models 459
- interface timing 459
- IP
 - encryption-decryption flow 530
 - re-encryption 533
- IP design hierarchy
 - hyper source 548
- IP encryption flow overview 529
- IP encryption scheme 534
- IP vendors
 - directory structure for package 546
 - encrypting IP 534
 - package file list for encrypted IP flow 546
 - packaging for evaluation 545
 - supplying vendor information 546
- IPs
 - encrypting 534
 - encryption flow 529
 - SYNCore 484
 - SYNCore byte-enable RAMs 500
 - SYNCore counters 520

- SYNCore FIFOs [484](#)
- SYNCore RAMs [491](#)
- SYNCore ROMs [507](#)
- SYNCore subtractors [512](#)
- using hyper source for debug [548](#)
- Isolate Paths command
 - different from Expand Paths [348, 349](#)

J

- job management
 - up-to-date checking [198](#)

K

- key assignments
 - customizing [572](#)
- key block [532](#)
- keywords
 - completing words in Text Editor [34](#)

L

- Lattice
 - constraint file [51](#)
 - forward annotation [51](#)
 - I/O insertion [436](#)
 - macro libraries [618](#)
 - PICs [620](#)
- Lattice netlist [633](#)
- library extensions [40](#)
- license queuing [561](#)
 - blocking-style [561](#)
- license release (synthesis)
 - after P&R [641](#)
- license_release [641](#)
- log file
 - remote access [210](#)
- log files
 - checking information [203](#)
 - retiming report [422](#)
 - setting default display [203](#)
 - state machine descriptions [439](#)
 - viewing [203](#)
- logic
 - expanding between objects [348](#)
 - expanding from net [283, 345](#)

- expanding from pin [281, 344](#)
- logic preservation
 - syn_hier [430](#)
 - syn_keep for nets [426](#)
 - syn_keep for registers [426](#)
 - syn_noprune [426](#)
 - syn_preserve [426](#)
- logical folders
 - creating [64](#)

M

- macro libraries
 - Lattice [618](#)
- manual compile points
 - flow [469](#)
- max_parallel_jobs variable [566](#)
- maximum parallel jobs [565, 576](#)
- MaxParallelJobs variable [565](#)
- memory usage
 - maximizing with HDL Analyst [353](#)
- Message viewer
 - filtering messages [220](#)
 - keyboard shortcuts [219](#)
 - saving filter expressions [221](#)
 - searching [219](#)
 - using [218](#)
 - using the F3 key to search forward [219](#)
 - using the Shift-F3 key to search backward [219](#)
- messagefilter.txt file [228](#)
- messages
 - demoting [225](#)
 - filtering [220](#)
 - promoting [225](#)
 - saving filter information from command line [222](#)
 - saving filter information from GUI [221](#)
 - severity levels [226](#)
 - suppressing [225](#)
 - writing messages to file [223](#)
- mixed designs
 - troubleshooting [46](#)
- mixed language files [43](#)
- mouse strokes
 - pushing/popping objects [252, 310](#)
- multicycle constraints

- forward-annotating 628
- multicycle paths
 - setting constraints 140
 - setting constraints (Legacy) 182
- multiple implementations 70
 - hierarchical projects 104
 - running from project 70
- multiprocessing
 - maximum parallel jobs 565, 576
- multisheet schematics 304
 - for nested internal logic 341
 - searching just one sheet 319
 - transparent instances 304

N

- name spaces
 - output netlist 327
 - technology view 321
- navigating among design views 247, 305
- netlists for different vendors 633
- nets
 - expanding logic from 283, 345
 - preserving for probing with
 - syn_probe 426
 - preserving with syn_keep 426
 - properties 236, 298
 - selecting drivers 348
- new Hierarchy Browser 258
- New property 300
- notes
 - filtering 219
 - sorting 219
- notes, definition 33

O

- objects
 - finding on current sheet 319
 - flagging by property 299
 - selecting/deselecting 303
- open_design
 - with find and expand 164
- optimization
 - for area 415
 - for timing 416

- logic preservation. *See* logic preservation.
- preserving hierarchy 430
- preserving objects 426
 - tips for 414
- orig_inst_of property 301
- output constraints, setting 142
- output constraints, setting (Legacy) 193
- output files 633
 - specifying 78
- output netlists
 - finding objects 327
- overutilization 215

P

- package library, adding 56
- pad types
 - industry standards 143
- parallel jobs 565
- parameter passing 47
 - boolean generics 46
- parameters
 - extracting from Verilog source code 82
- part selection options 72
- path constraints
 - false paths 155
 - false paths (Legacy) 194
- pathnames
 - using wildcards for long names (Find) 322
- paths
 - crossprobing 334
 - tracing between objects 348
 - tracing from net 283, 345
 - tracing from pin 281, 344
- pattern matching
 - Find command (Tcl) 157
- pattern searching 114
- PDF
 - cutting from 35
- PICs 620
- pin names, displaying 343
- pins
 - expanding logic from 281, 344

- properties 236, 298
 - ports
 - false path constraint 155
 - false path constraint (Legacy) 194
 - properties 236, 298
 - POS interface
 - using 151
 - post-synthesis constraints with adc 372
 - preferences
 - crossprobing to place-and-route file 306
 - displaying Hierarchy Browser 306
 - displaying labels 307
 - RTL and Technology views 306
 - sheet size (UI) 306
 - primitives
 - pin name display 343
 - pushing into with mouse stroke 252, 311
 - viewing internal hierarchy 339
 - probes
 - adding in source code 443
 - definition 443
 - retiming 424
 - Product of Sums interface. *See* POS interface
 - project command
 - archiving projects 117
 - copying projects 125
 - unarchiving projects 121
 - project file hierarchy 64
 - project files
 - adding files 58
 - adding source files 54
 - batch mode 558
 - creating 54
 - definition 54
 - deleting files from 58
 - opening 58
 - replacing files in 58
 - updating include paths 63
 - VHDL library 56
 - project status report
 - remote access 210
 - projects
 - archiving 117
 - copying 125
 - restoring archives 121
 - properties
 - displaying with tooltip 236, 298
 - finding objects with Tcl find -filter 158
 - orig_inst_of 301
 - reporting for collections 172
 - viewing for individual objects 236, 298
 - prototyping
 - using hyper source threading 548
 - Push/Pop mode
 - HDL Analyst 310
 - keyboard shortcut 312
 - using 252, 253, 310, 312
- ## Q
- question mark wildcard, Find command 265, 322
- ## R
- RAM inference 398
 - using attributes 399
 - RAMs 408
 - compiling with SYNCORE 491
 - inferring block RAM 401
 - initializing 408
 - SYNCORE 491
 - SYNCORE, byte-enable 500
 - register balancing. *See* retiming
 - register constraints, setting (Legacy) 181
 - registers
 - false path constraint 155
 - false path constraint (Legacy) 194
 - Registers panel
 - using SCOPE 139
 - remote access
 - status reports 210
 - replication
 - controlling 433
 - reports
 - gated clock conversion 594
 - resource sharing 619
 - optimization technique 415
 - overriding option with syn_sharing 436
 - results example 436
 - using 435
 - resource usage 215

resource utilization. *See* resource usage

resynthesis

- compile points 464
- forcing with Resynthesize All 464
- forcing with Update Compile Point Timing Data 464

retiming

- effect on attributes and constraints 423
- example 422
- overview 419
- probes 424
- report 422
- simulation behavior 424

return codes 559

rom.info file 312

ROMs

- SYNCore 507
- viewing data table 312

RTL view

See also HDL Analyst

- analyzing clock trees 363
- crossprobing collection objects 167
- crossprobing description 330
- crossprobing from 331
- crossprobing from Text Editor 333
- defined 297
- description 296
- filtering 279, 342
- finding objects with Find 319
- finding objects with Hierarchy Browser 257, 317
- flattening hierarchy 290, 349
- highlighting collections 172
- opening 233, 298
- selecting/deselecting objects 303
- setting preferences 306
- state machine implementation 440
- traversing hierarchy 309

RTL views

- hierarchical projects 112
- hierarchical subprojects 103

running P&R

- license release (synthesis) 641

S

Schematic Options 260

schematic view

setting preferences (New Analyst) 247

schematics

- multisheet. *See* multisheet schematics
- page size 306
- selecting/deselecting objects 239, 303

SCOPE

- adding attributes 89
- adding probe insertion attribute 444
- Attributes panel 139
- case sensitivity for Verilog designs 157
- Clocks panel 138
- collections compared to Tcl script window 165
- Collections panel 138
- Compile Points panel 139
- creating compile-point constraint file 476
- defining compile points 472
- Delay Paths panel 139
- drag and drop 147
- editing operations 148
- Generated Clocks panel 138
- I/O pad type 143
- I/O Standards panel 139
- Inputs/Outputs panel 139
- multicycle paths 154
- Registers panel 139
- setting compile point constraints 476
- setting constraints (FDC) 132
- specifying constraints 138
- state machine attributes 395
- TCL View 139

SCOPE editor

- using 132

SCOPE panels

- entering and editing constraints 138

SCOPE TCL View

- using 145

search

- browsing objects with the Find command 318
- browsing with the Hierarchy Browser 257, 317
- finding objects on current sheet 319
- setting limit for results 321
- setting scope 320
- using the Find command in HDL Analyst views 319

search in Analyst

- browsing objects with the Find command 262
- See also* search
- set command
 - collections 173
- set_option command 74
- sheet connectors
 - navigating with 305
- sheet size
 - setting number of objects 306
- Shift-F3 key
 - Message Viewer 219
- Show Cell Interior option 339
- Show Context command
 - different from Expand 341
 - using 341
- signals
 - threading with hyper source. *See* hyper source
- simulation, effect of retiming 424
- single-port RAMs
 - SYNCore parameters 496
- slack 366
 - setting margins 363
- slack time display 360
- Slow property 300
- source code
 - commenting with synthesis on/off 85
 - crossprobing from Tcl window 336
 - defining FSMs 393
 - fixing errors 36
 - opening automatically to crossprobe 332
 - optimizing 414
- source files
 - See also* Verilog, VHDL.
 - adding comments 35
 - adding files 54
 - checking 32
 - column editing 35
 - copying examples from PDF 35
 - creating 28
 - crossprobing 333
 - editing 34
 - editing operations 34
 - mixed language 43
 - specifying default encoding style 84
 - specifying top level file for mixed language projects 44
 - specifying top-level file 84
 - state machine attributes 395
 - using bookmarks 35
- special characters
 - Tcl collections 171
- STA 368
- STA, generating custom timing reports 368
- STA, using analysis design constraints (adc) 371
- stand-alone timing analyst. *See* STA
- state machines
 - See also* FSM Compiler, FSM Explorer, FSM viewer, FSMs.
 - attributes 395
 - descriptions in log file 439
 - implementation 440
 - parameter and 'define comparison 394
- statemachine.info file 357
- sub-projects
 - creating 95
 - linking updates to top level 97
 - multiple implementations 104
 - synchronizing device options with top level 111
- subprojects
 - block-based 95
 - compiling 101
 - nested 98
- subtractors
 - SYNCore 512
- syn_allow_retiming
 - using for retiming 420
- syn_allowed_resources
 - compile points 478
- syn_encoding attribute 396
- syn_enum_encoding directive
 - FSM encoding 396
- syn_force_pad attribute
 - using 436, 627
- syn_hier attribute
 - controlling flattening 429
 - preserving hierarchy 430
 - using with compile points 477

- syn_isclock
 - black box clock pins 392
 - syn_keep
 - inferring Lattice PICs 620
 - replicating redundant logic 427
 - syn_keep attribute
 - preserving nets 426
 - preserving shared registers 426
 - syn_keep directive
 - effect on buffering 433
 - syn_macro
 - specifying encrypted IP as white box 543
 - syn_maxfan attribute
 - setting fanout limits 431
 - syn_noprune directive
 - preserving instances 426
 - syn_preserve
 - effect on buffering 433
 - preserving power-on for retiming 421
 - syn_preserve directive
 - preserving FSMs from optimization 395
 - preserving logic 426
 - syn_probe attribute 443
 - inserting probes 443
 - preserving nets 426
 - syn_reference_clock constraint (Legacy) 176
 - syn_replicate attribute
 - using buffering 433
 - syn_sharing directive
 - overriding default 436
 - syn_state_machine directive
 - using with value=0 441
 - SYN_TCL_HOOKS environment variable 571
 - syn_tco attribute
 - adding in SCOPE 391
 - syn_tco directive 390
 - adding black box constraints 389
 - syn_tpd attribute
 - adding in SCOPE 391
 - syn_tpd directive 390
 - adding black box constraints 389
 - syn_tsu attribute
 - adding in SCOPE 391
 - syn_tsu directive 390
 - adding black box constraints 389
 - syn_useioff
 - preventing flops from moving during retiming 421
 - SYNCore
 - adders 512
 - counters 520
 - FIFO compiler 484
 - RAMs 491
 - RAMs, byte-enable 500
 - RAMs, dual-port parameters 497
 - RAMs, single-port parameters 496
 - ROMs 507
 - ROMs, parameters 511
 - subtractors 512
 - synhooks
 - automating message filtering 223
 - synhooks.tcl file 571
 - Synopsys
 - FPGA product family 14
 - Synplify Pro synthesis tool
 - overview 14
 - SYNPLIFY_REMOTE_REPORT_LOCATION 211
 - syntax
 - checking source files 32
 - syntax check 32
 - synthesis
 - hierarchical projects 108
 - synthesis check 32
 - synthesis_on/off
 - using 85
 - SystemVerilog keywords
 - context help 30
- ## T
- ta file 368
 - Tcl
 - max_parallel_jobs variable 566
 - tcl callbacks
 - customizing key assignments 572
 - Tcl commands
 - batch script 560

- running 564
- Tcl expand
 - using 156
- Tcl expand command
 - crossprobing objects 167
 - usage tips 161
 - using in SCOPE 166
- Tcl files 564
 - creating 566
 - for bottom-up synthesis 570
 - guidelines 48
 - naming conventions 49
 - recording from commands 565
 - synhooks.tcl 571
 - using variables 568
 - wildcards 49
- Tcl find
 - batch mode 164
 - filtering results by property 158
 - search patterns 156
 - using 156
- Tcl find command
 - annotating properties 158
 - case sensitivity 157
 - crossprobing objects 167
 - database differences 166
 - pattern matching 157
 - Tcl window vs SCOPE 165
 - usage tips 160
 - useful -filter examples 160
 - using in SCOPE 166
- Tcl Script window
 - crossprobing 336
 - message viewer 218
- Tcl script window
 - collections compared to SCOPE 165
- Tcl scripts
 - See Tcl files.
- TCL View 145
 - using 145
 - using SCOPE 139
- tclcmd 558
- Technology view
 - See also HDL Analyst
 - critical paths 363
 - crossprobing 330, 331
 - crossprobing collection objects 167
 - crossprobing from source file 333
 - filtering 279, 342
 - finding objects 321
 - finding objects with Find 319
 - finding objects with Hierarchy Browser 257, 317
 - flattening hierarchy 290, 349
 - general description 296
 - highlighting collections 172
 - opening 298
 - selecting/deselecting objects 303
 - setting preferences 306
 - state machine implementation in 440
 - traversing hierarchy 309
- text editor
 - built-in 33
 - external 38
 - using 33
- Text Editor view
 - crossprobing 331
- Text Editor window
 - colors 36
 - crossprobing 36
 - fonts 36
- text files
 - crossprobing 333
- The Synopsys FPGA Product Family 14
- third-party vendor tools
 - invoking 634
- through constraints 151
 - AND lists 152
 - OR lists 151
- time stamp, checking on files 59
- timing analysis 360
- timing analysis using STA 368
- timing budgeting
 - compile points 459
- timing constraints (Legacy) 176
- timing exceptions, adding constraints after synthesis 372
- timing exceptions, modifying with adc 372
- timing failures 366
- timing information commands 360
- timing information in HDL views 361
- timing information, critical paths 365

- timing optimization [416](#)
- timing report, stand-alone [368](#)
- timing reports
 - specifying format options [80](#)
- timing reports, custom [368](#)
- tips
 - memory usage [353](#)
- to constraints
 - specifying [151](#)
- tool tags
 - creating [634](#)
 - definition [634](#)
- top level
 - specifying [84](#)
- top-down design flow
 - compile point advantages [448](#)
- top-down hierarchical development flow
 - subprojects [95](#)
- transparent instances
 - flattening [350](#)
 - lower-level logic on multiple sheets [304](#)

U

- up-to-date checking [198](#)
 - copying job logs to log file [201](#)
 - limitations [202](#)

V

- vendor-specific netlists [633](#)

Verilog

- 'define statements [82](#)
- adding attributes and directives [88](#)
- adding probes [443](#)
- black boxes [384](#)
- black boxes, instantiating [384](#)
- case sensitivity for Tcl Find
 - command [157](#)
- checking source files [32](#)
- choosing a compiler [81](#)
- creating source files [28](#)
- crossprobing from HDL Analyst
 - view [331](#)
- defining FSMs [393](#)
- defining state machines with parameter
 - and 'define [394](#)
- editing operations [34](#)

- extracting parameters [82](#)
- include paths, updating [63](#)
- initializing RAMs [408](#)
- mixed language files [43](#)
- specifying compiler directives [82](#)
- specifying top-level module [84](#)
- using library extensions [40](#)

Verilog 2001

- setting global option from the Project
 - view [81](#)
- setting option per file [81](#)

Verilog library files

- using library extensions [40](#)

Verilog macro libraries

- Lattice [618](#)

Verilog model (.vmd) [459](#)

VHDL

- adding attributes and directives [87](#)
- adding probes [443](#)
- black boxes [386](#)
- black boxes, instantiating [386](#)
- case sensitivity for Tcl Find
 - command [157](#)
- checking source file [32](#)
- constants [84](#)
- creating source files [28](#)
- crossprobing from HDL Analyst
 - view [331](#)
- defining FSMs [394](#)
- editing operations [34](#)
- extracting generics [84](#)
- file order in mixed designs [46](#)
- global signals in mixed designs [46](#)
- initializing RAMs with variable
 - declarations [411](#)
- initializing with signal declarations [409](#)
- mixed language files [43](#)
- specifying top-level entity [84](#)

VHDL files

- adding library [56](#)
- adding third-party package library [56](#)

VHDL macro libraries

- Lattice [618](#)

vi text editor [38](#)

- virtual clock, setting (Legacy) [181](#)

W

warning messages

definition [33](#)

warnings

feedback muxes [417](#)

filtering [219](#)

sorting [219](#)

Watch window [214](#)

moving [214](#), [218](#)

multiple implementations [71](#)

resizing [214](#), [218](#)

wildcards

effect of search scope [322](#)

Find command (Tcl) [157](#)

message filter [221](#)

wildcards (Find)

examples [324](#)

how they work [265](#), [322](#)

WNID_UG_PNR_SCRIPTS [640](#)