

Lattice Radiant Software 2022.1

User Guide



November 21, 2022

Copyright

Copyright © 2022 Lattice Semiconductor Corporation. All rights reserved. This document may not, in whole or part, be reproduced, modified, distributed, or publicly displayed without prior written consent from Lattice Semiconductor Corporation (“Lattice”).

Trademarks

All Lattice trademarks are as listed at www.latticesemi.com/legal. Synopsys and Synplify Pro are trademarks of Synopsys, Inc. Aldec and Active-HDL are trademarks of Aldec, Inc. Modelsim and Questa are trademarks or registered trademarks of Siemens Industry Software Inc. or its subsidiaries in the United States or other countries. All other trademarks are the property of their respective owners.

Disclaimers

NO WARRANTIES: THE INFORMATION PROVIDED IN THIS DOCUMENT IS “AS IS” WITHOUT ANY EXPRESS OR IMPLIED WARRANTY OF ANY KIND INCLUDING WARRANTIES OF ACCURACY, COMPLETENESS, MERCHANTABILITY, NONINFRINGEMENT OF INTELLECTUAL PROPERTY, OR FITNESS FOR ANY PARTICULAR PURPOSE. IN NO EVENT WILL LATTICE OR ITS SUPPLIERS BE LIABLE FOR ANY DAMAGES WHATSOEVER (WHETHER DIRECT, INDIRECT, SPECIAL, INCIDENTAL, OR CONSEQUENTIAL, INCLUDING, WITHOUT LIMITATION, DAMAGES FOR LOSS OF PROFITS, BUSINESS INTERRUPTION, OR LOSS OF INFORMATION) ARISING OUT OF THE USE OF OR INABILITY TO USE THE INFORMATION PROVIDED IN THIS DOCUMENT, EVEN IF LATTICE HAS BEEN ADVISED OF THE POSSIBILITY OF SUCH DAMAGES. BECAUSE SOME JURISDICTIONS PROHIBIT THE EXCLUSION OR LIMITATION OF CERTAIN LIABILITY, SOME OF THE ABOVE LIMITATIONS MAY NOT APPLY TO YOU.

Lattice may make changes to these materials, specifications, or information, or to the products described herein, at any time without notice. Lattice makes no commitment to update this documentation. Lattice reserves the right to discontinue any product or service without notice and assumes no obligation to correct any errors contained herein or to advise any user of this document of any correction if such be made. Lattice recommends its customers obtain the latest version of the relevant information to establish that the information being relied upon is current and before ordering any products.

Type Conventions Used in This Document

Convention	Meaning or Use
Bold	Items in the user interface that you select or click. Text that you type into the user interface.
<i><Italic></i>	Variables in commands, code syntax, and path names.
Ctrl+L	Press the two keys at the same time.
<code>Courier</code>	Code examples. Messages, reports, and prompts from the software.
<code>...</code>	Omitted material in a line of code.
<code>.</code> <code>.</code> <code>.</code>	Omitted lines in code and report examples.
[]	Optional items in syntax descriptions. In bus specifications, the brackets are required.
()	Grouped items in syntax descriptions.
{ }	Repeatable items in syntax descriptions.
	A choice between items in syntax descriptions.

Contents

Chapter 1	Introduction	9
	Radiant Software Overview	9
	User Guide Organization	10
Chapter 2	Getting Started	11
	Prerequisites	11
	Creating a New Project	12
	Opening an Existing Project	15
	Importing a Lattice Diamond Project	16
	Next Steps	16
Chapter 3	Design Environment Fundamentals	17
	Overview	17
	Project-Based Environment	18
	Process Flow	19
	Shared Memory	20
	Context-Sensitive Data Views	21
Chapter 4	User Interface Operation	23
	Start Page	23
	Menus and Toolbars	24
	Reports and Messages Views	24
	File List Views	25
	Tool View Area	26
	Output and Tcl Console	27
	Basic UI Controls	27
	File List	28
	Source Template	28
	IP Catalog	30

	Process	30
	Task Detail View	31
	Hierarchy	32
	Reports	32
	Tool Views	33
	Tcl Console	34
	Output	34
	Message	35
	Find Results	35
	Common Tasks	35
	Controlling Views	36
	Cross-Probing	37
Chapter 5	Working with Projects	41
	Overview	41
	Implementations	43
	Adding Implementations	43
	Cloning Implementations	44
	Input Files	45
	Pre-Synthesis Constraint Files	46
	Post-Synthesis Constraint Files	46
	Debug Files	47
	Script Files	47
	Analysis Files	48
	Programming Files	48
	Strategies	48
	Area	50
	Timing	51
	User-Defined	52
	Constraint Propagation Options	52
	Synplify Pro Options	52
	LSE Options	58
	Post-Synthesis Options	64
	Post-Synthesis Timing Analysis Options	65
	Map Design Options	66
	Map Timing Analysis Options	67
	Place & Route Design Options	68
	Place & Route Timing Analysis Options	71
	IO Timing Analysis Options	71
	Timing Simulation Options	72
	Bitstream Options	73
	Common Tasks	75
	Creating a Project	75
	Changing the Target Device	75
	Setting the Top Level of the Design	75
	Editing Files	76
	Saving Project Data	76
Chapter 6	Radiant Software Design Flow	77
	Overview	77

	Design Flow Processes	78
	Running Processes	78
	IP Encryption Flow	79
	HDL File Encryption Flow	81
	HDL File Encryption Steps	82
	Implementation Flow and Tasks	85
	Synthesis Constraint Creation	85
	Constraint Creation	87
	Simulation Flow	88
	Simulation Wizard Flow	89
Chapter 7	Working with Tools and Views	93
	Overview	93
	View Menu Highlights	93
	Start Page	94
	Reports	94
	Tools	96
	Timing Constraint Editor	96
	Unified Constraints Flow	98
	Constraint Propagation	99
	Device Constraint Editor	99
	Specifying Virtual I/O Ports	110
	Checking the Virtual I/O Ports	111
	Verifying Virtual I/O Ports	112
	Netlist Analyzer	113
	Physical Designer	114
	Timing Analyzer	117
	Using Standalone Timing Analyzer	118
	Reveal Inserter	121
	Configuring User Memory Setup	123
	Configuring User Control Register Setup	124
	Configuring User Status Register Setup	124
	Reveal Analyzer	125
	Reveal Controller	129
	Power Calculator	129
	ECO Editor	130
	Programmer	131
	Run Manager	132
	Synplify Pro for Lattice	132
	Mentor ModelSim	133
	Simulation Wizard	133
	Source Template	134
	IP Catalog	134
	IP Packager	135
	SEI Editor	136
	Common Tasks	136
	Controlling Tool Views	136
	Using Zoom Controls	138
	Displaying Tool Tips	139
	Setting Display Options	139

Chapter 8	Command Line Reference Guide	141
	Command Line Program Overview	141
	Command Line Basics	143
	Command Line Data Flow	143
	Command Line General Guidelines	144
	Command Line Syntax Conventions	145
	Setting Up the Environment to Run Command Line	146
	Invoking Core Tool Command Line Programs	147
	Invoking Core Tool Command Line Tool Help	147
	Command Line Tool Usage	148
	Running cmpl_libs.tcl from the Command Line	149
	Running HDL Encryption from the Command Line	151
	Running SYNTHESIS from the Command Line	158
	Running Postsyn from the Command Line	164
	Running MAP from the Command Line	165
	Running PAR from the Command Line	167
	Running Timing from the Command Line	173
	Running Backannotation from the Command Line	176
	Running Bit Generation from the Command Line	178
	Running Programmer from the Command Line	181
	Running Various Utilities from the Command Line	184
	Using Command Files	188
	Using Command Line Shell Scripts	190
Chapter 9	Tcl Command Reference Guide	193
	Running the Tcl Console	194
	Accessing Command Help in the Tcl Console	196
	Creating and Running Custom Tcl Scripts	196
	Running Tcl Scripts When Launching the Radiant Software	199
	Radiant Software Tool Tcl Command Syntax	200
	Radiant Software Tcl Console Commands	200
	Radiant Software Timing Constraints Tcl Commands	202
	Radiant Software Physical Constraints Tcl Commands	205
	Radiant Software Project Tcl Commands	206
	Simulation Libraries Compilation Tcl Commands	214
	Reveal Inserter Tcl Commands	216
	Reveal Analyzer Tcl Commands	221
	Power Calculator Tcl Commands	227
	Programmer Tcl Commands	229
	Engineering Change Order Tcl Commands	231
	Control Message Tcl Commands	232
Chapter 10	Advanced Topics	233
	Shared Memory Environment	233
	Clear Tool Memory	233
	Environment and Tool Options	234
	Batch Tool Operation	235
	Tcl Scripts	235
	Creating Tcl Scripts from Command History	235
	Creating Tcl Scripts from Scratch	236
	Sample Tcl Script	236

Running Tcl Scripts	237
Project Archiving	237
File Descriptions	238
Revision History	241

Chapter 1

Introduction

Lattice Radiant® software is the leading-edge software design environment for cost-sensitive, low-power Lattice Field Programmable Gate Arrays (FPGA) architectures. The Radiant software integrated tool environment provides a modern, comprehensive user interface for controlling the Lattice Semiconductor FPGA implementation process. Its combination of new and enhanced features allows users to complete designs faster, more easily, and with better results than ever before.

This user guide describes the main features, usage, and key concepts of the Radiant software design environment. It should be used in conjunction with the Release Notes and reference documentation included with the product software. The Release Notes document is also available on the Lattice Web site and provides a list of supported devices.

Radiant Software Overview

The Radiant software uses an expanded project-based design flow and integrated tool views so that design alternatives and what-if scenarios can easily be created and analyzed. The *Implementations* and *Strategies* concepts provide a convenient way for users to try alternate design structures and manage multiple tool settings.

System-level information—including process flow, hierarchy, and file lists—is available, along with integrated HDL code checking and consolidated reporting features.

A fast Timing Analysis loop and Programmer provide capabilities in the integrated framework. The cross-probing feature and the shared memory architecture ensure fast performance and better memory utilization.

The Radiant software is highly customizable and provides Tcl scripting capabilities from either its built-in console or from an external shell.

The Radiant software has many of the same features as Lattice Diamond software, and adds new features, such as:

- ▶ Constraints support utilizing industry standard SDC format.
- ▶ Efficient, easy-to-use integrated graphical user interface (GUI) with a new look-and-feel that gives users more efficient access to popular tools.

- ▶ Unified timing analysis engine with enhanced timing reports for faster design timing closure.

User Guide Organization

This user guide contains all the basic information for using the Radiant software. It is organized in a logical sequence from introductory material, through operational descriptions, to advanced topics.

Key concepts and work flows are explained in [“Design Environment Fundamentals” on page 17](#) and [“Radiant Software Design Flow” on page 77](#).

Basic operation of the design environment is described in [“User Interface Operation” on page 23](#).

The chapter [“Working with Projects” on page 41](#) shows how to set up project implementations and strategies.

The chapter [“Working with Tools and Views” on page 93](#) describes the many tool views available.

Chapter 2

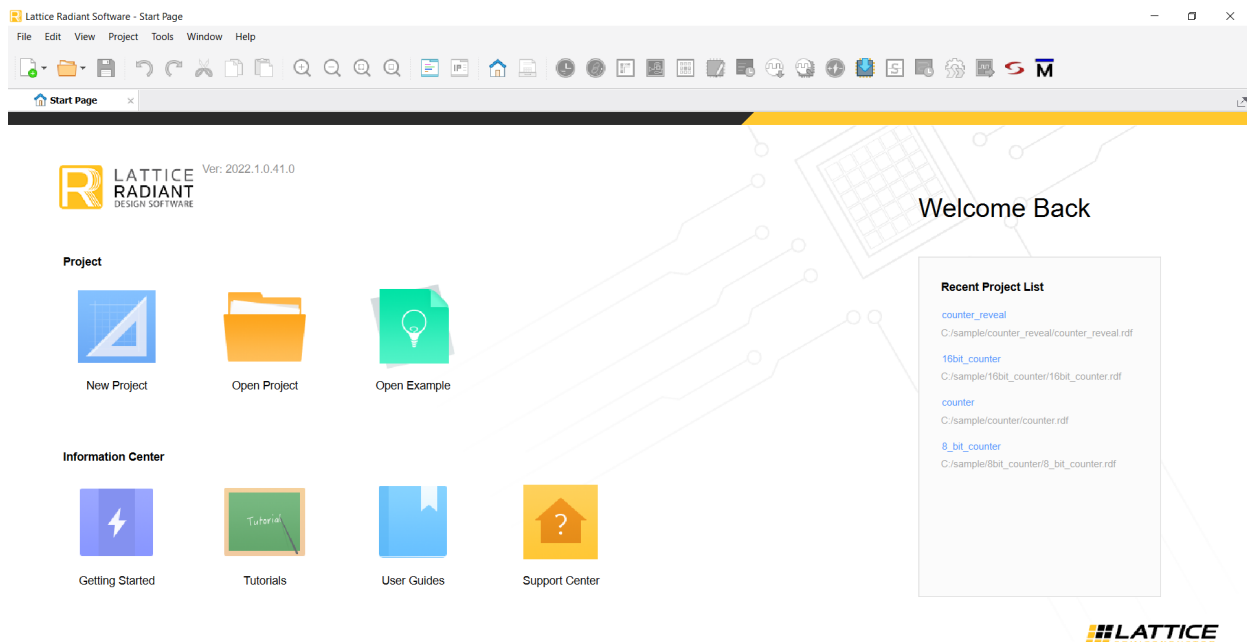
Getting Started

This chapter explains how to run the Radiant software and open or create a project. For more information about project fundamentals, see the chapters [“Design Environment Fundamentals” on page 17](#) and [“Working with Projects” on page 41](#).

Prerequisites

To run the Radiant software, select **Radiant Software** from the installation location. This opens the default Start Page, shown in Figure 1.

Figure 1: Default Start Page



Creating a New Project

A project is a collection of all files necessary to create and download your design to the selected device. The New Project wizard guides you through the steps of specifying a project name and location, selecting a target device, and adding existing sources to the new project.

Note

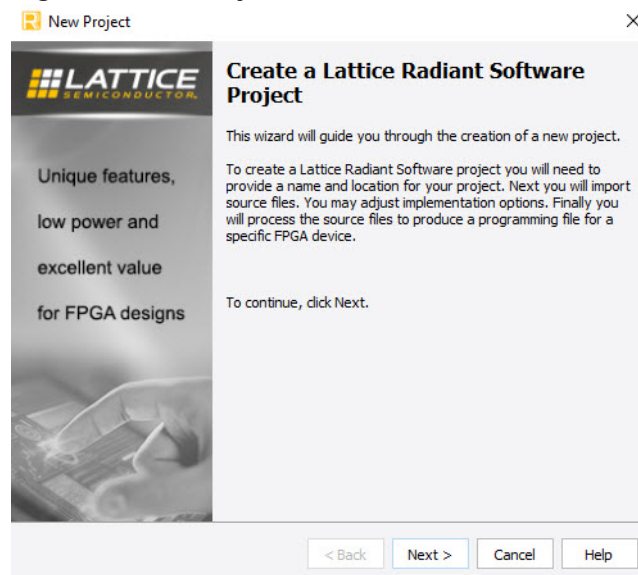
Do not place more than one project in the same directory.

To create a new project:

1. From the Radiant main window, click the **New Project**  button, or choose **File > New > Project**.

The New Project confirmation window opens, shown in Figure 2.

Figure 2: New Project Confirmation Window



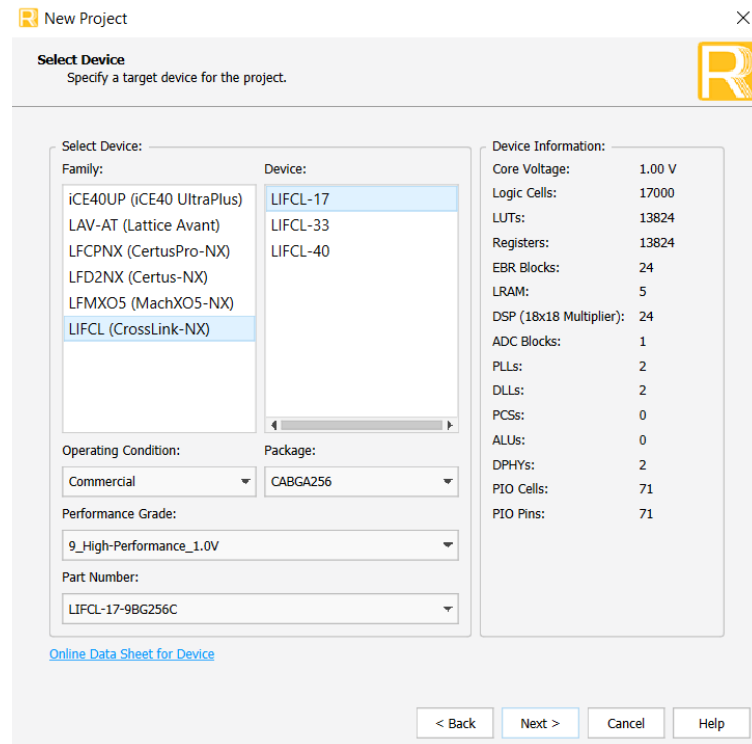
2. Click **Next**. The New Project wizard opens, shown in Figure 3.

Figure 3: New Project Wizard

3. In the Project Name dialog box, do the following:
 - ▶ Under Project, specify the name for the new project.
File names for Radiant software projects and project source files must start with a letter (A-Z, a-z) and must contain only alphanumeric characters (A-Z, a-z, 0-9) and underscores (_). Spaces are allowed.
 - ▶ To specify a location for your project, click **Browse**. In the Project Location dialog box, you can specify a desired location.
 - ▶ Under Implementation, specify the name for the first version of the project. You can have more than one version, or “implementation,” of the project to experiment with. For more information on implementations, refer to [“Implementations” on page 43](#).
 - ▶ To create a sub-directory with the same name as your location directory, click **Create Subdirectory**. This will allow you to keep your project implementations separate. If this box is left unchecked, no sub-directory will be created in the project directory.
 - ▶ When you finish, click **Next**.
4. In the Add Source dialog box, do the following if you have an existing source file that you want to add to the project. If there are no existing source files, click **Next**.
 - a. Click **Add Source**. You can import HDL files at this time. In the Import File dialog box, browse for the source file you want to add, select it, and click **Open**.
The source file is then displayed in the Source files field.
 - b. Repeat the above to add more files.
 - c. To copy the added source files to the implementation directory, select **Copy source to implementation directory**. If you prefer to reference these files, clear this option.

- d. To create empty Lattice Design Constraint (.ldc) file and Physical Constraint File (.pdc) files that can be edited at a later time, select **Create empty constraint files**. Refer to the chapter [“Implementations” on page 43](#) for more information about constraint files.
 - e. When you finish, click **Next**.
5. In the Select Device dialog box, shown in Figure 4, select a device family and a specific device within that family. Then choose the options you want for that device. When you finish, click **Next**.

Figure 4: Select Device Dialog Box



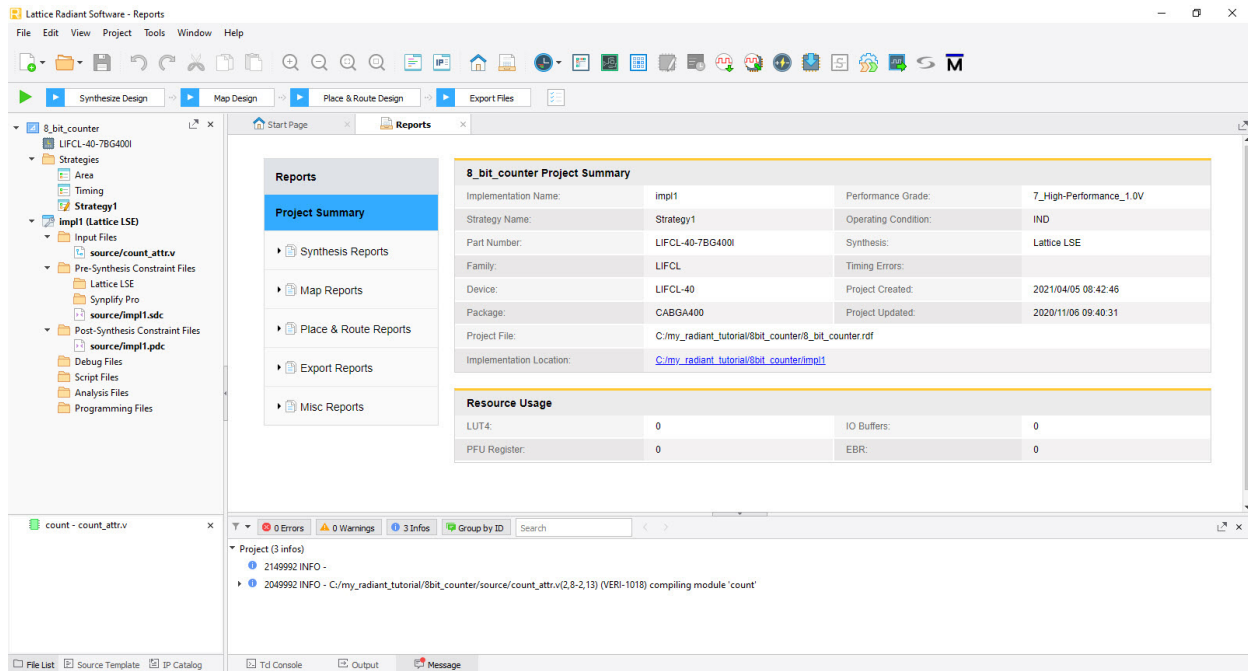
6. In the Select Synthesis Tool dialog box, select the synthesis tool that you want to use. This choice can be changed at any time. When you finish, click **Next**.
7. In the Project Information dialog box, make sure the project settings are correct.

Note

If you want to change some of the settings, click **Back** to modify them in the previous dialog boxes of the New Project Wizard.

Click **Finish**. The newly created project, shown in Figure 5, is now created and open.

Figure 5: Opened Project



Select the **File List** tab under the left pane, to view the Test project file list.

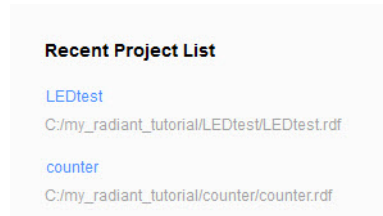
To close a project, choose **File > Close Project**.

Opening an Existing Project

Use one of the following methods to open an existing Radiant software project:

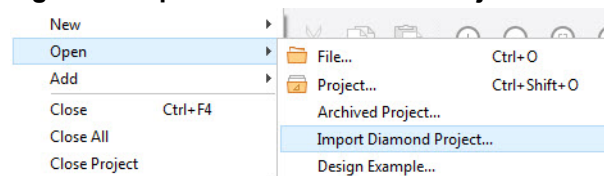
- ▶ On the Start Page, click the **Open Project**  button.
- ▶ From the File menu, choose **Open > Project**.
- ▶ On the Start Page, select the desired project from the Recent Projects List. Alternatively, choose a recent project from the **File > Recent Projects** menu.

You can use the Options dialog box to increase the number of projects that are shown in the Recent Projects list and to automatically load the previous project at startup. Choose **Tools > Options** to open the dialog box. To increase the number of recent projects listed, click the **General** tab and enter a number for “Maximum items shown in Recent Project List” (up to 32). To automatically open the previous project during startup, click the **Startup** tab and then choose **Open Previous Project** from the “At Lattice Radiant Software startup” menu.

Figure 6: Recent Project List

Importing a Lattice Diamond Project

To import a Lattice Diamond project into the Radiant software, choose **File > Open > Import Diamond Project**.

Figure 7: Import Lattice Diamond Project

The file browser applies an *.ldf file filter to help you find Lattice Diamond project files. The Lattice Diamond project is converted to a Radiant project.

For more information about importing Lattice Diamond projects into the Radiant software, refer to the [Lattice Radiant Software Guide for Diamond Users](#).

Next Steps

After you have a project opened in the Radiant software, you can go sequentially through the rest of this user guide to learn how to work with the entire design environment, or you can go directly to any topic of interest.

- ▶ The chapters [“Design Environment Fundamentals” on page 17](#) and [“Radiant Software Design Flow” on page 77](#) provide explanations of key concepts.
- ▶ [“User Interface Operation” on page 23](#) provides descriptions of the functions and controls that are available in the Radiant software environment.
- ▶ The chapters [“Working with Projects” on page 41](#) and [“Working with Tools and Views” on page 93](#) explain how to run processes and use the design tools.
- ▶ [“Tcl Command Reference Guide” on page 193](#) provides an introduction to the scripting capabilities available, plus command-line shell examples.
- ▶ [“Advanced Topics” on page 233](#) provides further details about environment options, shared memory, and Tcl scripting.

Chapter 3

Design Environment Fundamentals

This chapter provides background and discussion on the technology and methodology underlying the Radiant software design environment. Important key concepts and terminology are defined.

Overview

Understanding some of the fundamental concepts behind the Radiant software framework technology will increase your proficiency with the tool and allow you to quickly come up to speed on its use.

The Radiant software is a next-generation software design environment that uses a new project-based methodology. A single project can contain multiple implementations and strategies to provide easily managed alternate design structures and tool settings.

The process flow is managed at a system level with run management controls and reporting. Context-sensitive views ensure that you only see the data that is available for the current state in the process flow.

The shared memory technology enables many of the advanced functions in the Radiant software. Easy cross-probing between tool views and faster process loops are among the benefits.

Note

By loading the Radiant software multiple times, you can run different Radiant projects simultaneously. However, you must not load the same project in more than one Radiant software instance, as software conflicts can occur.

The Radiant software can also be run remotely. Refer to the [Lattice Radiant Software Installation Guide for Windows](#) or [Lattice Radiant Software Installation Guide for Linux](#) for more information.

Project-Based Environment

A project in the Radiant software consists of the following file types:

- ▶ HDL source files
- ▶ Constraint files
- ▶ Reveal debug files
- ▶ Script files for simulation
- ▶ Analysis files for power calculation and timing analysis
- ▶ Programming files

The Radiant software also includes settings for the targeted device and the different tools. The project data is organized into implementations, which define the project structural elements, and strategies, which are collections of tool settings.

The following File List shows the items in a sample project.

Figure 8: File List



Each item that is displayed in **bold** means that it has been selected as the active item for an implementation. An implementation displayed in **bold** means that it has been selected as the currently active implementation for the project. Your project must have one active implementation, and the implementation must have one active strategy. Optional items, such as Reveal hardware debugger files, can be set as active or inactive.

The project is the top-level organizational element in the Radiant software, and it can contain multiple implementations and multiple strategies. This enables you to try different design approaches within the same project. If you want to have a Verilog version of your design, for example, make an implementation that consists of only the Verilog source files. If you want another version of the design with primarily Verilog files but a Structural Verilog (.vm) netlist for one module, create a new implementation using the Verilog and .vm source files. Each implementation can have Verilog, VHDL or Structural Verilog source or mixed of them. The same project and design is used, but with a different set of modular blocks.

Similarly, if you want to try different implementation tool options, you can create a new strategy with the new option values.

You manage these multiple implementations and strategies for your project by setting them as active. There can only be a single active implementation with its one active strategy at a time.

Process Flow

A process is a specific task in the overall processing of a source or project. Typical processing tasks include synthesizing, mapping, placing, and routing. You can view the available processes for a design in the Process Toolbar.

Figure 9: Process Toolbar



Click the Task Detail View  to see detailed information of the processes.

Processes are grouped into categories according to their functions.

▶ Synthesize Design

Click on this process and Lattice Synthesis Engine (LSE) runs to synthesize the design. By default, this process runs the LSE tool.

If you are using Synplify Pro, choose Synplify Pro as the synthesis tool (**Project > Active Implementation > Select Synthesis Tool**).

▶ Post-Synthesis Timing Analysis

Runs timing analysis after the Synthesize Design process.

▶ Post-Synthesis Simulation File

Generates a netlist file `<file_name>_syn.vo` used for functional verification.

▶ Map Design

This process maps a design to an FPGA. Map Design is the process of converting a design represented as a network of device-independent components (such as gates and flip-flops) into a network of device-specific components (for example, configurable logic blocks).

▶ Map Timing Analysis

Runs timing analysis after the Map Design process.

▶ Place & Route Design

After a design has undergone the necessary translation to bring it into the Unified Database (.udb) format, you can run the Place & Route Design process. This process takes a mapped physical design .udb file, places and routes the design, and then outputs a file that can then be processed by the design implementation tools.

▶ **Place & Route Timing Analysis**

Runs timing analysis after Place & Route process.

▶ **I/O Timing Analysis**

Runs I/O timing analysis that allows you to view the path delay tables and Timing Analyzer report of your timing constraints after placement and routing.

▶ **Export Files**

You can check the desired file you want to export and run this process.

▶ **Bitstream File**

This process takes a fully routed physical design as input and produces a configuration bitstream (bit images). The bitstream file contains all of the configuration information from the physical design defining the internal logic and interconnections of the FPGA, as well as device-specific information from other files associated with the target device.

▶ **IBIS Model**

This process generates a design-specific IBIS (I/O Buffer Information Specification) model file (<project_name>.ibs).

IBIS models provide a standardized way of representing the electrical characteristics of a digital IC's pins (input, output, and I/O buffers).

▶ **Gate-Level Simulation File**

This process backannotates the routed design with timing information so that you may run a simulation of your design. The backannotated design is a Verilog netlist.

The Reports view allows you to examine and print process reports.

Messages are displayed in the Messages window at the bottom of the Radiant software main window.

The process status icons are defined as follows:

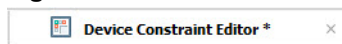
-  Process in initial state (not processed)
-  Process completed successfully
-  Process completed with unprocessed subtasks
-  Process failed

Shared Memory

The Radiant software uses a shared memory architecture. All tool and data views look at the same design data at any point in time. This means that when you change a data element in one view of your design, all other views will see the change, whether they are active or not.

When project data has been changed but not yet saved, an asterisk (*) is displayed in the title tab of the view.

Figure 10: Title Tab with Changed Content Indication



Notice that the asterisks indicating changed data will appear in all views referencing the changed data.

If a tool view becomes unavailable, the Radiant software environment will need to be closed and restarted.

Context-Sensitive Data Views

The data in shared memory reflects the state or context of the overall process flow. This means that views such as Device Constraint Editor Spreadsheet View will display only the data that is currently available, depending on process steps that have been completed.

For example, Figure 11 shows the Process flow before Synthesis. Therefore, Spreadsheet View shows no IO Type or PULLMODE.

Figure 11: Process Completed Before Synthesis

Name	Group By	Pin	BANK	IO_TYPE	Virtual	CLAMP
▼ All Port	N/A	N/A	N/A	N/A	N/A	N/A
▼ Unconnected	N/A	N/A	N/A	N/A	N/A	N/A
▶ c[0]	N/A			N/A	FALSE	N/A
▶ c[1]	N/A			N/A	FALSE	N/A
▶ c[2]	N/A			N/A	FALSE	N/A

Port Pin Global SSO

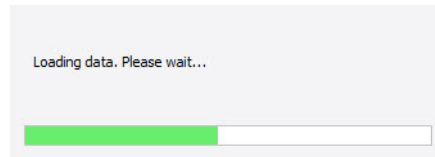
After Synthesis has been completed, Spreadsheet View displays IO Type and PULLMODE assignments, as shown in Figure 12.

Figure 12: Process Completed Through Synthesis

Name	Group By	Pin	BANK	IO_TYPE	Virtual	CLAMP
▼ All Port	N/A	N/A	N/A		N/A	N/A
▼ Input	N/A	N/A	N/A	N/A	N/A	N/A
rst	N/A	(N15)	(2)	LVC MOS33(...	FALSE	ON(ON)
▼ Clock	N/A	N/A	N/A	N/A	N/A	N/A
clk	N/A	(E12)	(0)	LVC MOS33(...	FALSE	ON(ON)
▼ Output	N/A	N/A	N/A	N/A	N/A	N/A
c[7]	N/A	(N6)	(6)	LVC MOS33(...	FALSE	OFF(O...

Port Pin Global SSO

When you see the “Loading Data” message displayed in Figure 13, it means that a process has been completed and that the shared memory is being updated with new data.

Figure 13: Loading Data

All tool views are dynamically updated when new data becomes available. This means that when you rerun an earlier process while a view is open and displaying data, the view will remain open but dimmed because its data is no longer available.

Chapter 4

User Interface Operation

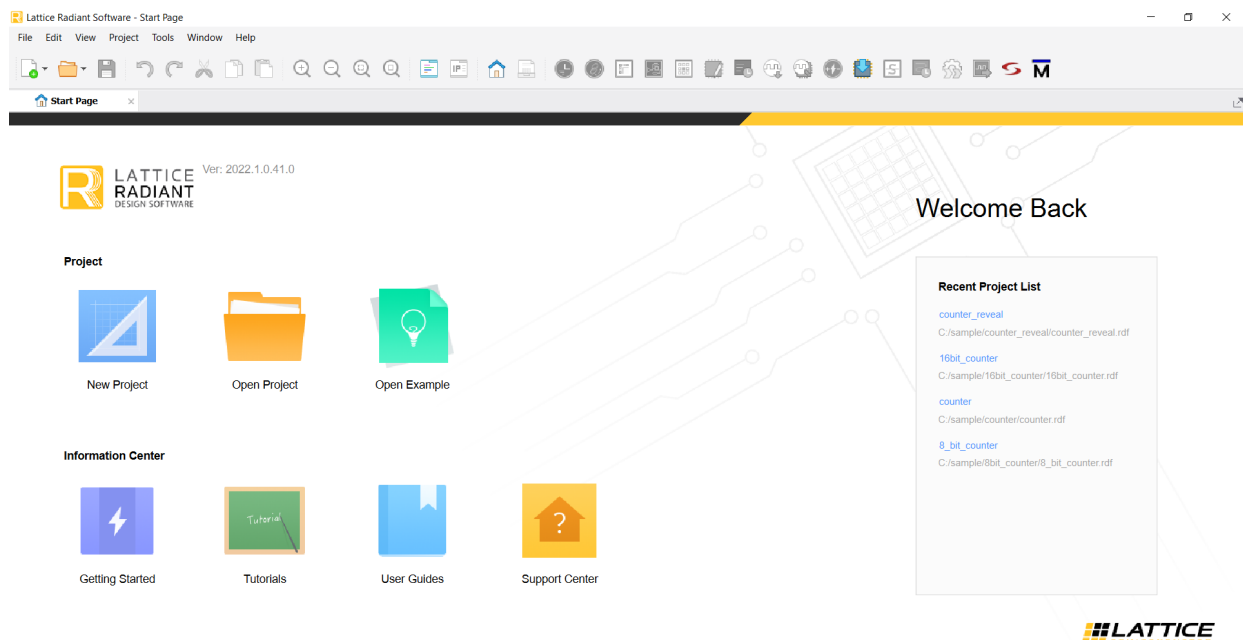
The Radiant user interface (UI) provides a comprehensive, integrated tool environment. The UI is very flexible and configurable, enabling you to store constraints for the layout you choose.

This chapter describes the user interface, controls, and basic operation of the Radiant software. Each major element of the interface is explained. The last section in the chapter describes common user interface tasks.

Start Page

The Start Page contains three major sections, as shown in Figure 14.

Figure 14: Default Start Page



- ▶ **Project:** This section allows you to create a new project; open an existing Project, and open an example.
- ▶ **Information Center:** This section has a links to Getting Started, Tutorials, User Guides, and Support Center.
- ▶ **Recent Project List:** Provides a quick way to load a recent project you've been working on.

The Start Page appears in the View area by default when the Radiant software is first launched, and can be opened from the **View** tab on the menu.

The Start Page can be closed, opened, detached, and attached using the Attach button. See ["Basic UI Controls" on page 27](#).

Menus and Toolbars

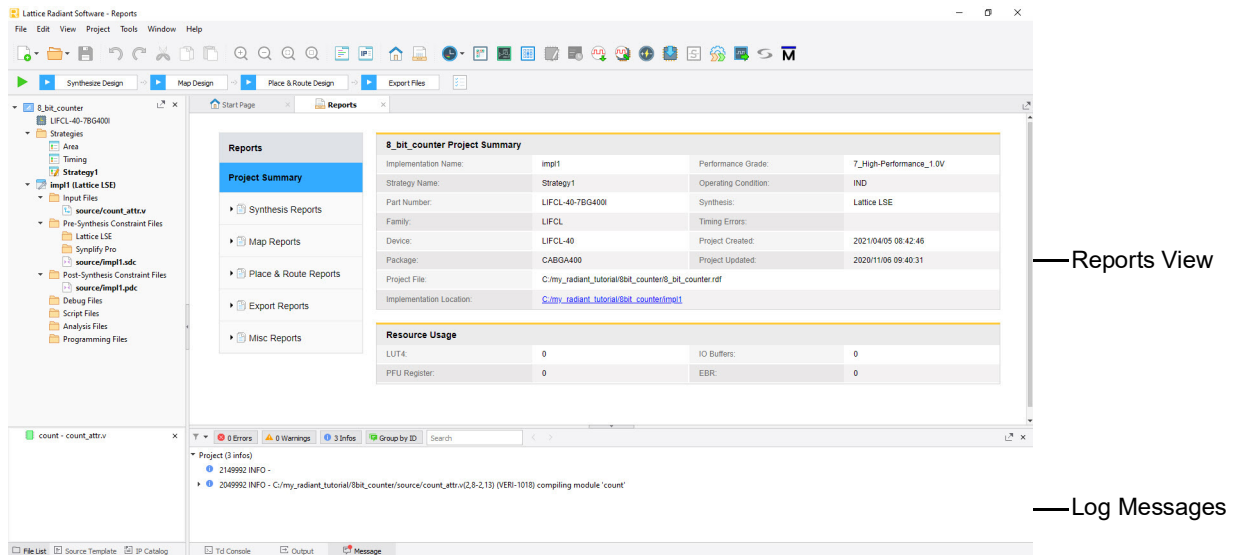
At the top of the main window is the menu and toolbar area. High-level controls for accessing tools, managing files and projects, and controlling the layout are contained here. All toolbar functionality is also contained in the menus. The menus also have functions for system, project and toolbar control.

The Process Toolbar lists all the processes available, such as Synthesize Design, Map Design, Place & Route Design, and Export Files. A process is a specific task in the overall processing of a source or project. You can view the available processes for a design in the Process Toolbar. Click **Task Detail View**  to see detailed information of the processes available.

Reports and Messages Views

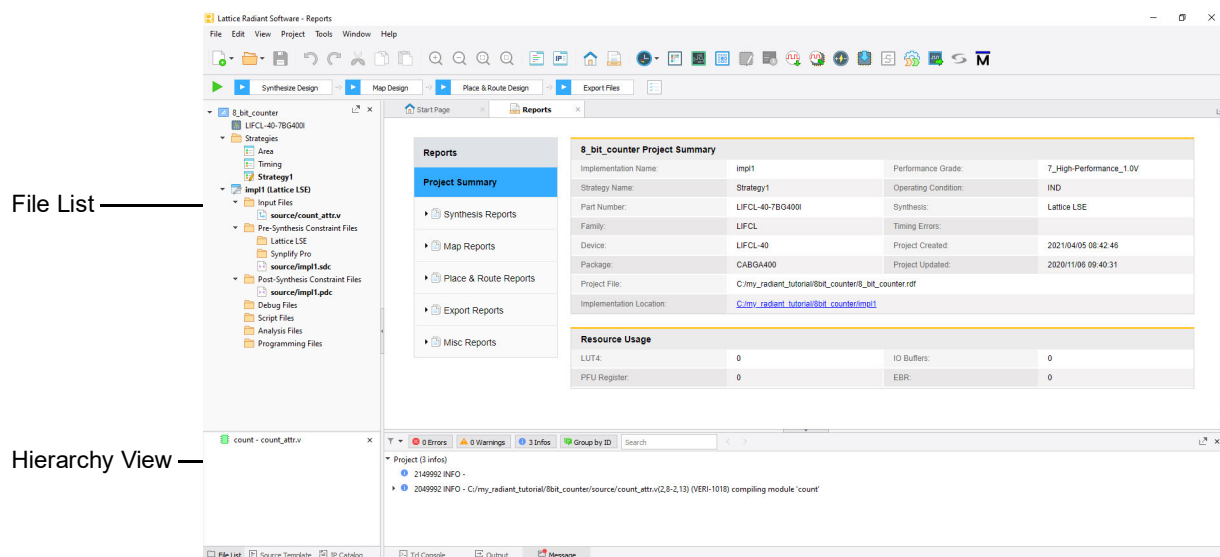
The Reports view allows you to examine and print process reports. There are two panes in the Reports view. The left pane lists the reports. The right pane displays the reports.

Log messages are displayed in the Output frame of the Radiant software main window.

Figure 15: Reports and Log Message Views

File List Views

In the middle of the main window on the left side is the File List area. This is where the overall project and process flow is displayed and controlled.

Figure 16: File List Area

Tabs at the bottom of the File List area allow you to select between the following views:

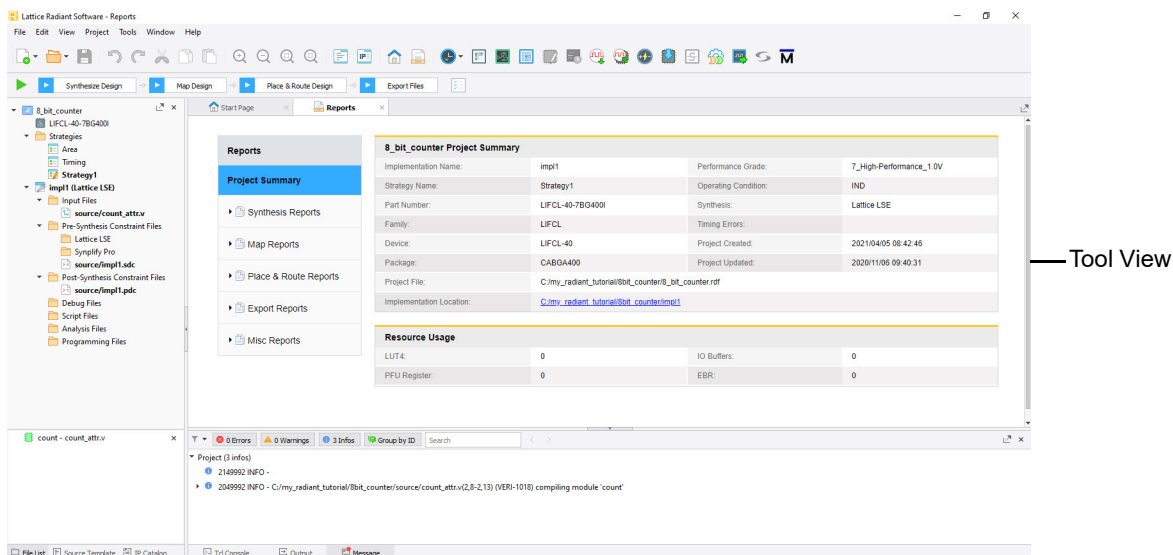
- ▶ File List – shows the files in the project organized by implementations and strategies. This is not a hierarchical listing of the design.
- ▶ Source Template – provides templates for creating VHDL, Verilog, and Constraint files.
- ▶ IP Catalog – lists available modules/intellectual properties (IP).

Underneath the File List is the Hierarchy View area. It allows you to view the hierarchical design representation. Hierarchy view shares the left pane with File List view.

Tool View Area

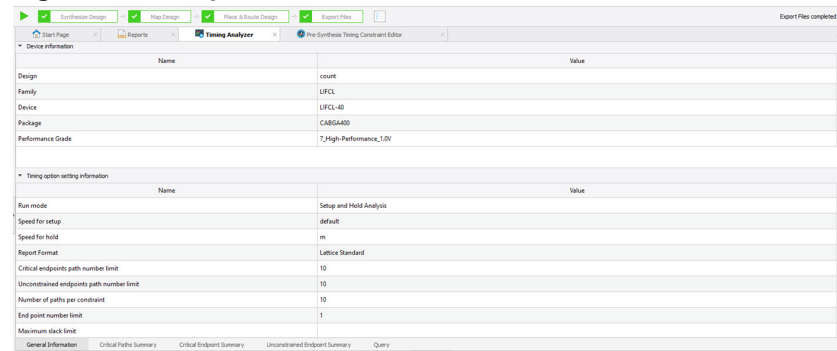
In the middle of the main window on the right side is the Tool View area. This is where the Start Page, Reports View, and all the Tool views are displayed.

Figure 17: Tool View Area



Multiple tools can be displayed at the same time. The tool tabs include controls for grouping the tool views as well as integrating all tool views back into the main window.

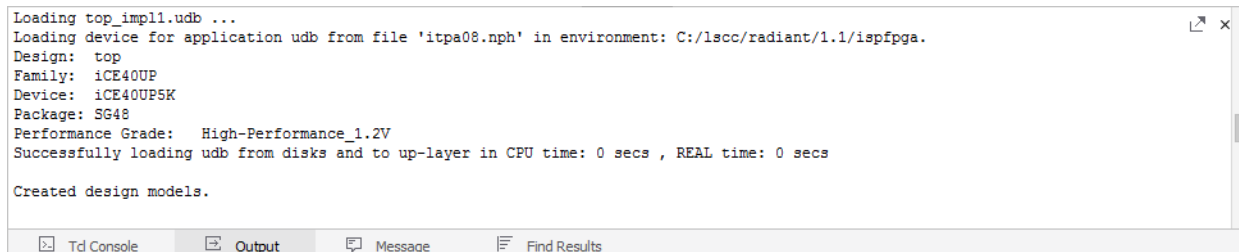
Each tool view is specific to its tool and can contain additional toolbars and multiple panes or windows controlled by additional tabs. The chapter [“Working with Tools and Views” on page 93](#) provides more details about each tool and view.

Figure 18: Multiple Tools

Output and Tcl Console

Near the right bottom of the main window is the Tcl Console, Output, and Message area.

Tabs at the bottom of this area allow you to select between Tcl Console, Output, and Message. Tool output is automatically displayed in the Output tab, and Errors and Warnings in the Message tab.

Figure 19: Output and Tcl Console Area

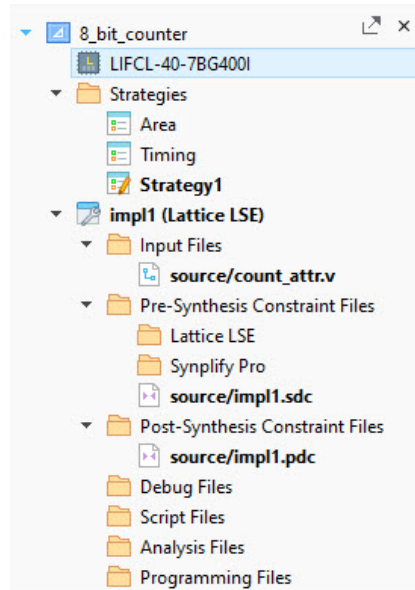
Basic UI Controls

The Radiant software environment is based on modern industry standard user interface concepts. The menus, toolbars, and mouse pointer all behave in familiar ways. You can resize any of the window panes, drag and drop elements, right-click a design element to see available actions, and hold the mouse pointer over an object to view the tool tip. Window panes can also be detached from the main window and operated as independent windows.

File List

The File List is a project view that shows the files in the project, including implementations and strategies. It is not a hierarchical listing of the design, but rather a list of all the design source, configuration and control files that make up the project.

Figure 20: File List



At the top of the File List is the project name. Directly below the project name is the target device, followed by the strategies, and then the implementations. There must be one active implementation, and it must have one active strategy. Active elements are indicated in **bold**.

You can right-click any file or item in the File List to access a pop-up menu of currently available actions for that item. The pop-up menu contents vary, depending on the type of item selected.

The File List view can be hidden by clicking the small arrow in right border: "Click to show/hide side panel."

Source Template

The Source Template is a project view that provides templates for creating VHDL, Verilog, and constraint files. Templates increase the speed and accuracy of design entry. You can drag and drop a template directly to the source file. You can also create your own templates.

To access templates, choose **View > Show Views > Source Template**, or click  icon in the toolbar, or click on the **Source Template** tab in the bottom-left pane, to locate and access the following templates:

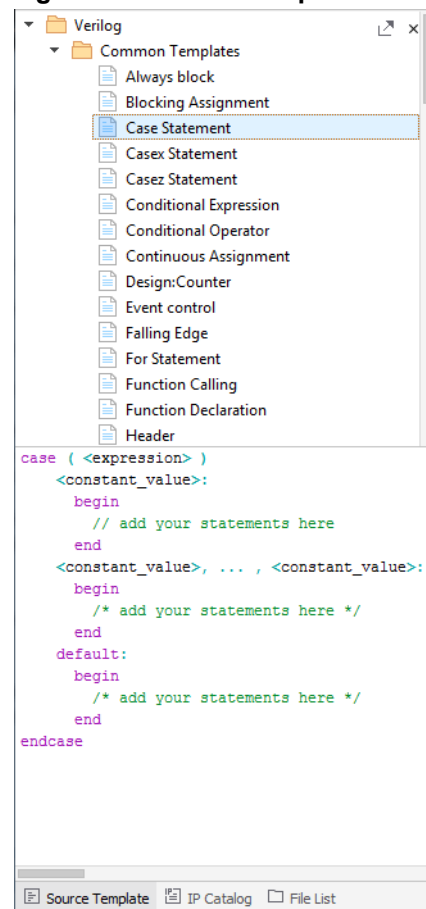
- ▶ Verilog, including common and Parameterized Module Instantiation (PMI), Primitives, Attributes, Encryption, and User Templates
- ▶ VHDL, including common, PMI, Primitives, Attributes, Encryption, and User Templates
- ▶ Constraints for LSE, including Timing and Physical constraints and User Templates

Note

For more information on PMI, refer to the Radiant Software Help. See **User Guides > Entering the Design > Designing with Soft IP, Modules, and PMI > PMI or IP Catalog?**

You can simply drag any template and drop it into your source file.

Figure 21: Source Template

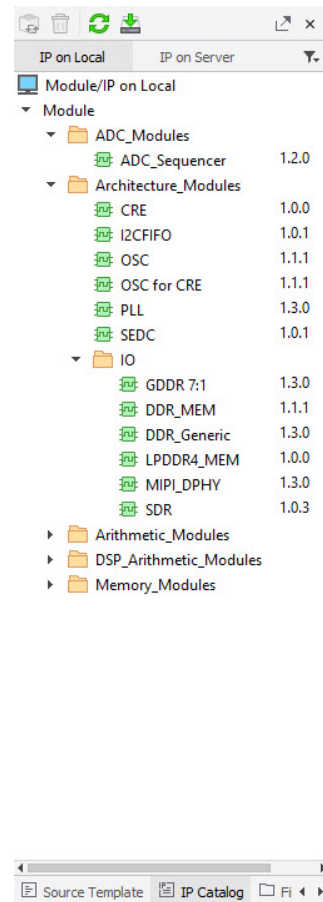


IP Catalog

IP Catalog enables you to customize a collection of functional blocks from Lattice Semiconductor. Through the IP Catalog, you can access two types of functional blocks, Modules and IP.

To access IP catalog, choose **View > Show Views > IP Catalog**, or click  icon in the toolbar, or click on the **IP Catalog** tab in the bottom-left pane.

Figure 22: IP Catalog



Each module is configurable with a unique set of properties. Once generated, the module or IP appears in your design's File List.

Process

A process is a specific task in the overall processing of a source or project. Typical processing tasks include synthesizing, mapping, placing, and routing. You can view the available processes for a design in the Process Toolbar.

Figure 23: Process Toolbar

The process status icons are defined as follows:

-  Process in initial state (not processed)
-  Process completed successfully
-  Process completed with unprocessed subtasks
-  Process failed

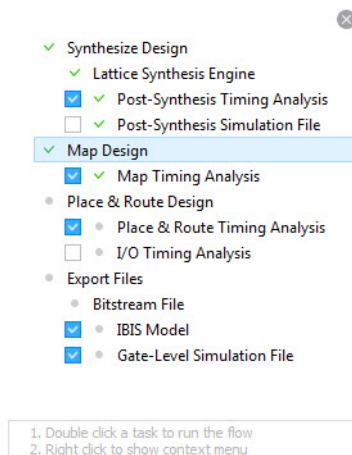
For more detail of different designs and Export Files available, see [“Process Flow” on page 19](#).

Task Detail View

Click Task Detail View  to see detailed information of each process.

The default design flow processes are marked by check marks. To enable the remaining tasks, either check-mark the specific task and rerun the process step, or double-click the task’s name. You can also right-click on the task to show the context menu.

Once the process has finished, the process status icon next to the task replaces the gray dot.

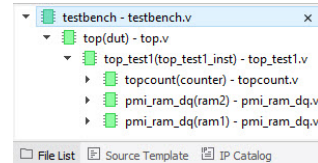
Figure 24: Task Detail View

Processes are grouped into categories according to their functions. To learn more about each process, view [“Design Flow Processes” on page 78](#).

Hierarchy

The Hierarchy view is a project view that displays the design hierarchy and is displayed by default. The hierarchical view is available when File List tab is selected.

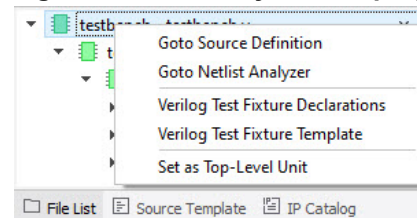
Figure 25: Hierarchy View



If you would prefer that it not open by default, simply close Hierarchy View. The next time you launch the Radiant software, the Hierarchy View will not be opened. You can open it manually by selecting it from the View > Show View menu.

Right-click any of the objects in the Hierarchy View to see the available actions.

Figure 26: Hierarchy Item Pop-up Menu



The Hierarchy view can be selected, closed, and opened.

Reports

The Reports View provides a centralized reporting mechanism in the Tools view area. The Reports View is automatically displayed and updated when processes are run. It provides a separate tab for the current implementation, enabling you to compare results quickly.

The right pane displays the report for the selected step. You can also click the  icon in the toolbar.

Figure 27: Reports View

Project Summary	
▸ Synthesis Reports	Implementation Name:
▸ Map Reports	Strategy Name:
▸ Place & Route Reports	Part Number:
▸ Export Reports	Family:
▸ Misc Reports	Device:
	Package:
	Project File:
	Implementation Location:
	Resource Usage
	LUT4:
	PFU Register:

The Reports pane on the right shows the detail of the project summary and resource usage.

The Report View can be selected, closed, opened, detached, and attached with the Attach button. See [“Basic UI Controls” on page 27](#).

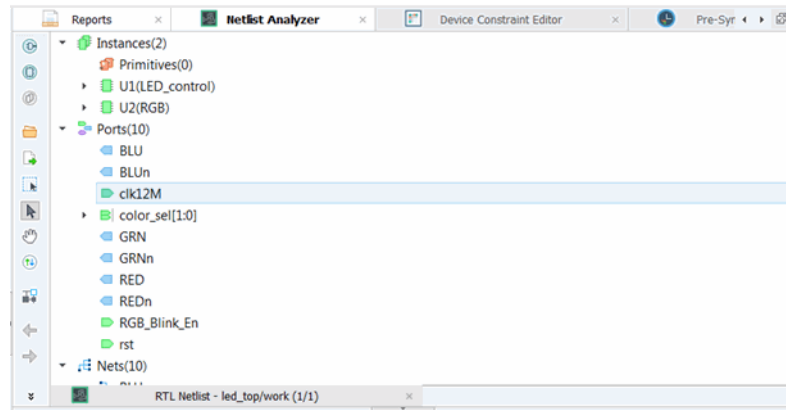
Tool Views

The Tool view area of the UI displays the tools that are currently active. Each tool that you have opened from the toolbar or the Tools menu is displayed. The Reports and Start page, which can be opened from the toolbar or the Windows menu, are also displayed. When multiple tools are active, the display can be controlled with the tab group functions in the Window menu. See [“Common Tasks” on page 35](#) for more information on tab group functions.

Each tool view is specific to its tool and can contain additional toolbars, multiple panes, or multiple windows controlled by additional tabs. See [“Working with Tools and Views” on page 93](#) for descriptions of each tool and view, plus details on controlling their display.

The Tool views can be selected, closed, opened, detached, and attached using the Attach button. See [“Basic UI Controls” on page 27](#)

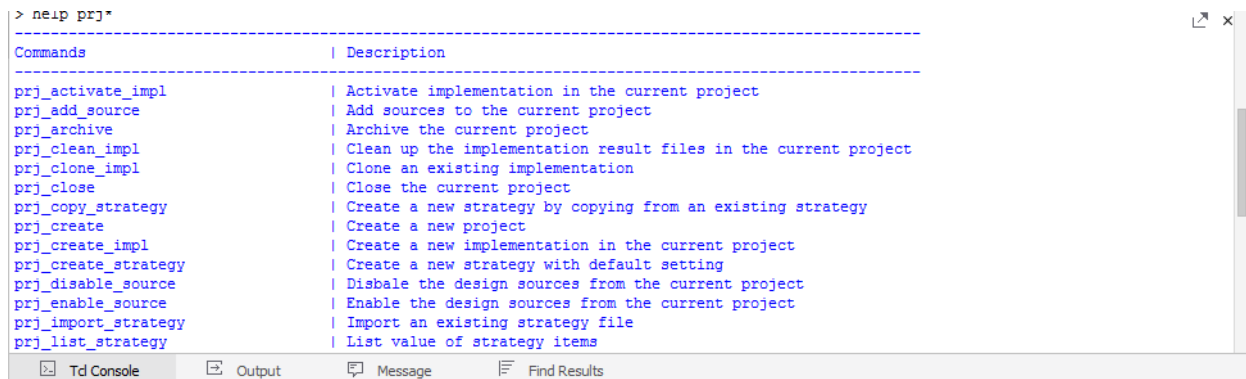
Figure 28: Tool View Tab Title



Tcl Console

The Tcl Console is an integrated console for Tcl scripting. You can enter Tcl commands in the console to control all of the functionality of the Radiant software. Use the Tcl help command (`help <tool_name>*`) to display a list of valid extended Tcl commands.

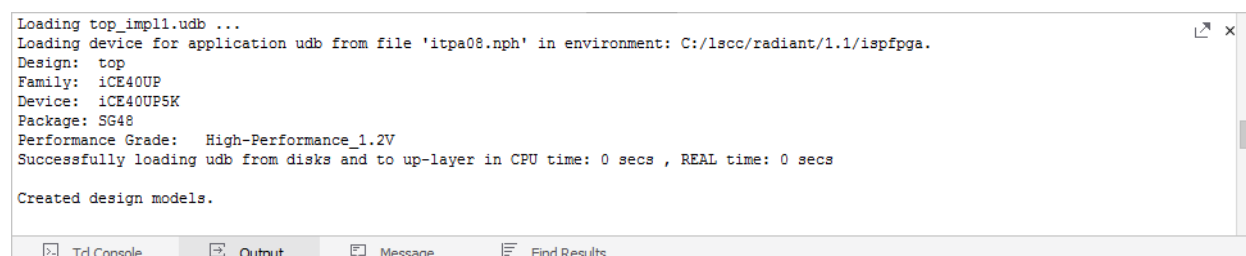
Figure 29: Tcl Console



Output

The Output View is a read-only area where tool output is displayed.

Figure 30: Output View



Message

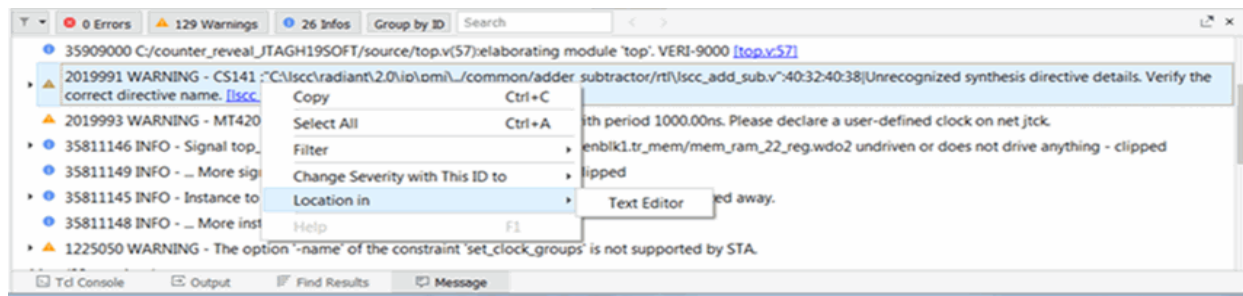
There are three message types available:

- ▶ Errors are displayed in red.
- ▶ Warnings are displayed in orange.
- ▶ General Information is displayed in blue.

 **Message** A red dot in the Message tab provides a visual notification that a new message/warning was received. Once you view the notification, the dot disappears.

Right-clicking a message provides a menu of commands, including **Location in > Text Editor**, which opens the source file in the Source Editor and highlights the location of the problem.

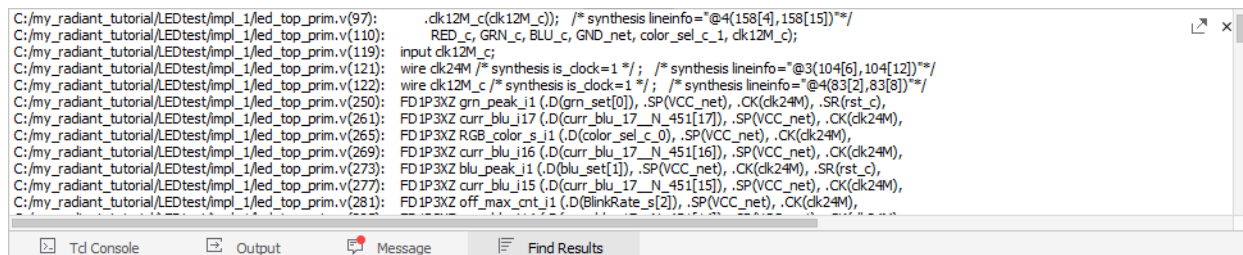
Figure 31: Message Display



Find Results

The **Edit > Find in Files** command enables you to search for information in the files within your project directory. The search results are then displayed in the Find Results view.

Figure 32: Find Results View



Common Tasks

The Radiant software UI controls many tools and processes. The following sections describe some of the more commonly performed tasks.

Controlling Views

All of the views in the Radiant software are controlled in a similar manner, even though the information they contain varies widely. Here are some of the most common operations:

- ▶ **Open** – Use the **View > Show Views** menu selections or right-click in the menu or toolbar areas to select a view from the pop-up menu.
- ▶ **Select** – If a view is already open you can select its tab to bring it to the front.
- ▶ **Detach** – Click the detach button  in the upper right corner of the view.
- ▶ **Attach** – Click the attach button  in the upper right corner of the view.
- ▶ **Move** – Click and hold a view's tab, and then drag and drop the view to a different position among the open views.

Using a Tab Group You can use the Window menu to split off a view and control it as a separate tab group. This allows you to examine two open views side by side. The controls work as follows:

- ▶ **Split Tab Group** – displays two views side by side. See Figure 33.
- ▶ **Move to Another Tab Group** – moves the selected tab to the other tab group. See Figure 34.
- ▶ **Merge Tab Group** – merges a split tab group back into the primary view
- ▶ **Switch Tab Group Position** – switches the positions of the two tab groups.

Figure 33: After Split Tab Group Command Used on Physical Designer

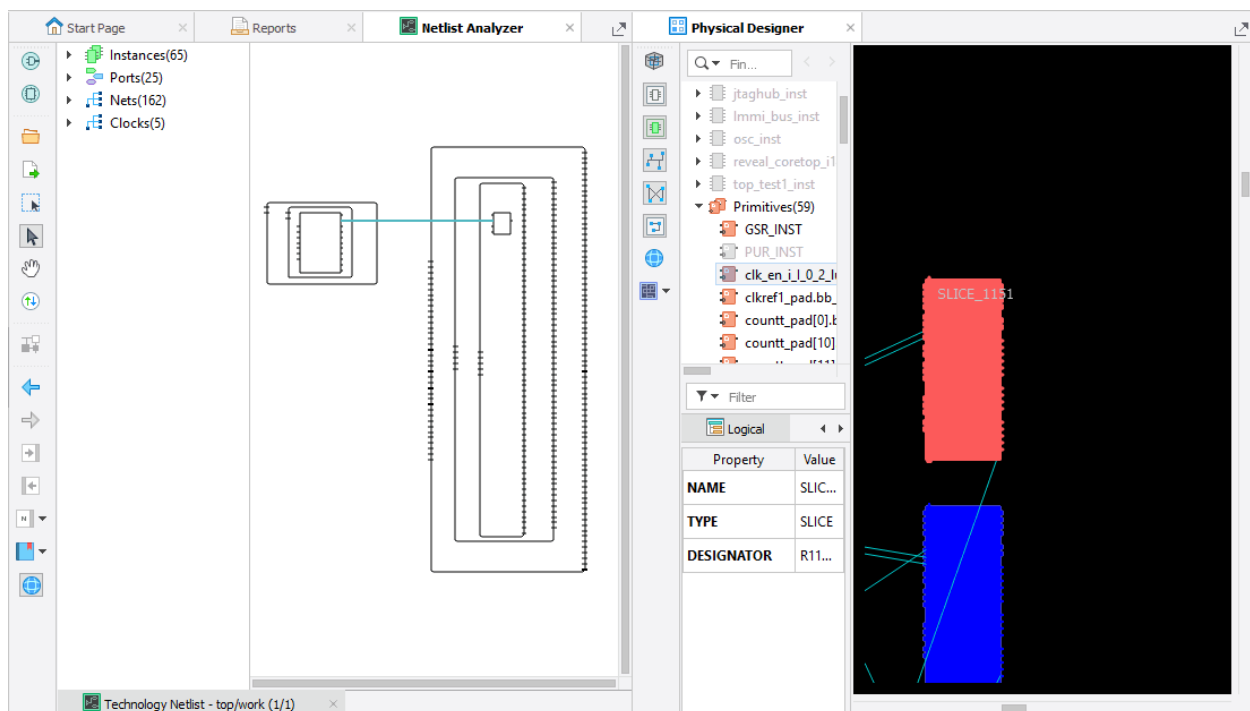
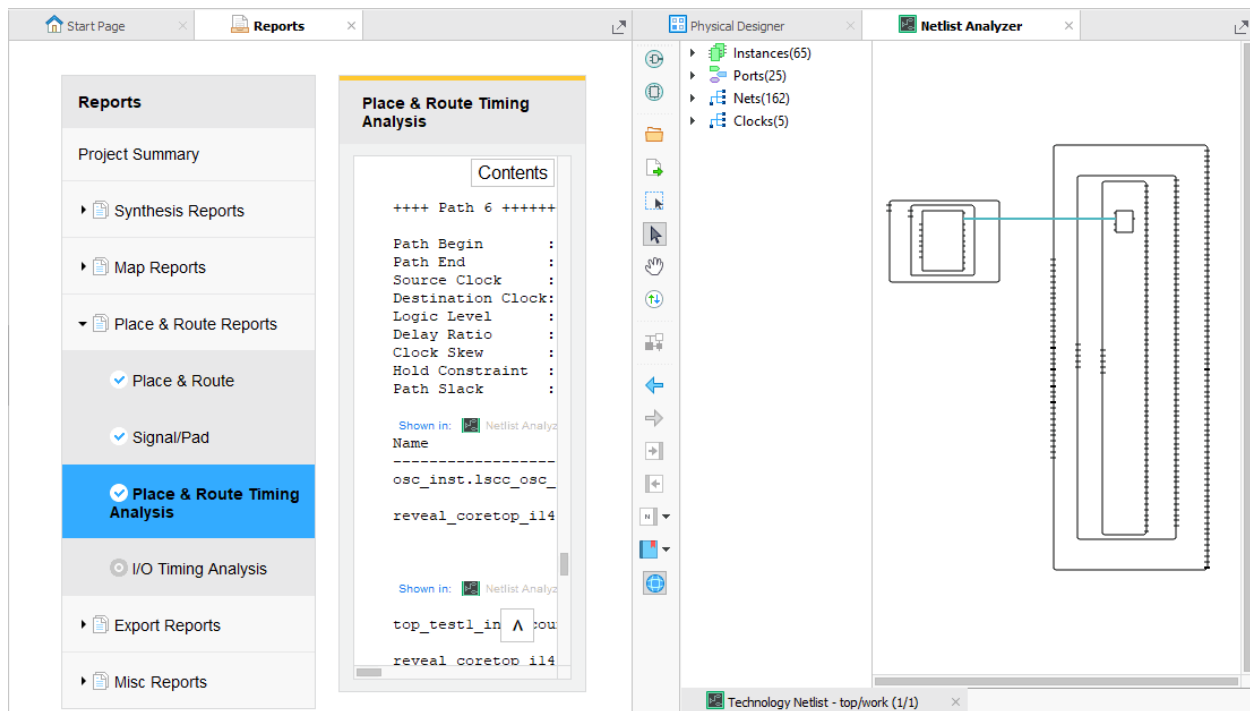
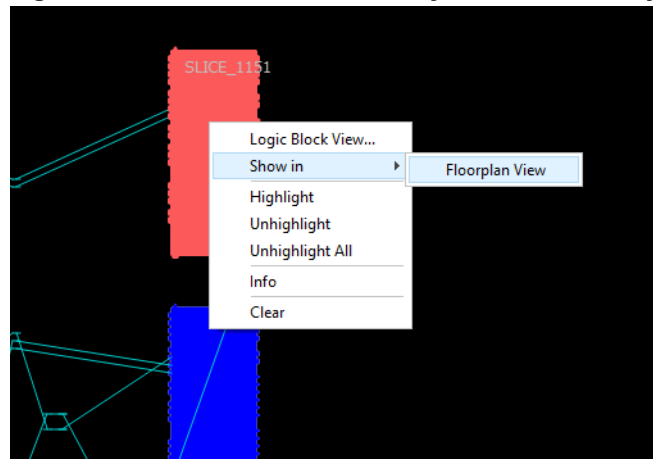


Figure 34: After Move to Another Tab Group Used on Netlist Analyzer

Cross-Probing

It is possible to select a data object in one view and see that same data object in a different view or views. Right-click an object to see if cross-probing is available. If it is, you will see a **Show In** sub-menu with the available views listed. If you select a view that is not yet open, the Radiant software will open it automatically. Cross-probing is available between Floorplan View and Physical View of Physical Designer, and from Netlist Analyzer to Physical Designer.

Figure 35: Show In Menu from Physical View of Physical Designer

During the Radiant flow, various timing analyses and reports are created. You can view a specific path in Netlist Analyzer, Physical Designer's Floorplan View, and Physical Designer's Physical View. This allows for flexibility and reduced debugging effort.

NOTE

Cross-probing to Netlist Analyzer is available only if the selected synthesis tool is LSE.

In the Reports tab, view any timing analysis report and identify a path to view. If cross-probing is available, the specific icon tools become visible, as shown in the following figure.

Figure 36: Available tools for Path Cross-Probing

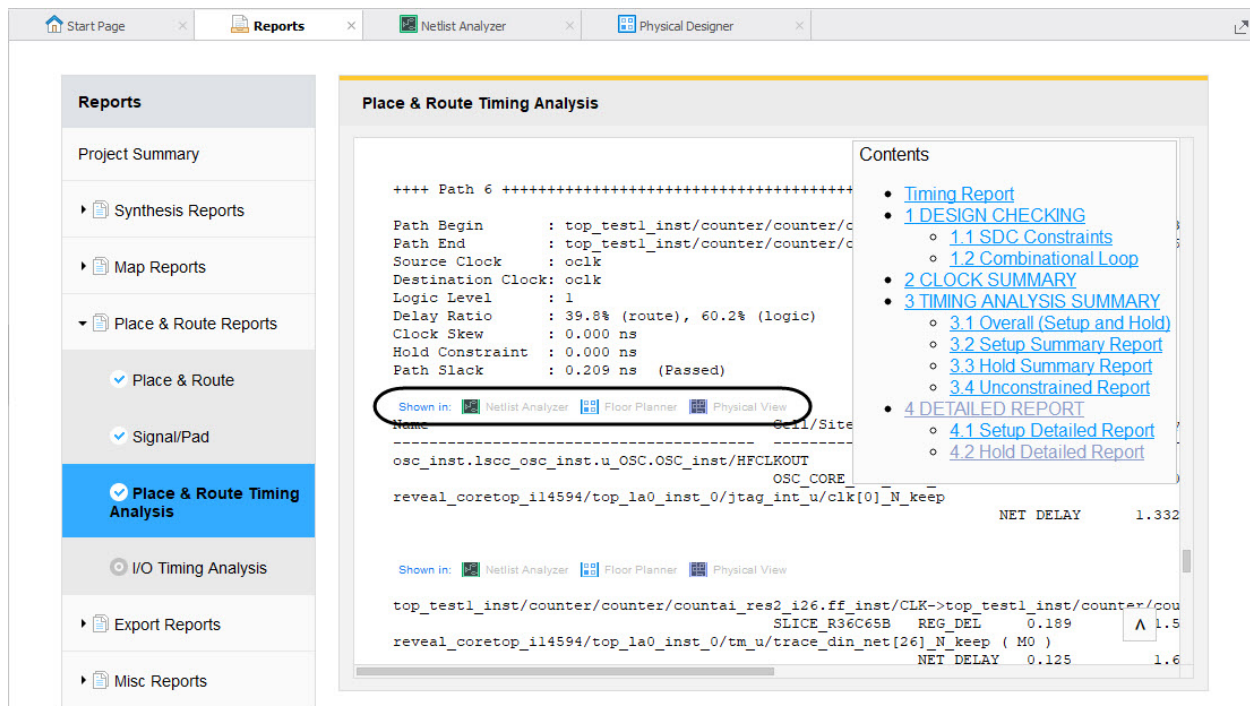
Shown in:  Netlist Analyzer  Floor Planner  Physical View

Click on an icon and the tool opens with the selected path.

In some cases, the tool is unable to find the path. The message “Can’t show the schematic of this timing path.” appears. In an encrypted design, in some cases, cross-probing is not available. The message “Cannot open encrypted design.” appears.

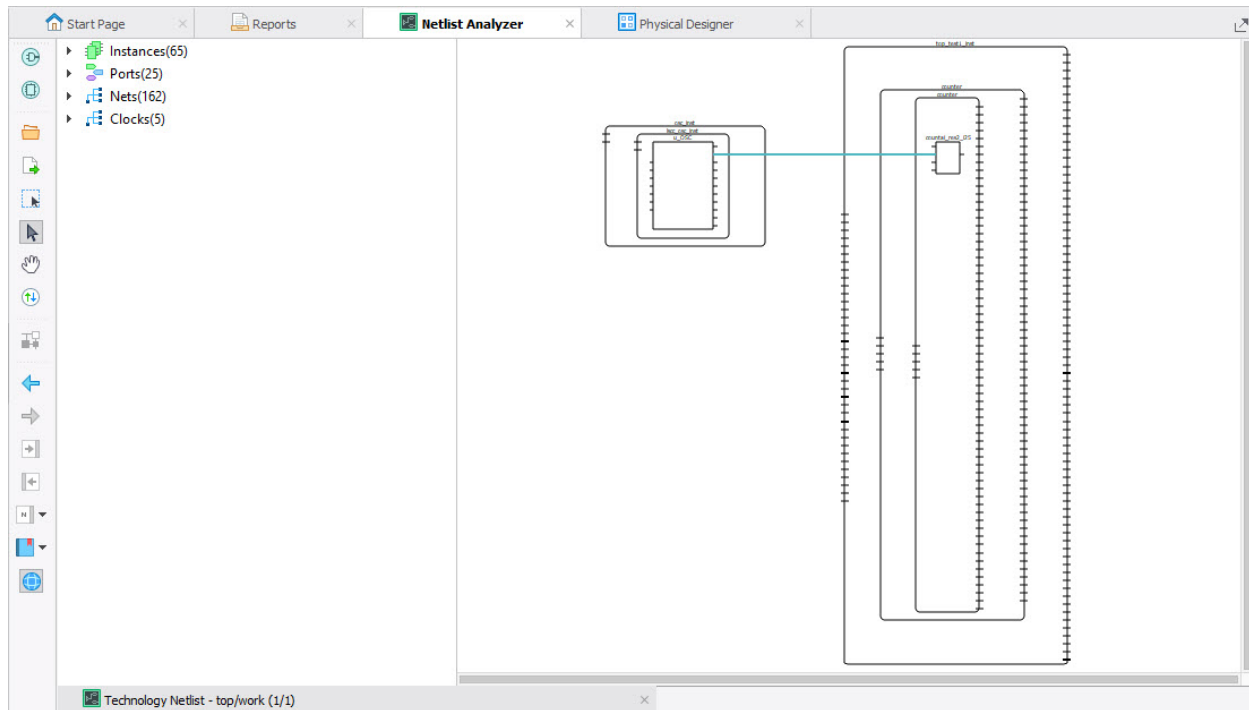
The following figures show cross-probing a path from the Place & Route Timing Analysis report to Netlist Analyzer, Physical Designer's Floorplan View, and Physical Designer's Physical View.

Figure 37: Path Cross-Probing in Reports

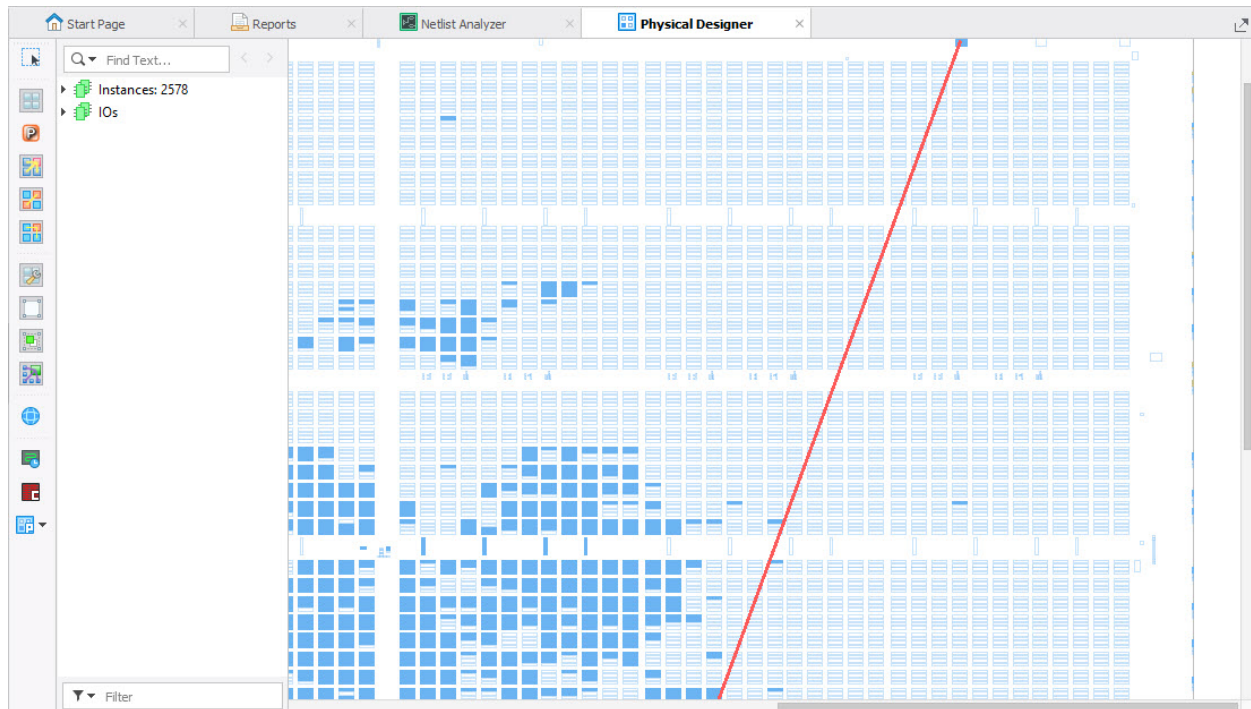


By clicking on the Netlist Analyzer icon, you can preview the data path in Netlist Analyzer.

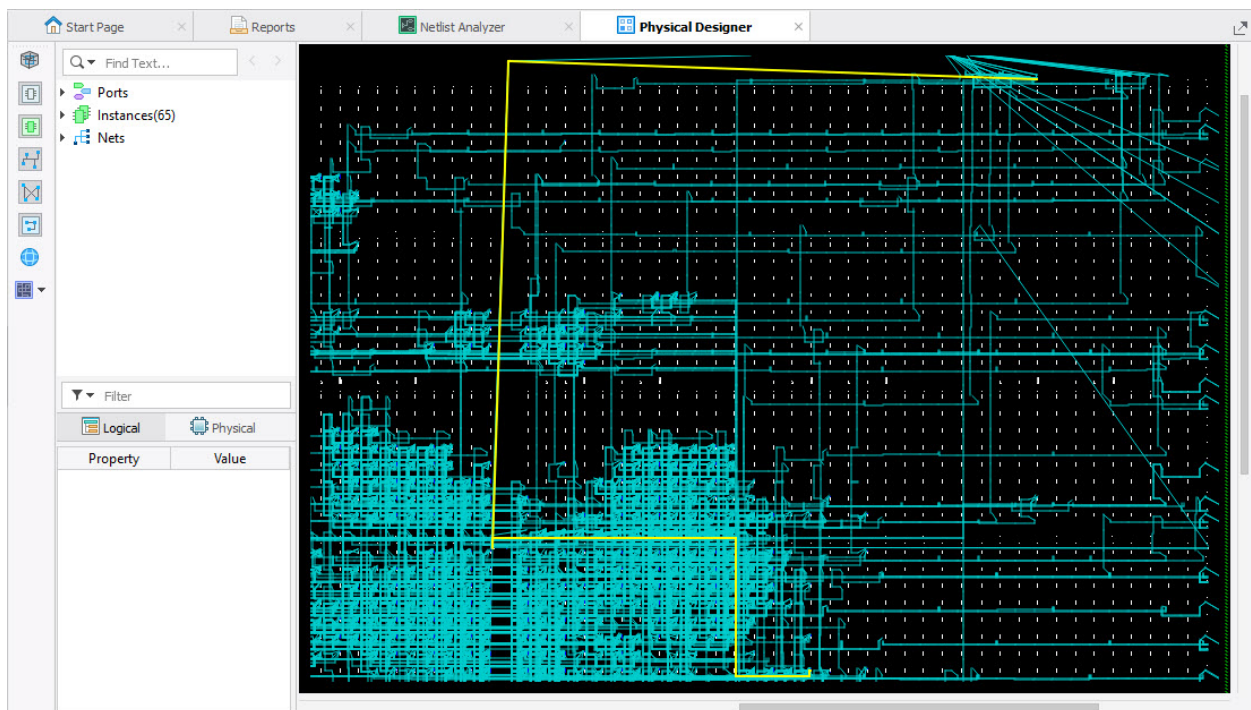
Figure 38: Path Cross-Probing in Netlist Analyzer



Similarly, by clicking on the Floor Planner icon, you can easily view the same path in Physical Designer's Placement Mode.

Figure 39: Path Cross-Probing in Physical Designer's Placement Mode

The same path is viewable in Physical Designer's Routing Mode by clicking on the Physical View icon in a timing report.

Figure 40: Path Cross-Probing in Physical Designer's Routing Mode

Chapter 5

Working with Projects

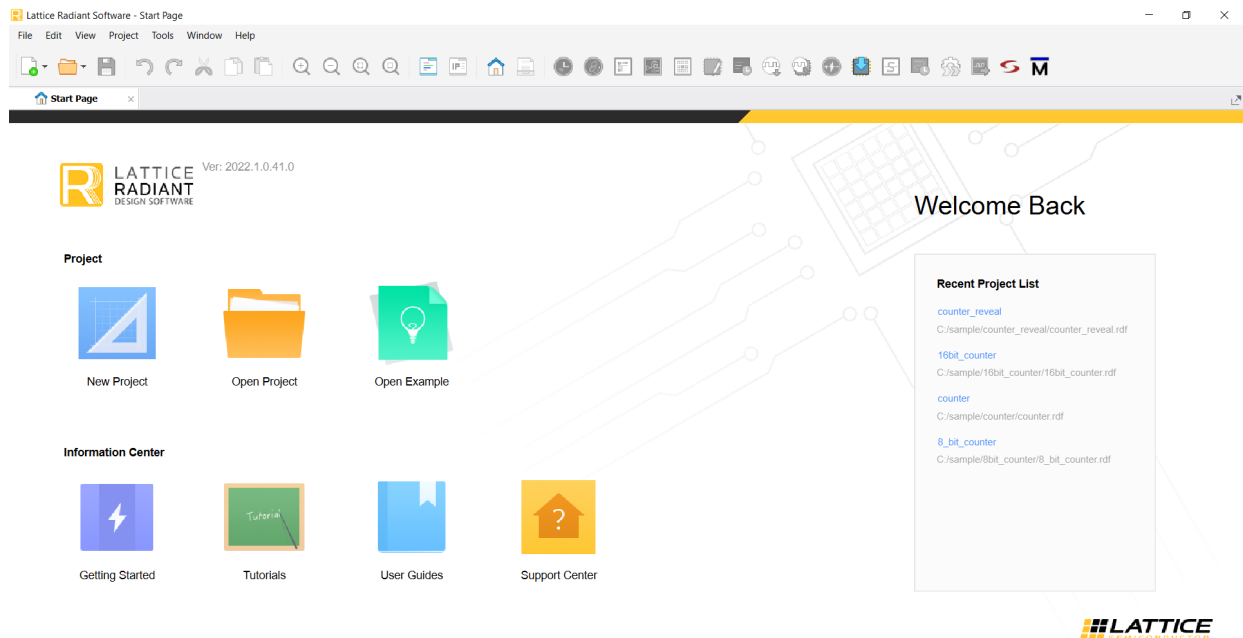
This chapter covers projects and their elements. Implementations and strategies are explained and some common project tasks are shown.

Overview

A project is the top organizational element in the Radiant software design environment. Projects consist of design, constraint, configuration and analysis files. Only one project can be open at a time, and a single project can include multiple design structures and tool settings.

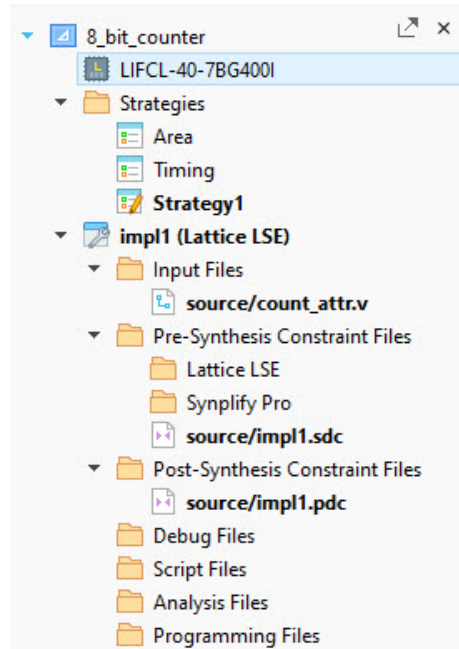
You can create, open, or import a project from the Start Page. Refer to [“Getting Started” on page 11](#) for instructions on creating a new project.

Figure 41: Default Start Page



The File List view shows a project and its main elements.

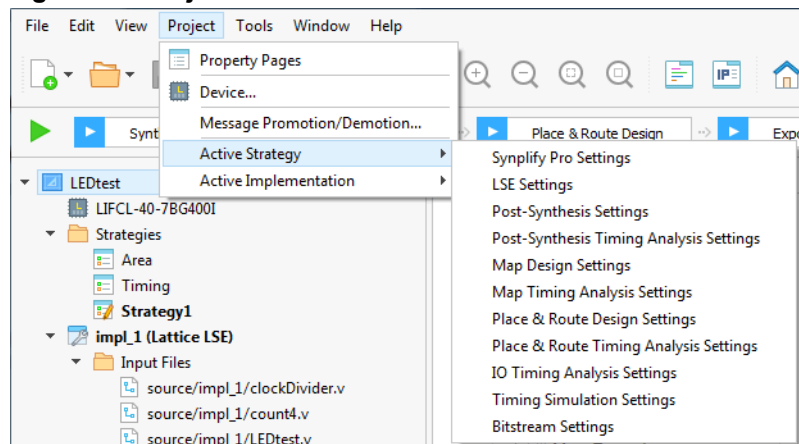
Figure 42: Project Files in File List



The Project menu commands enable you to do the following:

- ▶ Examine the project properties.
- ▶ Change the target device.
- ▶ Change the severity level of warning messages.
- ▶ Set the synthesis tool.
- ▶ Show the active strategy tool settings.
- ▶ Set the top level design unit.

Figure 43: Project Menu



Implementations

An implementation is the structure of a design and can be thought of as *what* is in the design. For example, one implementation might use inferred memory while another implementation uses instantiated memory. Implementations also define the constraint and analysis parameters for a project.

There can be multiple implementations in a project, but only one implementation can be active at a time. And there must be one active implementation. Every implementation has an associated active strategy. Strategies are a shared pool of resources for all implementations and are discussed in the next section. An implementation is created whenever you create a new project.

Implementations consist of the following files:

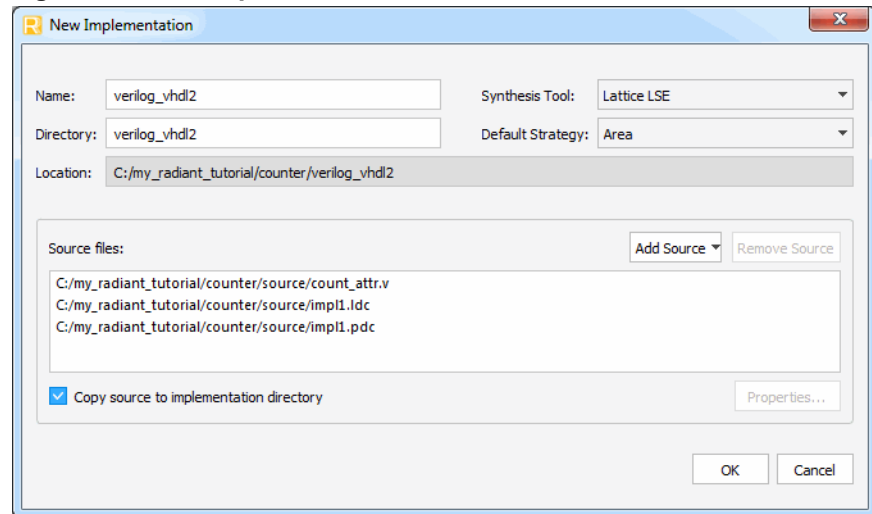
- ▶ Input files
- ▶ Pre-Synthesis constraint files
- ▶ Post-Synthesis constraint files
- ▶ Debug files
- ▶ Script files
- ▶ Analysis files
- ▶ Programming files

Adding Implementations

To add a new implementation to an existing project:

1. Right-click the project name in the File List project view.

Select **Add > New Implementation**. In the New Implementation dialog box, you can set the implementation name, directory, default strategy, and add source files. When you select **Add Source** you have a choice of browsing for the source files or using a source from an existing implementation.

Figure 44: New Implementation

Notice that you have the option to “Copy source to implementation directory.” If this option is selected, the source files will be copied from the existing implementation to the new implementation, and you will be working with different source files in the two implementations. If you want the two implementations to share the same source files and stay in sync, make sure that this option is not selected.

To make an implementation active, right-click its name in the File List and choose **Set as Active Implementation**.

To add a file to an implementation, right-click the implementation name or any file folder in the implementation and choose **Add > New File** or **Add > Existing File**.

Cloning Implementations

To clone an implementation:

1. In the File List view, right-click on the name of the implementation that you want to copy and choose **Clone Implementation**.

The Clone Implementation dialog box opens.

2. In the dialog box, enter a name for the new implementation. This name also becomes the default name for the folder of the implementation.
3. Change the name of the implementation’s folder in the Directory text box, if desired.
4. Decide how you want to handle files that are outside of the original implementation directory. Select one of the following options:

▶ **Continue to use the existing references**

The same files will be used by both implementations.

► **Copy files to new implementation source directory**

The new implementation will have its own copies that can be changed without effecting the original implementation.

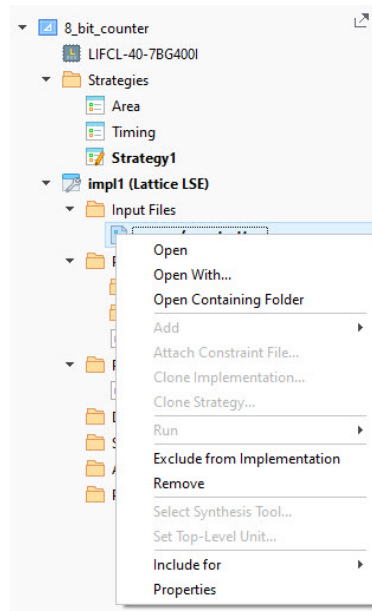
5. The Synthesis Tool text box specifies the currently selected synthesis tool. Go to **Project > Active Implementation > Select Synthesis Tool** to update your selection.
6. The Default Strategy text box specifies the currently selected default strategy.
7. Click **OK**.

Input Files

Input files are the design source files for the project. Input files can be any combination of Verilog, SystemVerilog, and VHDL.

Right-click an input file name to open a pop-up menu of possible actions for that file.

Figure 45: Input File Actions



You can use the “Include for” commands to specify that a source file be included for both synthesis and simulation, synthesis only, or simulation only.

Pre-Synthesis Constraint Files

Synopsys timing constraints are specified in the new .fdc file format. Legacy .sdc formats are still supported in the Radiant software and Synopsys has provided a script called sdc2fdc, which does a one-time conversion of .sdc files to the new .fdc format. More information about this script can be found in the Synplify Pro release notes.

An .fdc file can be added to an implementation if the selected synthesis tool is Synplify Pro. When using Synplify Pro or the Lattice Synthesis Engine (LSE), the constraints files can be saved as an .sdc file. If the selected synthesis tool is LSE, a Lattice design constraint (.ldc) synthesis file can be added. Constraints in the .ldc file use the Synopsys constraint format.

An implementation can have multiple synthesis constraint files. Only one synthesis constraint file can be active at a time. Unlike Post-Synthesis constraints, a synthesis constraint file must be set as active by the user.

Post-Synthesis Constraint Files

Post-Synthesis constraint files (.pdc) contain both timing and non-timing constraint .pdc source files for storing logical timing/physical constraints. Constraints that are added using the Device Constraint Editor are saved to the active .pdc file. The active post-synthesis design constraint file is then used as input for post-synthesis processes.

An implementation can have multiple .pdc files, but only one can be active at a time.

Figure 46: Sample .pdc File



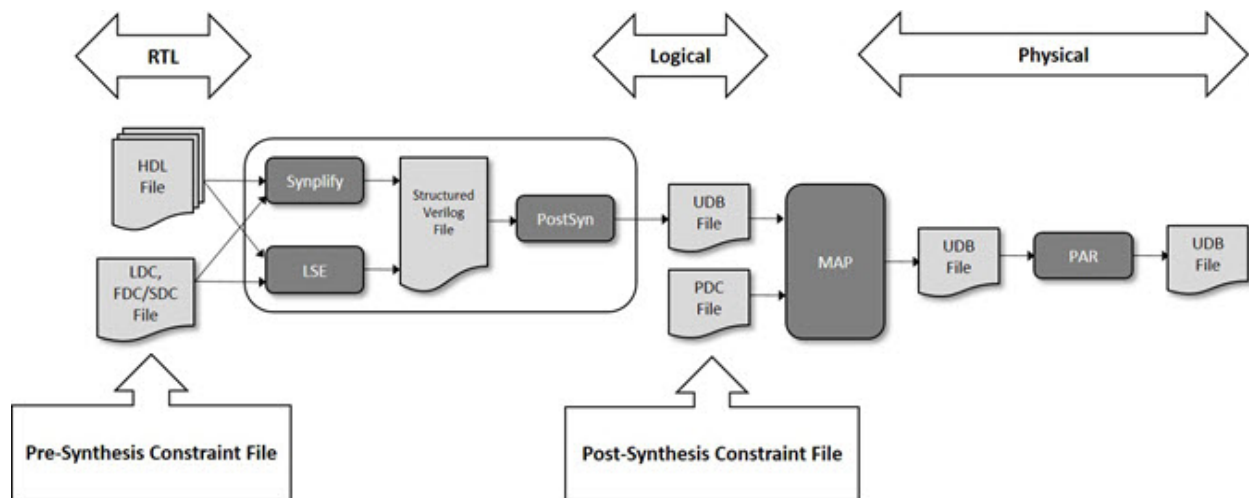
```

1 ldc_set_location -site P5 [get_ports clk]
2 #ldc_create_region -name RgnA -site R8C2D -width 18 -height 12
3 #ldc_create_group -name UgrpA [get_cells counter1]
4 #ldc_set_location -region RgnA [ldc_get_groups UgrpB]
5 #ldc_create_group -name UgrpC -bbox {2 3} [get_cells counter3]
6 #ldc_set_location -site R8C25D [ldc_get_groups UgrpC]
7 ldc_set_port -iobuf {IO_TYPE=LVCMS25} [get_ports {count2t[2]}]
8 ldc_set_port -iobuf {IO_TYPE=LVCMS25} [get_ports {count2t[3]}]
9 ldc_set_port -iobuf {IO_TYPE=LVCMS25} [get_ports {count2t[0]}]
10 ldc_set_port -iobuf {IO_TYPE=LVCMS25} [get_ports {count2t[1]}]
11 ldc_set_port -iobuf {IO_TYPE=LVCMS25} [get_ports {count3t[0]}]
12 ldc_set_port -iobuf {IO_TYPE=LVCMS25} [get_ports {count3t[1]}]
13 ldc_set_port -iobuf {IO_TYPE=LVCMS25} [get_ports {count3t[2]}]
14 ldc_set_port -iobuf {IO_TYPE=LVCMS25} [get_ports {count3t[3]}]
15 ldc_set_port -iobuf {IO_TYPE=LVCMS25} [get_ports {count3t[4]}]
16 ldc_set_port -iobuf {IO_TYPE=LVCMS25} [get_ports {count3t[5]}]
17 ldc_set_port -iobuf {IO_TYPE=LVCMS25} [get_ports {count3t[6]}]
18 ldc_set_port -iobuf {IO_TYPE=LVCMS25} [get_ports {count3t[7]}]
19 #ldc_set_location -site 35 [get_ports clk1]
20 create_clock -period 15 [get_nets counter1/clk1_c]
21

```

Figure 47 shows a high-level flow of how constraints from multiple sources can be used and modified in the Radiant software.

Figure 47: Radiant software Constraints Flow Chart

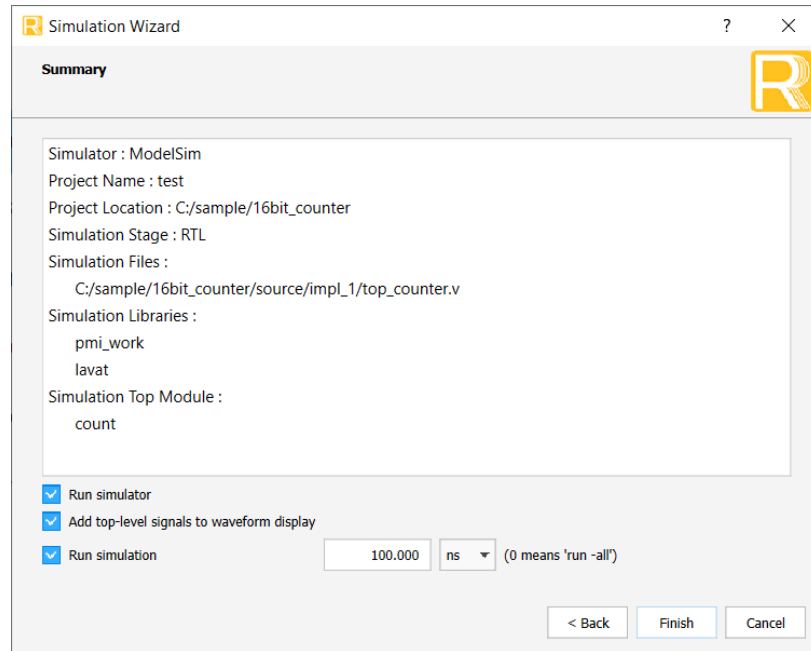


Debug Files

The files in the Debug folder are project files for Reveal Inserter. They are used to insert hardware debug into your design. There can be multiple debug files, and one can be set as active. To insert hardware debug into your design, right-click a debug file name and choose **Set as Active Debug File** from the pop-up menu. The debug file name becomes bold, indicating that it is active. It is not required to have an active debug file.

Script Files

The Script Files folder contains the scripts that are generated by the Simulation Wizard. After you run the Simulation Wizard, the steps are stored in a simulation project file (.spf), which can be used to control the launching of the simulator.

Figure 48: Simulation Script File

Analysis Files

The Analysis Files folder contains Power Calculator files (.pcf). The folder can contain multiple analysis files, and one (or none) can be set as active. The active or non-active status of an analysis file affects the behavior of the associated tool view.

Programming Files

Programming files (.xcf) are configuration scan chain files used by the Radiant Programmer for programming devices. The .xcf file contains information about each device, the data files targeted, and the operations to be performed.

An implementation can have multiple .xcf files, but only one can be active at a time. The file must be set as active by the user.

Strategies

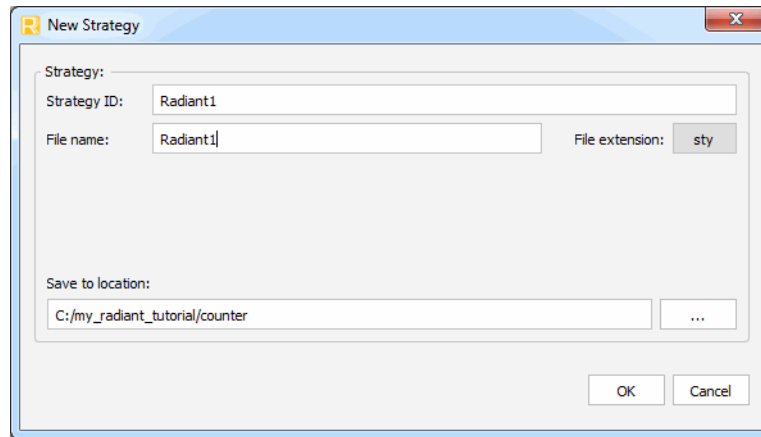
Strategies are collections of all the implementation-related tool settings in one convenient location. Strategies can be thought of as recipes for how the design will be implemented. An implementation defines *what* is in the design,

and a strategy defines *how* that design will be run. There can be many strategies, but only one can be active at a time. There must be one active strategy for each implementation.

The Radiant software provides two predefined strategies: Area and Timing. It also enables you to create customized strategies. Predefined strategies cannot be edited, but they can be cloned, modified, and saved as customized user strategies. Customized user strategies can be edited, cloned, and removed. All strategies are available to all of the implementations, and any strategy can be set as the active one for an implementation.

To create a new strategy from scratch, choose **File > New > Strategy**. In the New Strategy dialog box, enter a name for the new strategy. Specify a file name for the new strategy and choose a directory to save the strategy file (.sty).

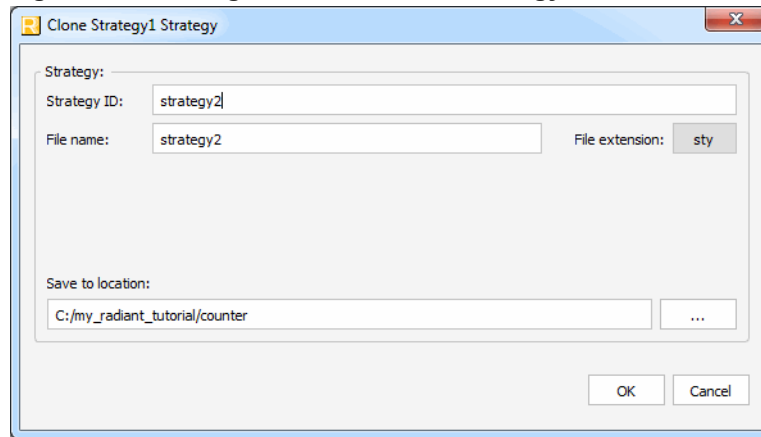
Figure 49: Creating a New Strategy from Scratch



The new strategy is with all the default settings of the current design. You can modify its settings in the Strategies dialog box.

If you want to save the strategy changes to your current project, choose **File > Save Project** from the Radiant software main window.

To create a new strategy from an existing one, right-click the existing strategy and choose **Clone <strategy name> Strategy**. Set the new strategy's ID and file name.

Figure 50: Cloning to Create a New Strategy

To make a strategy active, right-click the strategy name and choose **Set as Active Strategy**.

To change the settings in a strategy:

1. Double-click the strategy name in the File List view
2. Select the option type to modify
3. Double-click the Value of the option to be changed

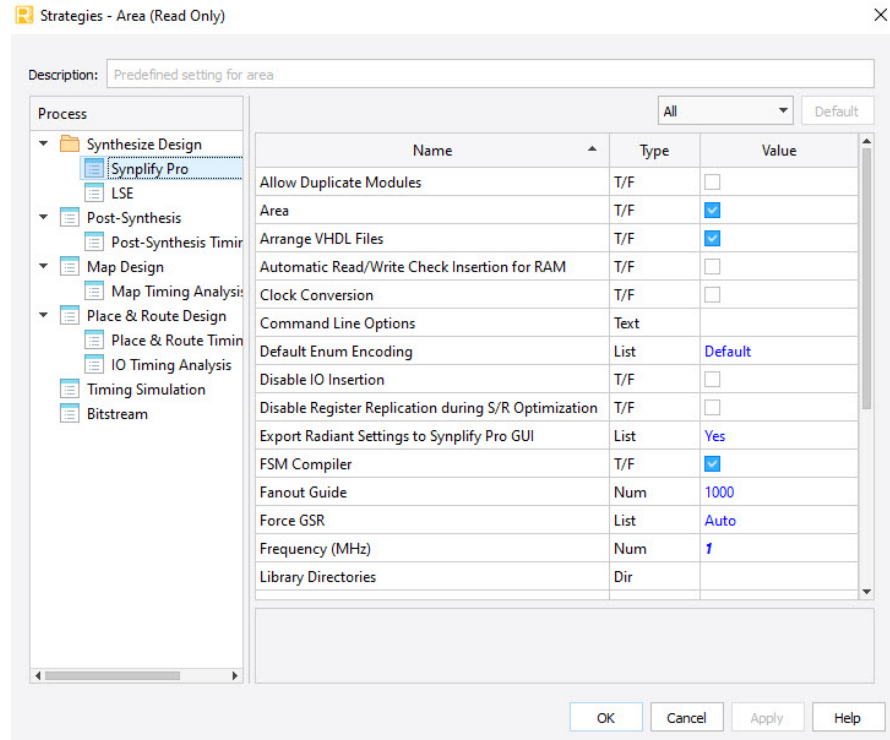
The default values are displayed in plain blue text. Modified values are displayed in italic bold text.

Strategies are design data independent and can be exported and used in multiple projects.

Area

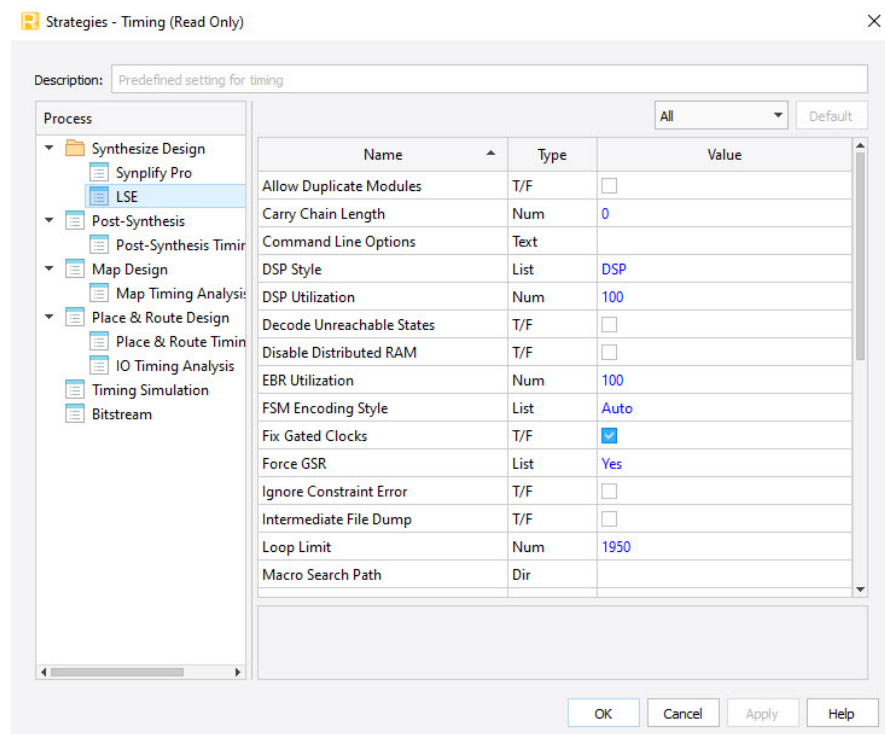
The Area strategy is a predefined strategy for area optimization. Its purpose is to minimize the total logic gates used while enabling the tight packing option available in Map.

Applying this strategy to large and dense designs might cause difficulties in the place and route process, such as longer time or incomplete routing.

Figure 51: Area Predefined Strategy

Timing

The Timing strategy is a predefined strategy for timing optimization. Its purpose is to achieve timing closure. The Timing strategy uses a very high effort level in placement and routing. Use this strategy if you are trying to reach the maximum frequency on your design. If you cannot meet your timing requirements with this strategy, you can clone it and create a customized strategy with refined settings for your design. This strategy might increase your place-and-route run time compared to the Area strategy.

Figure 52: Timing Predefined Strategy

User-Defined

You can define your own customized strategy by cloning and modifying any existing strategy. You can start from either a predefined or a customized strategy.

Constraint Propagation Options

This page lists all the strategy options associated with the Constraint Propagation process.

Propagation Debug Mode Using Propagation Debug Mode can affect timing of other circuits and can lead to false timing reports.

Synplify Pro Options

This page lists all the strategy options associated with the Synplify Pro Synthesis process. For information on their use in Synplify Pro, see the *Synopsys Synplify Pro for Lattice Reference Manual*.

Allow Duplicate Modules (for Synplify Pro) Allows the use of duplicate modules in your design.

When it is set to True, the last definition of the module is used by the software and any previous definitions are ignored. The default is False.

Area (for Synplify Pro) Specifies optimization preference for area reduction over timing delay reduction.

The True option specifies the area reduction mode. When set to True, this setting overrides the setting in [Frequency \(MHz\) \(for Synplify Pro\)](#).

The default is False.

This option is equivalent to the “set_option -frequency 1” command in Synplify Pro.

Arrange VHDL Files Allows Synplify Pro to reorder the VHDL source files for synthesis.

The default is True for VHDL or VHDL design entry type projects, and False for other projects. When this is set to False, Synplify Pro will use the file order in the Radiant software File List view.

Auto Compile Point With Automatic Compile Points, synthesis decides based on existing hierarchy and utilization estimates.

Compile Points minimize the amount of netlist changes resulting from RTL code changes. Compile points create logical boundaries where optimization will not occur. This may sacrifice some design performance for design result predictability. No user intervention is needed aside from enabling the flow.

Note:

Auto Compile Point only supports the LAV-AT-E device.

Automatic Read/Write Check Insertion for RAM When this option is set in the strategy, the synthesis tool inserts glue logic around inferred RAM to avoid simulation mismatches caused by indeterminate output values when the read and write addresses are same. By default this option is OFF.

Users should design to make sure the read and write addresses are never the same.

Clock Conversion Controls gated and generated clock conversion.

Values are True and False.

Command Line Options (for Synplify Pro) Enables additional command line options for the Synplify Pro Synthesis process.

To enter a command line option:

1. In the Strategy dialog box, select **Synplify Pro** in the Process list.

2. Double-click the Value column for the Command line Options option.
3. Type in the option and its value (if any) in the text box.
4. Click **Apply**.

For example:

```
set_option -library_path c:/source
```

Default Enum Encoding (For VHDL designs) Defines how enumerated data types are implemented.

The type of implementation affects the performance and device utilization. Available options are:

- ▶ Default – Automatically assigns an encoding style based on the number of states:
 - ▶ Sequential: 0-4 enumerated types
 - ▶ Onehot: 5-40 enumerated types
 - ▶ Gray: more than 40 enumerated types
- ▶ Gray – Only one bit of the state register changes at a time, but because more than one bit can be hot, the value must be decoded to determine the state. For example: 000, 001, 011, 010, 110
- ▶ Onehot – Only two bits of the state register change (one goes to 0; one goes to 1) and only one of the state registers is hot (driven by a 1) at a time. For example: 0000, 0001, 0010, 0100, 1000
- ▶ Sequential – More than one bit of the state register can change at a time, but because more than one bit can be hot, the value must be decoded to determine the state. For example: 000, 001, 010, 011, 100

This option is equivalent to the “set_option -default_enum_encoding default | onehot | gray | sequential” command in Synplify Pro.

Disable IO Insertion (for Synplify Pro) Controls whether the synthesis tool will add I/O buffers into your design.

If this is set to True, Synplify Pro will not add I/O buffers into your design. If it is set to False (default), the synthesis tool will insert I/O buffers into your design.

This option is equivalent to the “set_option -disable_io_insertion 1 | 0” command in Synplify Pro.

Disable Register Replication During S/R Optimization (Not applicable for iCE40 UltraPlus): When this option is set in the strategy, the synthesis tool will NOT duplicate the registers while inferring the address pointers for the RAM during Shift-register inference. By default this option is OFF and the tool will duplicate the address pointer registers to get better performance.

Export Radiant Software Settings to Synplify Pro GUI Controls whether the strategy settings are exported to Synplify Pro during interactive synthesis (opening Synplify Pro through the Tools menu). After opening Synplify Pro, you can change settings in Synplify Pro's interface. This option has no effect with integrated or stand-alone synthesis.

Available options are:

- ▶ No – Synplify Pro opens with its own defaults, ignoring the strategy settings.
- ▶ Yes (default) – Synplify Pro opens with the strategy settings every time. Options set and saved in a previous Synplify Pro session are ignored.
- ▶ Only on First Launch – Synplify Pro opens with the strategy settings the first time only. After that, Synplify Pro opens with settings saved in a previous session or with its own defaults. After the first time, the strategy settings are ignored.

For more information, see “Interactive Synthesis” on page 584.

FSM Compiler (for Synplify Pro) Enables or disables the FSM Compiler and controls the use of FSM synthesis for state machines.

When Synplify Pro is selected as the synthesis tool, it enables or disables the FSM Compiler and controls the use of FSM synthesis for state machines. When this is set to True (default), the FSM Compiler automatically recognizes and optimizes state machines in the design. 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.

This option is equivalent to the “set_option -symbolic_fsm_compiler 1 | 0” command in Synplify Pro.

Fanout Guide Controls fanout during synthesis. When the specified fanout limit is achieved, logic will be duplicated.

The default is 1000.

This option is equivalent to the “set_option -maxfan <number>” command in Synplify Pro.

Force GSR (for Synplify Pro) Forces Global Set/Reset Pin usage.

Available options are:

- ▶ Auto (default) – Allows the software to decide whether to infer Global Set/Reset in your design.
- ▶ False – Does not infer Global Set/Reset in your design.
- ▶ True – Always infers Global Set/Reset in your design.

This option is equivalent to the “set_option -force_gsr auto | yes | no” command in Synplify Pro.

Frequency (MHz) (for Synplify Pro) Specifies the global design frequency (in MHz). Nothing in the Value column means "auto" and Synplify Pro will try to maximize the frequency of the clocks.

The setting is ignored when [Area \(for Synplify Pro\)](#) is set to True.

This option is equivalent to the "set_option -frequency <number> | auto" command in Synplify Pro.

Library Directories Specifies all the paths to the directories which contain the Verilog library files to be included in your design for the project.

You can also add custom library files with module definitions for the design in a single file. The names of files read from the library path must match module names. Mismatches result in error messages.

Number of Critical Paths (for Synplify Pro) Specifies the number of critical timing paths to be reported in the timing report.

This option is equivalent to the "set_option -num_critical_paths <number>" command in Synplify Pro.

Number of Start/End Points Specifies the number of start and end points you want the software to report in the critical path section of the timing report.

This option is equivalent to the "set_option -num_startend_points <number>" command in Synplify Pro.

Output Netlist Format (for Synplify Pro) Outputs a mapped VHDL netlist for post-synthesis simulation.

Available options are: None (default) and VHDL.

This option is equivalent to the "set_option -write_vhdl 1 | 0" command in Synplify Pro.

Pipelining and Retiming Enables the pipelining and retiming features to improve design performance.

Values are:

- ▶ None – Disables the pipelining and retiming features.
- ▶ Pipelining Only (default) – Runs the design at a faster frequency by moving registers into the multiplier, creating pipeline stages.
- ▶ Pipelining and Retiming – When enabled, registers may be moved into combinational logic to improve performance.

This option is equivalent to the "setup_option -pipe 1 | 0 -retiming 1 | 0" command in Synplify Pro.

Push Tristates When this is set to True, the Synplify Pro compiler pushes tristates through objects such as muxes, registers, latches, buffers, nets, and tristate buffers, and propagates the high impedance state.

The high-impedance states are not pushed through combinational gates such as ANDs or ORs.

The default is False.

This option is equivalent to the “set_option -compiler_compatible 1 | 0” command in Synplify Pro.

Resolve Mixed Drivers (for Synplify Pro) If a net is driven by a VCC or GND and active drivers, setting this option to True will connect the net to the VCC or GND driver.

This option is equivalent to the “set_option -resolve_multiple_driver 1 | 0” command in Synplify Pro.

Resource Sharing (for Synplify Pro) When this is set to True (default), the synthesis tool uses resource sharing techniques to optimize area.

With resource sharing, synthesis 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.

This option is equivalent to the “set_option -resource_sharing 1 | 0” command in Synplify Pro.

Resynthesize All When this is set to True (default), Synplify Pro will resynthesize all portions, modules, or files of the RTL source. When this is set to False, Synplify Pro will synthesize only the updated portions, modules or files of the RTL source since the last run result.

If you run the “Force Run From Start” process, all portions, modules, or files of the RTL source will be resynthesized, regardless of whether this option is set to True or False.

Update Compile Point Timing Data Determines whether (True) or not (False) changes inside a compile point can cause the compile point (or top-level) containing it to change accordingly.

When this is set to False (default), Synplify Pro keeps the top level module the same, which is desired by incremental flow.

When this is set to True, changes in low level partitions will be propagated to top partitions up to top module. Synplify Pro will possibly optimize timing data and certainly will write a new timestamp onto the partition for the top level module.

This option is equivalent to the “set_option -update_models_cp 1 | 0” command in Synplify Pro.

Use Clock Period for Unconstrained I/O Controls whether to forward annotate constraints for I/O ports without explicit user-defined constraints.

When this is set to True, only explicit I/O port constraints are forward annotated. When it is set to False (the default), all I/O port constraints are forward annotated.

This option is equivalent to the “set_option -auto_constraint_io 1 | 0” command in Synplify Pro.

VHDL 2008 (for Synplify Pro) When this is set to True, VHDL 2008 is selected as the VHDL standard for the project.

LSE Options

This page lists all the strategy options associated with the LSE synthesis process.

Allow Duplicate Modules (for LSE)

When set to True, allows the design to keep duplicate modules. LSE issues a warning and uses the last definition of the module. Any previous definitions are ignored. The default is False, which causes an error if there are duplicate modules.

Allow Mixed Assignments (for LSE) When set to True, having both blocking and non-blocking assignments to the same variable will be allowed. A warning will be issued instead of an error.

Carry Chain Length Specifies the maximum number of carry chain cells (CCUs) that get mapped to a single carry chain. Default is 0, which is interpreted as infinite length.

This option is equivalent to the “-carry_chain_length” option in the SYNTHESIS command.

This option is equivalent to the “-allow_duplicate_modules” option in the SYNTHESIS command.

Command Line Options (for LSE) Enables additional command line options for the LSE Synthesis process.

To enter a command line option:

1. In the Strategy dialog box, select **LSE** in the Process list.
2. Double-click the Value column for the Command line Options option.
3. Type in the option and its value (if any) in the text box.
4. Click **Apply**.

For detailed description on LSE command line options, see [“Running SYNTHESIS from the Command Line” on page 158](#).

DSP Style Specifies how DSP modules should be implemented: with DSP resources or with Logic (LUTs).

This option is equivalent to the “-use_dsp” option in the SYNTHESIS command.

DSP Utilization Specifies the percentage of DSP sites that LSE should try to use.

This option is equivalent to the “-dsp_utilization” option in the SYNTHESIS command.

Decode Unreachable States When set to True, synthesis infers safe recovery logic from unreachable states in all the state machines of the design.

This option is equivalent to the “-decode_unreachable_states” option in the SYNTHESIS command.

Disable Distributed RAM When set to True, inferred memory will not use the distributed RAM of the PFUs.

EBR Utilization Specifies EBR utilization target setting in percent of total vacant sites. LSE will honor the setting and do the resource computation accordingly. Default is 100 (in percentage).

This option is equivalent to the “-bram_utilization” option in the SYNTHESIS command.

FSM Encoding Style Specifies the encoding style to use with the design.

This option is equivalent to the “-fsm_encoding_style” option in the SYNTHESIS command. Valid options are auto, one-hot, gray, and binary. The default value is auto, meaning that the tool looks for the best implementation.

Note

The encoding type “gray” only works with less than or equal to four machine states. When the number of machine states is large than four, LSE will use other encoding styles and issue the following warning message:

WARNING - Gray encoding is not supported for state machines with more than four states.

Fix Gated Clocks When set to True, LSE changes standard gated clocks to forms more effective for FPGAs. Clocks are gated with AND or OR gates to conserve power, but in FPGAs such clocks cause skew and prevent global clock resources from being used. The Fix Gated Clocks option is ignored if the Optimization Goal option is set to Area.

The gated clocks must be specified in the .ldc file with create_clock constraints. All inputs of the gating logic must be driven by primary inputs and the gating logic must be decomposable. Instantiated primitives and black boxes are not affected.

Converted clocks and the associated registers are reported in the synthesis.log file.

Force GSR (for LSE) (Not applicable for iCE40 UltraPlus)

Forces Global Set/Reset Pin usage. (Not applicable for iCE40 UltraPlus).

Available options are:

- ▶ Auto – Allows the software to decide whether to infer Global Set/Reset in your design.
- ▶ Yes (default) – Always infers Global Set/Reset in your design.
- ▶ No – Does not infer Global Set/Reset in your design.

Ignore Constraint Errors Enables or disables the LSE synthesis process to ignore constraint errors.

By default, the box is unchecked (False), and LSE terminates processing and issue an error message when a constraint errors are encountered.

When the box is checked (True), LSE will ignore constraint errors and continue processing.

The LSE synthesis report includes the setting of this option and the constraint errors found.

Intermediate File Dump If you set this to True, LSE will produce intermediate encrypted Verilog files. If you supply Lattice with these files, they can be decrypted and analyzed for problems. This option is good for analyzing simulation issues.

This option is equivalent to the “-ifd” option in the SYNTHESIS command.

Loop Limit Specifies the maximum number of iterations of “for” and “while” loops in the source code. The limit is applied when the loop index is a variable, not when it is a constant. The higher the loop_limit, the longer the run time. The default value is 1950. Setting a higher value may cause stack overflow during some of the optimizations during synthesis. A lower value will be ignored and the default used instead.

This option is equivalent to the “-loop_limit” option in the SYNTHESIS command.

Macro Search Path (for LSE) Allows you to specify a path (or paths) to locate physical macro files used in a given design. The software will add the specified paths to the list of directories to search when resolving file references. The option can also be used for indicating the directories containing include files that are specified in the RTL design files.

You don't need to specify a search path if the necessary file is in the directory containing the top-level .ngo file or if the FILE attribute in the design gives a complete path name for the file (instead of a relative path name).

The software follows the following order to search:

1. Current implementation directory
2. Project directory

3. Directories where the LPC or IPX source files reside
4. User-specified macro search paths

To specify a macro search path, double-click the Value box, and directly enter the path or click the ... button to browse for one or more paths.

This option is equivalent to the “-p” option in the SYNTHESIS command.

Max Fanout Limit Specifies the maximum fanout setting. LSE will make sure that any net in the design is not exceeding this limit. Default is 1000 fanouts.

This option is equivalent to the “-max_fanout” option in the SYNTHESIS command.

Memory Initial Value File Search Path (for LSE) Allows you to specify a path (or paths) to locate memory initialization file (.mem) used in a given design. The software will add the specified path(s) to the list of directories to search when resolving file references.

To specify a search path, double-click the Value box, and directly enter the path or click the ... button to browse for one or more paths.

This option is equivalent to the “-p” option in the SYNTHESIS command.

Optimization Goal Enables LSE to optimize the design for area or speed.

Valid options are:

- ▶ **Area** – Optimizes the design for area by reducing the total amount of logic used for design implementation.

When Optimization Goal is set to Area, LSE honors the LDC constraints if there are any. If [Use IO Registers](#) is set to Auto, LSE packs input and output registers into I/O pad cells.

Note

With the Area setting, LSE also ignores all SDC constraints. These constraints are not used by LSE and are not added for use by the later stages of implementation.

- ▶ **Timing** – Optimizes the design for speed by reducing the levels of logic.

When Optimization Goal is set to Timing and a create_clock constraint is available in an .ldc file, LSE ignores the [Target Frequency](#) setting and uses the value from the create_clock constraint instead.

If there are multiple clocks, and if not all the clocks use create_clock constraint, then LSE will assign 200 MHz constraint on the remaining clocks in Timing Mode.

If [Use IO Registers](#) is set to Auto, LSE does not pack input and output registers into I/O pad cells.

The default setting depends on the device type.

For more information, see “Optimizing LSE for Area and Timing” on page 256.

This option is equivalent to the “-optimization_goal” option in the SYNTHESIS command.

Propagate Constants When set to True (default), enables constant propagation to reduce area, where possible. LSE will then eliminate the logic used when constant inputs to logic cause their outputs to be constant.

You can turn off the operation by setting this option to False.

This option is equivalent to the “-propagate_constants” option in the SYNTHESIS command.

RAM Style Sets the type of random access memory globally to distributed, embedded block RAM, or registers.

The default is Auto which attempts to determine the best implementation, that is, the synthesis tool will map to technology RAM resources (EBR/Distributed) based on the resource availability.

This option will apply a syn_ramstyle attribute globally in the source to a module or to a RAM instance. To turn off RAM inference, set its value to Registers.

- ▶ Registers – Causes an inferred RAM to be mapped to registers (flip-flops and logic) rather than the technology-specific RAM resources.
- ▶ Distributed – Causes the RAM to be implemented using the distributed RAM or PFU resources.
- ▶ Block_RAM – Causes the RAM to be implemented using the dedicated RAM resources. If your RAM resources are limited, for whatever reason, you can map additional RAMs to registers instead of the dedicated or distributed RAM resources using this attribute.

This option is equivalent to the “-ramstyle” option in the SYNTHESIS command.

ROM Style Allows you to globally implement ROM architectures using dedicated, distributed ROM, or a combination of the two (Auto).

This applies the syn_romstyle attribute globally to the design by adding the attribute to the module or entity. You can also specify this attribute on a single module or ROM instance.

Specifying a syn_romstyle attribute globally or on a module or ROM instance with a value of:

- ▶ Auto (default) – Allows the synthesis tool to choose the best implementation to meet the design requirements for speed, size, and so on.
- ▶ Logic – Causes the ROM to be implemented using the distributed ROM or PFU resources. Specifically, the logic value will implement ROM to logic (LUT4) or ROM technology primitives.

- ▶ **EBR** – Causes the ROM to be mapped to dedicated EBR block resources. ROM address or data should be registered to map it to an EBR block. If your ROM resources are limited, for whatever reason, you can map additional ROM to registers instead of the dedicated or distributed RAM resources using this attribute.

Infer ROM architectures using a CASE statement in your code. For the synthesis tool to implement a ROM, at least half of the available addresses in the CASE statement must be assigned a value. For example, consider a ROM with six address bits (64 unique addresses). The CASE statement for this ROM must specify values for at least 32 of the available addresses.

This option is equivalent to the “-romstyle” option in the SYNTHESIS command.

Read Write Check on RAM (iCE40UP Only) Adds (True) or does not add (False) the glue logic to resolve read/write conflicts wherever needed. Default is False.

Remove Duplicate Registers Specifies the removal of duplicate registers.

When set to True (default), LSE removes a register if it is identical to another register. If two registers generate the same logic, the second one will be deleted and the first one will be made to fan out to the second one's destinations. LSE will not remove duplicate registers if this option is set to False.

This option is equivalent to the “-remove_duplicate_regs” option in the SYNTHESIS command.

Remove LOC Properties (for LSE) Setting this to On removes LOC properties in the synthesized design.

Report Trimmed User Nets (for LSE) When set to True, LSE will report whenever a user-declared HDL net is trimmed from the netlist.

Resolve Mixed Drivers (for LSE) If a net is driven by a VCC or GND and active drivers, setting this option to True connects the net to the VCC or GND driver.

Resource Sharing (for LSE) When this is set to True (default), the synthesis tool uses resource sharing techniques to optimize area.

With resource sharing, synthesis 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.

This option is equivalent to the “-resource_sharing” option in the SYNTHESIS command.

Target Frequency Specifies the target frequency setting. This frequency applies to all the clocks in the design. If there are some clocks defined in an .ldc file, the remaining clocks will get this frequency setting. When a

`create_clock` constraint is available in an .lxc file, LSE ignores the Target Frequency setting for that clock and uses the value from the `create_clock` constraint instead.

This option is equivalent to the “-frequency” option in the SYNTHESIS command.

Use Carry Chain Turns on (True) or off (False) carry chain implementation for adders. Default is True.

This option is equivalent to the “-use_carry_chain” option in the SYNTHESIS command.

Use IO Insertion When set to True, LSE uses I/O insertion and GSR.

This option is equivalent to the “-use_io_insertion” option in the SYNTHESIS command.

Use IO Registers When True, this option forces the synthesis tool to pack all input and output registers into I/O pad cells based on the timing requirements for the target device family. Auto, the default setting, enables this register packing if [Optimization Goal](#) is set to Area. If Optimization Goal is Timing or Balanced, Auto disables register packing.

This option is equivalent to the “-use_io_reg” option in the SYNTHESIS command.

You can also control packing on individual registers and ports. See “syn_useioff” on page 658.

VHDL 2008 (for LSE) When this is set to True, VHDL 2008 is selected as the VHDL standard for the project.

Post-Synthesis Options

This page lists all strategy options associated with the Post-Synthesis process. The options available for user setting are dependent on the target device of your project.

Command Line Options (for Post-Synthesis)

Enables additional command line options for the associated process.

External Module Files (.udb)

Allows the user to supply already synthesized modules for design assembly into the top level design. These modules must be already synthesized in Unified Database (.udb) format.

For example:

```
c:\case1\ip1.udb; d:\example\aaa\abc.udb
```


Multiple .udb files can be separated by ‘;’

This internal process, Post-Synthesis, performed after logic synthesis resolves and assembles lower level modules to top level to complete the design contents.

All modules contents must be resolved at this stage before the flow can continue to the next stage.

Hardware Evaluation

Enables or disables the ability to temporarily test IP in a device without an IP license. If enabled, a timer is added to the design that allows unlicensed IP to function for about 4 hours in a device. If disabled, you cannot generate a bitstream if there are any unlicensed IP in the design.

You might want to disable this option to refine your design while waiting for the license. You will not be able to generate a bitstream, but you will be able to see how resources are used (without the timer) and close timing. When you get the license, you can then generate the bitstream.

Note:

The Hardware Evaluation option is only for a device that does not support bitstream encryption ability even with a built-in hardware evaluation timer (i.e., LAV-ATE device).

Post-Synthesis Timing Analysis Options

This page lists all strategy options associated with the Post-Synthesis Timing Analysis process.

Number of End Points

Controls the number of endpoints in the critical endpoint summary.

Number of Paths Per Constraint (for Post-Synthesis Timing Analysis)

Lists detailed logic and route delays for all constrained paths and nets in the design.

Number of Paths Per Endpoint (for Post-Synthesis Timing Analysis)

Controls maximum number of paths that could be reported for each endpoint.

Number of Unconstrained Paths (for Post-Synthesis Timing Analysis)

Reports paths not covered by a timing preference.

Report Format (for Post-Synthesis Timing Analysis) Specifies the report format type.

- ▶ Lattice Standard – Displays timing report format that is similar to formats used in many industry timing tools.
- ▶ Diamond Style – Displays timing report that is similar to Diamond software TRACE report.

Timing Analysis Options (for Post-Synthesis Timing Analysis)

Specifies the analysis type.

- ▶ Hold Analysis – Performs hold analysis.
- ▶ Standard Setup Analysis – Performs setup time checks.
- ▶ Standard Setup and Hold Analysis – Performs both the Standard Setup Analysis and the Hold Analysis.

Map Design Options

This page lists all strategy options associated with the Map Design process. The options available for user setting are dependent on the target device of your project.

Command Line Options (for Map Design) Enables additional command line options for the associated process.

To enter a command line option:

1. In the Strategy dialog box, select the associated process in the Process list.
2. Double-click the Value column for the Command line Options option.
3. Type in the option and its value (if any) in the text box. For example: -exp parPathBased=ON
4. Click **Apply**.

To reference more information about command line options for the Map Design process, type `map -h <architecture>` in a command line window. For detailed options description, refer to “Running MAP from the Command Line” on page 165.

Ignore Constraint Errors Enables or disables the Map Design process to ignore constraint errors.

By default, the box is unchecked (False), and Map Design terminates processing and issues an error message when constraint errors are encountered.

When the box is checked (True), Map Design will ignore constraint errors and continue processing.

The Map Design process report includes the setting of this option and the constraint errors found.

Infer GSR (Not applicable for iCE40 UltraPlus): Enables or disables the GSR inferencing.

GSR inference is only applicable to PLC slice registers by default. Each black-box can have its own rules to guide mapper to perform GSR inference. Rules to be applied according to device architecture specification. For example, in LIFCL device, it is applicable to SLICE / IO registers, block RAM, large RAM, and DDR components.

When multiple GSR instances are found in the design, mapper is able to merge them if their outputs are actually the same wire in the netlist. Otherwise mapper will issue an error.

Mapper can also create a GSR component if user specified the GSR signal in the constraint file.

Report Signal Cross Reference When this is set to True, the map report (.mrp) will show where nets in the logical design were mapped in the physical design.

The default is False.

Report Symbol Cross Reference When this is set to True, the map report (.mrp) will show where symbols in the logical design were mapped in the physical design.

The default is False.

Map Timing Analysis Options

This page lists all strategy options associated with the Map Timing Analysis process.

Number of End Points Controls the number of endpoints in the critical endpoint summary.

Number of Paths Per Constraint (for Map Timing Analysis) Lists detailed logic and route delays for all constrained paths and nets in the design.

Number of Paths Per Endpoint (for Map Timing Analysis) Controls maximum number of paths that could be reported for each endpoint.

Number of Unconstrained Paths (for Map Timing Analysis) Reports paths not covered by a timing preference.

Report Format (for Map Timing Analysis) Specifies the report format type.

- ▶ Lattice Standard – Displays timing report format that is similar to formats used in many industry timing tools.

- ▶ **Diamond Style** – Displays timing report that is similar to Diamond software TRACE report.

Speed for Hold Analysis Specifies performance grade for hold analysis. This option allows you to override the default m (minimum) performance grade for hold time analysis, which represents faster silicon than the fastest performance grade of the device being targeted.

Speed for Setup Analysis Specifies performance grade for setup analysis. This option allows you to override the default performance grade which runs setup analysis against the performance grade of the device currently targeted by the project implementation.

Timing Analysis Options (for Map Timing Analysis) Specifies the analysis type.

- ▶ **Hold Analysis** – Performs hold analysis.
- ▶ **Standard Setup Analysis** – Performs setup time checks.
- ▶ **Standard Setup and Hold Analysis** – Performs both the Standard Setup Analysis and the Hold Analysis.

Place & Route Design Options

This page lists all strategy options associated with the Place & Route Design process. The options available for user setting are dependent on the target device of your project.

Command Line Options (for Place & Route Design) Allows you to specify options from the par command without directly using the command line. Type in a string of options without the par command. For example: `-exp parPathBased=ON:parHold=1`

For detailed descriptions of placement, routing, and PAR explorer (-exp) command line options, see “Running PAR from the Command Line” on page 167.

Disable Auto Hold Timing Correction (for Place & Route Design) When the switch is used, PAR will not check the hold timing of the design, so no correction of potential hold timing violations will be performed.

Disable Timing Driven (for Place & Route Design) Enables or disables the timing-driven option for the PAR run.

When this is set to True, the timing-driven option for the PAR run will not be used. If this is set to False, PAR automatically uses the timing-driven option if the Timing Wizard is present and if any timing constraints are found in the preference file. If selected, the timing-driven option is not invoked in any case and cost-based placement and routing are done instead.

Two examples of situations in which you might disable this option are:

- ▶ You have timing constraints specified in your preference file, but you want to execute a quick PAR run without using the timing-driven option to give you a rough idea of how difficult the design is to place and route.
- ▶ You only have a single license for the timing-driven option but you want to use this license for another application (for example, to perform timing-driven routing within EPIC) that will run at the same time as PAR. This option keeps the license free for the other application.

Multi-Tasking Node List Allows you to specify the node file name for the multi-tasking PAR.

The multi-tasking PAR allows you to use multiple machines (nodes) that are networked together for a multi-run PAR job, significantly reducing the total amount of time for completion.

For more information on multi-tasking PAR, see “Running Multiple PAR Jobs in Parallel” on page 277.

Number of Host Machine Cores Allows you to run multiple threads on your local machine by specifying how many cores to be used from your local machine. The range is from 0 to 64.

Pack Logic Block Utility Sets the relative density (of available slices) at which the slices within a device are to be packed, in terms of a percentage of the available slices in the device.

This option has great control of the packing density. The value range is 0-100 percent (100 = minimum packing; 0 = maximum density).

If this option is not specified (blank), it defaults to a percentage that depends on which device is selected. For example, for LIFCL, the default is 75 percent. The result will be a less dense packing, depending on the size of the design relative to the number of available slices in the device. If the design is large compared with the number of available slices in the device, the placer will make a reasonable effort to pack the design so that it fits in the device.

If you specify a density value for this option, the placer will attempt to pack the device to that density. The “0” setting results in the densest mapping. If the design is large compared with the target density, the placer will make an aggressive packing effort to meet your target. However, this may adversely impact the design f_{MAX} performance.

Path-based Placement Allows you to apply path-based placement. Path-based placement gives better performance and more predictable results.

Options are Off and On.

Note:

The Pack Logic Block Utility and Path-based Placement strategies are not supported in the LAV-AT-E device.

Placement Iteration Start Point Specifies the cost table to use (from 1-100) to begin the PAR run.

The default is 1. Cost tables are not an ordered set. There is no correlation between a cost table's number and its relative value. If cost table 100 is reached, placement does not begin at 1 again, even if command options specify that more placements should be performed.

Placement Iterations Specifies the maximum number of placement/routing passes (0-100, 0 = run until solved) to be run (regardless of whether they complete) at the Placement Effort Level.

Each iteration uses a different cost table when the design is placed and will produce a different Uniform Database (.udb) file. If you specify a Starting Cost Table, the iterations begin at that table number.

Placement Save Best Run Determines the number (1-100) of best outputs of the Place and Route run to save (defaults to 1).

If no number is specified, all output designs produced by the PLACE & ROUTE run are saved. The best outputs are determined by a scoring system described in the section titled Scoring the Routed Design.

This option does not care how many iterations you performed or how many effort levels were used. It compares every result to every other result.

Prioritize Hold Correction Over Setup Performance (for Place & Route Design) During hold timing correction, there may be situations when correcting a hold timing violation would cause a setup requirement to become violated. By default, we do not correct the hold violation on such connections so as to preserve the setup performance, but when this switch is used, we will attempt to correct the hold violation and let the setup requirement become violated.

Run Placement Only Setting this to True prevents the design from being routed. PAR will output a placed, but not routed Unified Design Database(.udb) file.

This option defaults to False.

Set Speed Grade for Hold Optimization (for Place & Route Design)

This overrides the default speed grade used to perform hold timing analysis.

Set Speed Grade for Setup Optimization Change performance grade for setup optimization.

Stop Once Timing is Met (for Place & Route Design) Setting this to True forces the Place and Route Design process to stop as soon as the timing requirement is satisfied. This option has no effect if the "Generate Timing Analysis report for each iteration" option is set to True or if using Run Manager to produce multiple place-and-route runs.

Place & Route Timing Analysis Options

This page lists all strategy options associated with the Place & Route Timing Analysis process.

Number of End Points (for Place & Route Timing Analysis) Controls the number of endpoints in the critical endpoint summary.

Number of Paths Per Constraint (for Place & Route Timing Analysis)

Lists detailed logic and route delays for all constrained paths and nets in the design.

Number of Paths Per Endpoint (for Place & Route Timing Analysis)

Controls maximum number of paths that could be reported for each endpoint.

Number of Unconstrained Paths (for Place & Route Timing Analysis)

Reports paths not covered by a timing preference.

Report Format (for Place & Route Timing Analysis) Specifies the report format type.

- ▶ Lattice Standard – Displays timing report format that is similar to formats used in many industry timing tools.
- ▶ Diamond Style – Displays timing report that is similar to Diamond software TRACE report.

Speed for Hold Analysis Specifies performance grade for hold analysis. This option allows you to override the default m (minimum) performance grade for hold time analysis, which represents faster silicon than the fastest performance grade of the device being targeted.

Speed for Setup Analysis Specifies performance grade for setup analysis. This option allows you to override the default performance grade which runs setup analysis against the performance grade of the device currently targeted by the project implementation.

Timing Analysis Options (for Place & Route Timing Analysis) Specifies the analysis type.

- ▶ Hold Analysis – Performs hold analysis.
- ▶ Standard Setup Analysis – Performs setup time checks.
- ▶ Standard Setup and Hold Analysis – Performs both the Standard Setup Analysis and the Hold Analysis.

IO Timing Analysis Options

The following option is associated with the IO Timing Analysis process.

All Performance Grade Controls whether the I/O timing report (.ior) will give an in-depth analysis on all available performance grades or will just produce a summary report on the worst-case performance grade.

- ▶ True - The I/O Timing Report will summarize the worst-case scenario of all available performance grades.
- ▶ False (default) - The I/O Timing Report will only contain a summary of the worst-case performance grade for the given device.

Timing Simulation Options

This page lists all strategy options associated with the Timing Simulation process. The options available for user setting are dependent on the target device of your project.

Generate PUR in the Netlist When this is set to False, the timing simulation file generation process will not write PUR instance in the Verilog/VHDL back-annotation netlist. Then you have to instantiate PUR in the test bench.

Generate X for Setup/Hold Violation When this is set to True, the Timing Simulation process will place X notifiers in the output file on flip-flops with setup and/or hold time violations.

Multichip Module Prefix Adds a prefix to module names to make them unique for multi-chip simulation.

Negative Setup-Hold Times Allows you to select negative setup time and negative hold time for better accuracy.

The default is True. You can set it to False for those simulators that might not be able to handle negative setup- hold times.

Retarget Speed Grade Retargets back annotation to a different performance grade than the one used to create the Uniform Database (.udb) file.

You are limited to those performance grades available for the device used in the UDB file.

Timing Check With Min Speed Grade Setting this to True replaces all timing information for back annotation with the minimum timing for all paths.

This option is used for simulation of hold time requirements. Separate simulations are required for hold time verification (-min switch) and delay time verification (normal output).

Timing Simulation Max Delay between Buffers (ps) Distributes routing delays by splitting the signal and inserting buffers. The delay value assigned represents the maximum delay number in picoseconds between each buffer (1000 ps by default).

Verilog Hierarchy Separator Specifies the hierarchy separator character which will be used in name generation when the design hierarchy is flattened.

You can specify the following two special characters as the hierarchy separator: "/" (back-slash) or "." (period).

Enter the character as is in the edit box. Encapsulate the character with double quotes, for example, "/", ".".

The option is only available for Verilog designs.

Bitstream Options

This page lists all strategy options associated with the bitstream generation process. The options available for user setting are dependent on the target device of your project.

Command Line Options Enables additional command line options for the associated process.

Enable Early IO Wakeup Enable IO output as soon as possible. (Not applicable for iCE40 UltraPlus)

Enable Timing Check When this is set to true and there is timing error in the Place and Route report file, a dialogue box will display before executing bitstream generation.

Enable Warm Boot Enables the Warm Boot functionality, provided the design contains an instance of the WARMBOOT primitive. (iCE40UP)

Initialize EBR Quadrant 0 Write the EBR initialization data into the bitstream for quadrant 0. (iCE40UP)

Initialize EBR Quadrant 1 Write the EBR initialization data into the bitstream for quadrant 1. (iCE40UP)

Initialize EBR Quadrant 2 Write the EBR initialization data into the bitstream for quadrant 2. (iCE40UP)

Initialize EBR Quadrant 3 Write the EBR initialization data into the bitstream for quadrant 3. (iCE40UP)

IP Evaluation When enabled, a bitstream will be generated for evaluation purposes when an IP license is not found, but will be limited to a maximum of four hours before the device resets itself. When disabled, a bitstream will not be generated if an IP license is not found (Not applicable for iCE40 UltraPlus).

No header Don't write the bitstream header section. (iCE40UP)

Set All Unused IO No Pullup

Removes the pullup on the unused I/Os, except Bank 3 I/Os which do not have pullup. (iCE40UP)

Oscillator Frequency Range

Options include Medium and Slow. Fast is not supported for iCE40UP. Only for NVCM configuration, not for programming

Depending on the speed of external PROM, this options adjusts the frequency of the internal oscillator used by the iCE40UP device during configuration. This is only applicable when the iCE40UP device is used in SPI Master Mode for configuration.

Output Format Specifies the type of bitstream to create for an FPGA device.

The following options are available:

- ▶ **Bit File (Binary)** – Generates a binary configuration file (.bin) that contains the default outputs of the Bit Generation process.
- ▶ **Raw Bit File (ASCII)** – Generates an ASCII raw bit text file (.rbt) of ASCII ones and zeros that represent the bits in the bitstream file. If you are using a microprocessor to configure a single FPGA, you can include the Raw Bit file in the source code as a text file to represent the configuration data. The sequence of characters in the Raw Bit file is the same as the bit sequence that will be written into the FPGA.
- ▶ **Hex File (Hexadecimal)** – Generates a Hexadecimal (.hex) PROM data file used for Programming into external non-volatile memory, such as parallel or Serial Peripheral Interface (SPI) Flash devices (iCE40UP only).
- ▶ **NVCM File (Non-volatile Configuration Memory)** – Generates a Non-volatile Configuration Memory (.nvcm). Each line in this file is a separate NVCM instruction for bitmap programming. The exceptions are comment lines (usually the first few lines in the file) which start with the # sign. Comment lines should be ignored when programming the NVCM array. The comment lines contain comments as well as header information (iCE40UP only).

Register Initialization Enable register initialization section of the bitstream (Not applicable for iCE40 UltraPlus).

Run DRC When this is set to True, the software runs a physical design rule check and saves the output to the Bit Generation report file (.bgn).

Running DRC before a bitstream or JEDEC file is produced will detect any errors that could cause the FPGA to function improperly. If no fatal errors are detected, it will produce a bitstream or JEDEC file. Run DRC is the default.

SPI Flash Low Power Mode

Places the PROM in low-power mode after configuration. (iCE40UP)

This option is applicable only when the iCE40UP device is used as SPI Master Mode for configuration.

Set NVCM Security

Ensures that the contents of the Non-Volatile Configuration Memory (NVCM) are secure and the configuration data cannot be read out of the device. (iCE40UP)

Common Tasks

Working with projects includes many tasks, including: creating the project, editing design files, modifying tool settings, trying different implementations and strategies, and saving your data.

Creating a Project

See “Creating a New Project” on page 12 for step-by-step instructions.

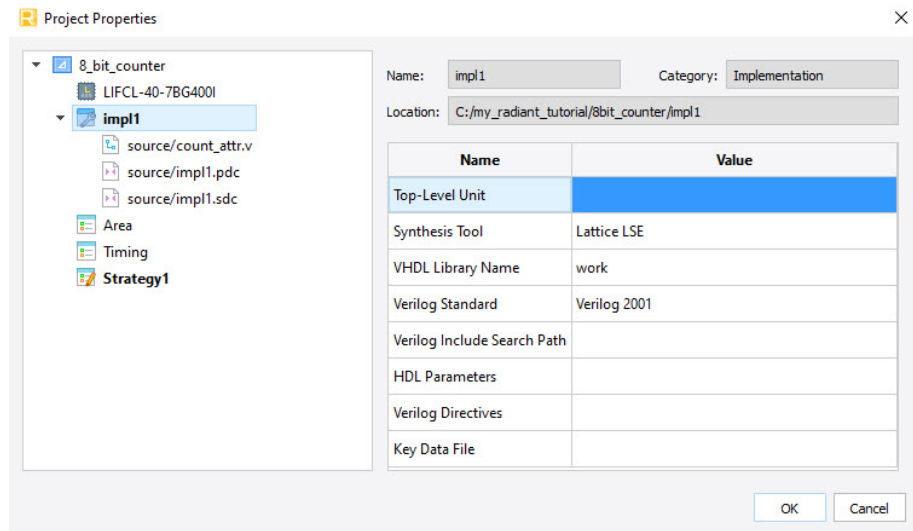
Changing the Target Device

There are two ways to access the Device Selector dialog box for changing the target device:

- ▶ Double-click the device in the project File List view or right-click it and choose **Edit**.
- ▶ Choose **Project > Device**.

Setting the Top Level of the Design

If multiple top levels exist in the hierarchy of your HDL source files, you will need to set the top-level design unit. After generating the hierarchy, choose **Project > Active Implementation > Set Top-Level Unit**. Alternatively, right-click the implementation and choose this command from the pop-up menu.

Figure 53: Top-Level Design Unit

In the Project Properties dialog box, select **Value** next to **Top-Level Unit** and select the desired top level from the list.

You can also use the Hierarchy View to set the top-level. Right-click a level you want to be the top-level in the Hierarchy View and choose **Set Top-Level Unit**.

Editing Files

You can open any of the files for editing by double-clicking or by right-clicking and choosing **Open** or **Open with**.

Saving Project Data

In the File menu are the following selections for saving your design and project data:

- ▶ Save – saves the currently active item.
- ▶ Save As – saves the active item using a different file name.
- ▶ Save All – saves all changed documents.
- ▶ Save Project – saves the current project.
- ▶ Save Project As – saves the active project using a different project name.
- ▶ Archive Project – creates a zip file of the current project in a location you specify.

Chapter 6

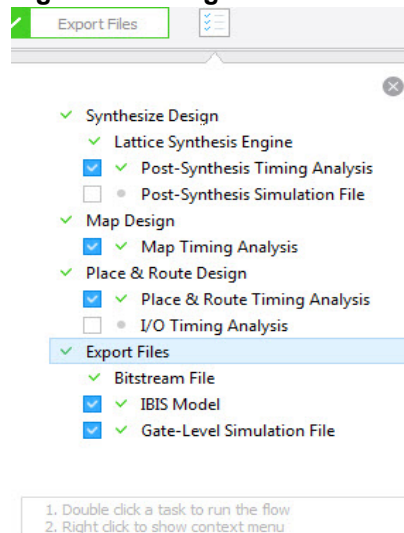
Radiant Software Design Flow

This chapter describes the design flow in the Radiant software. Running processes and controlling the flow for alternate what-if scenarios are explained.

Overview

The FPGA implementation design flow in the Radiant software provides extensive what-if analysis capabilities for your design. The design flow is displayed in the Task Detail View at the right end of the Process Toolbar.

Figure 54: Design Flow Shown in Task Detail View



Design Flow Processes

The design flow is organized into discrete processes, where each step allows you to focus on a different aspect of the FPGA implementation.

Synthesize Design This process runs the selected synthesis tool (Lattice Synthesis Engine is the default) in batch mode to synthesize your HDL design.

- ▶ Synthesis Tool - identifies the selected synthesis tool, Lattice Synthesis Engine or Synplify Pro.
- ▶ Post-Synthesis Timing Analysis - generates timing analysis files.
- ▶ Post-Synthesis Simulation File - generates a post-synthesis netlist file `<file_name>_syn.vo` used for Post-Synthesis Simulation.

Map Design This process maps the design to the target FPGA and produces a mapped Unified Database (.udb) design file. Map Design converts a design's logical components into placeable components.

- ▶ Map Timing Analysis - generates timing analysis files.

Place & Route Design This process takes mapped physical design files and places and routes the design. The output can be processed by the design implementation tools. Timing analysis files can also be generated.

- ▶ Place & Route Timing Analysis - generates timing analysis files.
- ▶ I/O Timing Analysis - generates I/O timing analysis files.

Export Files This process generates the IBIS, simulation, and programming files that you have selected for export:

- ▶ Bitstream File – generates a configuration bitstream (bit images) file, which contains all of the design's configuration information that defines the internal logic and interconnections of the FPGA, as well as device-specific information from other files.
- ▶ IBIS Model – generates a design-specific I/O Buffer Information Specification model file (.ibs). IBIS models provide a standardized way of representing the electrical characteristics of a digital IC's pins (input, output, and I/O buffers).
- ▶ Gate-Level Simulation File – generates a Verilog netlist of the routed design that is back annotated with timing information. This generated .vo file enables you to run a timing simulation of your design.

The files for export can also be generated separately by double-clicking each one.

Running Processes

You can perform the following actions for each step in the process flow:

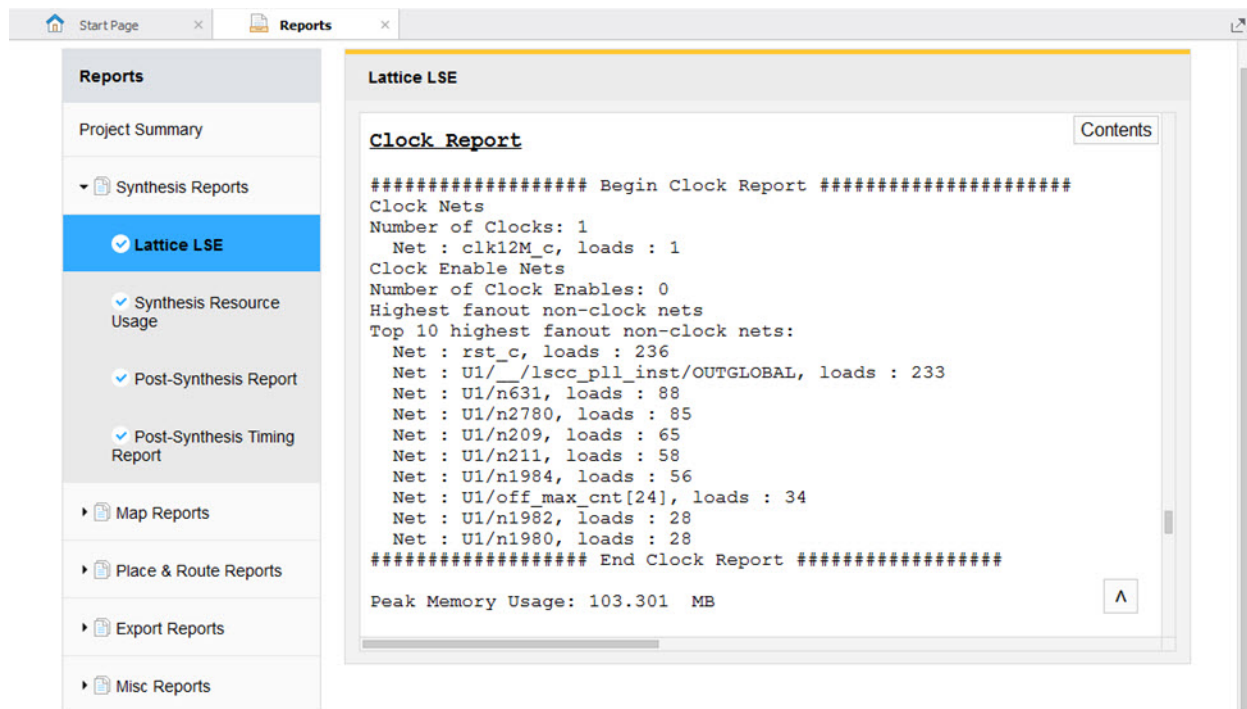
- ▶ Run – runs the process, if it has not yet been run.
- ▶ Force Run– reruns a process that has already been run.

- ▶ Force Run From Start – reruns all processes from the start to the selected process.
- ▶ Stop – stops a running process.
- ▶ Clean Up Process – clears the process state and puts a process into an initial state as if it had not been run.

The state of each process step is indicated with an icon to the left of the process. The process status icons description is described in [“Process” on page 30](#)

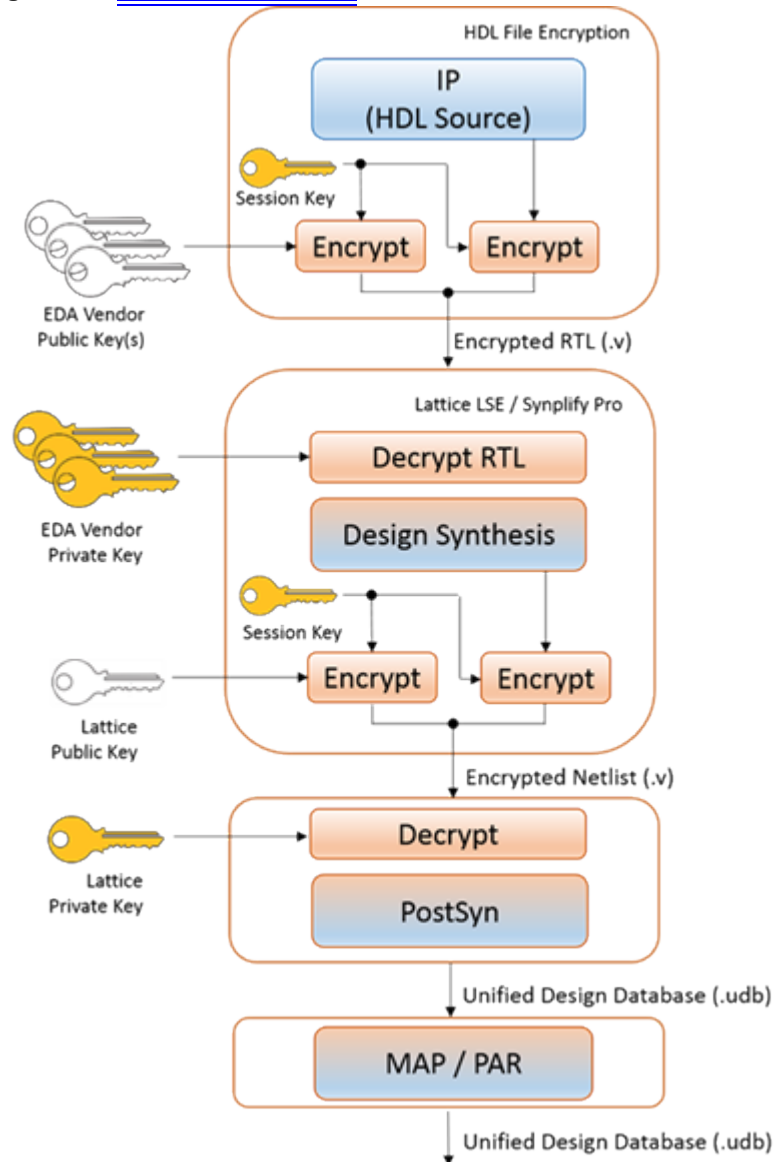
The Reports View displays detailed information about the process results, including the last process run. The Messages section shows warning and error messages and allows you to filter the types of messages that are displayed. See [“Reports” on page 32](#).

Figure 55: Reports View of Last Process Run



IP Encryption Flow

IP encryption flow enables you to protect your IP design. Following the industry standard, the Radiant software, through the IP encryption flow, allows the partnership between the IP Vendor, supported EDA vendor, and Lattice Semiconductor.

Figure 56: [IP Encryption Flow](#)

The encryption flow uses symmetric and asymmetric encryption methods to maximize the design security. The symmetric method only involves a single symmetric key for both, encryption and decryption. The asymmetric method involves the public-private key pair. The public key is published by a vendor and is used by the Radiant software. The private key is never revealed to the public.

The Radiant software supports these cryptographic algorithms:

- ▶ AES-128/AES-256: symmetric algorithm used to encrypt the content of the HDL source file.
- ▶ RSA-2048: asymmetric algorithm used to obfuscate a key used in HDL file encryption. The RSA is defined by the public-private key pair. You must be familiar with both keys in order to perform RSA decryption.

HDL File Encryption Flow

The current software version supports encryption of a single HDL source file per a single command.

The overall HDL file encryption flow is summarized in these steps:

- ▶ The source file of the IP design is AES encrypted using a symmetric session key. The AES encryption uses the CBC-128 or CBC-256 algorithm. In the source files, this section is referred to as a data block.
- ▶ The session key is RSA encrypted using the vendor's Public Key. In the source files, this section is referred to as a key block. Multiple key blocks may be present in the source file.
- ▶ The encrypted Session key and the encrypted design are merged to a file generally named the Encrypted RTL.

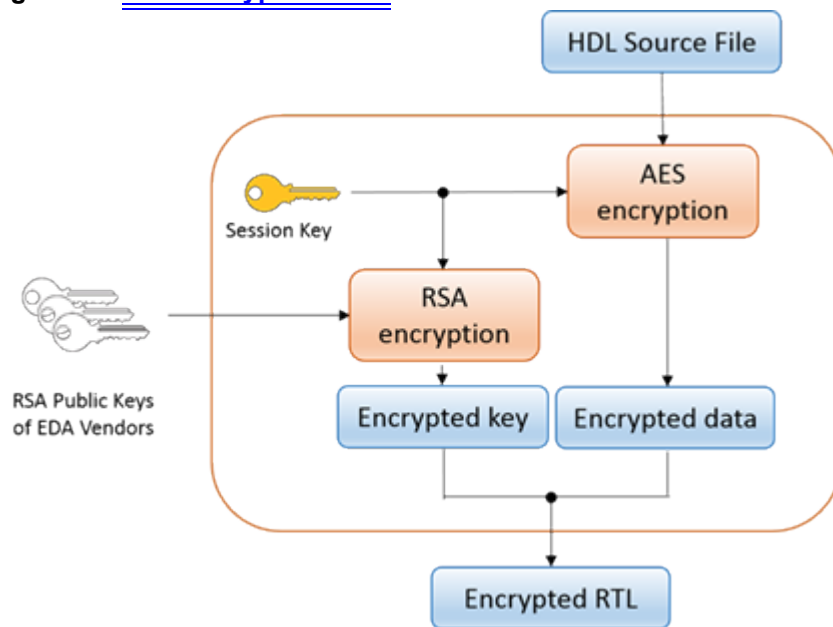
Each encrypted source file contains a single data block and one or more key blocks. The number of key blocks depends on the number of provided vendor's public keys.

Note

To decrypt an encrypted source file, you must perform the IP encryption flow steps in the reverse order.

During the next step in the design flow, typically synthesis, the Encrypted RTL is decrypted to access the original IP design, as shown in the following figure.

Figure 57: [HDL Encryption Flow](#)



By separating the encryption of data and key, you can use public keys from different vendors to encrypt the same HDL file.

For more information on how to perform HDL encryption, refer to [“Running HDL Encryption from the Command Line” on page 151](#).

HDL File Encryption Steps

This section provides step-by-step instructions on how to encrypt an HDL file.

The Radiant software provides the key templates you can simply drag-and-drop into an HDL file. Each key template is specific to an EDA vendor providing the value of a public key.

To view the templates in Project Navigator, go to the Source Template tool. Open the **Verilog > Encryption Templates** folder and select the EDA-specific key template.

Currently, the Radiant software supports these encryption templates:

- ▶ Lattice Semiconductor
- ▶ Synplicity-1
- ▶ Synplicity-2
- ▶ Mentor
- ▶ Synopsys
- ▶ Aldec
- ▶ Cadence
- ▶ Combined Sample: provides an example of file holding multiple keys.

Step 1: Prepare the HDL file.

Annotate the HDL source file with protected pragmas. Protected pragmas provide information regarding the type of the key used to encrypt the HDL file, the name of the key, and the encryption algorithm.

In this example, the HDL source file will be encrypted by the Lattice Public Key.

```
'pragma protect version=1
'pragma protect encoding=(enctype="base64")

// optional information
'pragma protect author="<Your_Name>"
'pragma protect author_info="<Your_Information>"
```

Step 2: Specify the portion of the HDL source file to encrypt.

Annotate the HDL file to specify the encryption. Only the portion defined within these protected pragmas is encrypted.

```
`pragma protect begin
// HDL portion that should be encrypted
`pragma protect end
```

Step 3: Prepare key.

Define the key with which the HDL file should be encrypted. Each key definition must contain the following information:

- ▶ **key_keyowner:** specify the owner of the key
- ▶ **key_keyname:** specify the name of the key. Same owner may provide multiple keys.
- ▶ **key_keymethod:** specify the used cryptographic algorithm. Current version supports RSA algorithm.
- ▶ **key_public_key:** specify the exact value of the key.

The key definition can be done in two ways:

Defining the key in the key.txt file: The public encryption key or keys can be defined in any .txt file. The key file may contain a single public key or a list of all available public keys. In the Radiant software, all common EDA vendor public keys are located in *<Radiant_install_directory>/isfpfpga/data/key.txt* file.

The following is an example of Lattice Public Key defined in key.txt file:

```
`pragma protect key_keyowner= "Lattice Semiconductor"
`pragma protect key_keyname= "LSCC_RADIAN2"
`pragma protect key_method="rsa"
`pragma protect key_public_key
MIIBIjANBgkqhkiG9w0BAQEFAAOCAQ8AMIIBCgKCAQEA0EZKUUhbuB6vSsc7OhQJ
iNAWJR5unW/OWp/LFI71eAl3s9bOYE2OlKdxbai+ndIeo8xFt2btXetUzuR6Srvh
xR2Sj9BbWlQToo2u8JfzD3X7AmRv1wKRX8708DPo4LDHZMA3qh0kFDDWkp2Eausf
LzE2cVxgq7fy/bDhUeN8xKQCSKJ7aguG6kOI6ROozZ211jzDLUQzhm2qYF8SpU1o
tD8/uw53wLfSuhR3MBOB++xcn2imvSLqdgHWuhX6CtZIx5CD4y8inCbcLy/0Qrf6
sdTN5Sag2OZhjeNdzmqSWqhL2JTDw+Ou2fWzhEd0i/HN0y4NM6h9fNn8nqxRyE7
IwIDAQAB
```

Defining the key directly in the HDL file: You may define the Public Key directly in the HDL file. You may define one or more keys.

```
`pragma protect key_keyowner= "Lattice Semiconductor"
`pragma protect key_keyname= "LSCC_RADIAN2"
`pragma protect key_method="rsa"
`pragma protect key_public_key
MIIBIjANBgkqhkiG9w0BAQEFAAOCAQ8AMIIBCgKCAQEA0EZKUUhbuB6vSsc7OhQJ
iNAWJR5unW/OWp/LFI71eAl3s9bOYE2OlKdxbai+ndIeo8xFt2btXetUzuR6Srvh
xR2Sj9BbWlQToo2u8JfzD3X7AmRv1wKRX8708DPo4LDHZMA3qh0kFDDWkp2Eausf
LzE2cVxgq7fy/bDhUeN8xKQCSKJ7aguG6kOI6ROozZ211jzDLUQzhm2qYF8SpU1o
tD8/uw53wLfSuhR3MBOB++xcn2imvSLqdgHWuhX6CtZIx5CD4y8inCbcLy/0Qrf6
sdTN5Sag2OZhjeNdzmqSWqhL2JTDw+Ou2fWzhEd0i/HN0y4NM6h9fNn8nqxRyE7
IwIDAQAB
```

If the key is defined directly in the HDL file, you don't need to provide the `-k` option in the `encrypt_hdl` command.

Note

The key defined directly in an HDL source file has preference over the key defined in the `key.txt` file.

Step 4: Select the encryption algorithm for data encryption.

The Radiant software supports both a 128-bit and a 256-bit advanced encryption standard (AES) with CBC mode. Select the type of algorithm by defining one of the two options. The default is set to 256-bit AES with CBC mode.

```
`pragma protect data_method="aes128-cbc"
```

or

```
`pragma protect data_method="aes256-cbc"
```

Step 5: Run the `encrypt_hdl` Tcl command.

In the Tcl console window, type in the command to encrypt an HDL file. The option `-k` may or may not be used depending the location of the key file. The language selection (`-l`) and creation of new output file (`-o`) are optional. The default is Verilog. If you don't specify the output file, the tool generates a new output file named `<input_file_name>_enc.v`.

If the key was defined in the `key.txt` file: The command will encrypt the HDL file with all keys defined in the `key.txt` file.

```
encrypt_hdl -k <keyfile> -l <verilog | vhdl> -o <output_file>
<input_file>
```

If the key was defined directly in the HDL file:

```
encrypt_hdl -l <verilog | vhdl> -o <output_file> <input_file>
```

The encrypted file is located at the path specified in the `encrypt_hdl` command.

Step 6: Activate the encrypted HDL source file in the project file.

In the File List view, add the generated file to the project. Right-click on the encrypted file and choose **Include in Implementation**.

To see an example of a Verilog or VHDL pragma-annotated HDL source file, see [“Defining Pragmas” on page 153](#).

Implementation Flow and Tasks

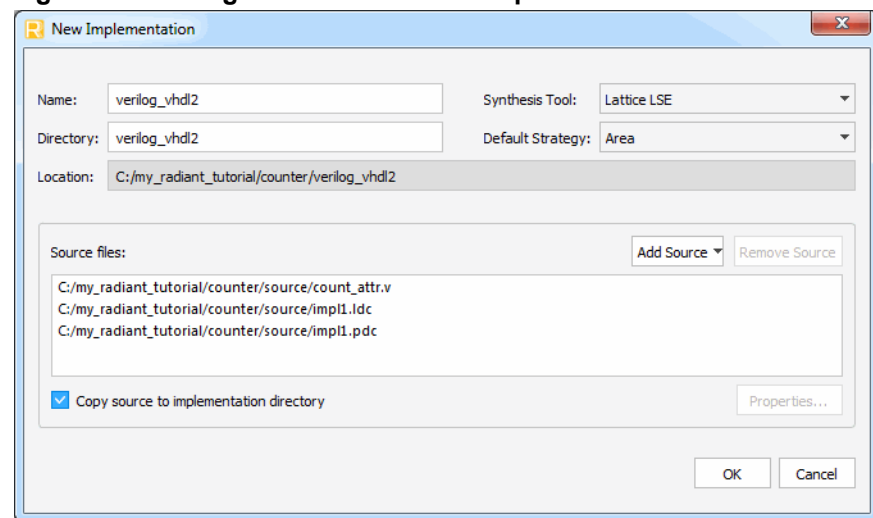
Implementations organize the structure of your design and allow you to try alternate structures and tool settings to determine which ones will give you the best results.

To help determine which scenario best meets your project goals, use a different implementation of a design using the same tool strategy, or run the same implementation with different strategies. Each implementation has an associated active strategy, and when you create a new implementation you must select its active strategy.

To try a new implementation with different strategies, you must create a new implementation/strategy combination.

1. Right-click the project name in File List.
2. Choose **Add > New Implementation**.
3. In the dialog box, choose a source file from an existing implementation using the Add Source drop-down menu.
4. Choose a strategy using the Default Strategy drop-down menu.

Figure 58: Adding a Source to a New Implementation



To use the same source for new and existing implementations, make sure that the “Copy source to implementation directory” option is not selected. This will ensure that your source is kept in sync between the two implementations.

Synthesis Constraint Creation

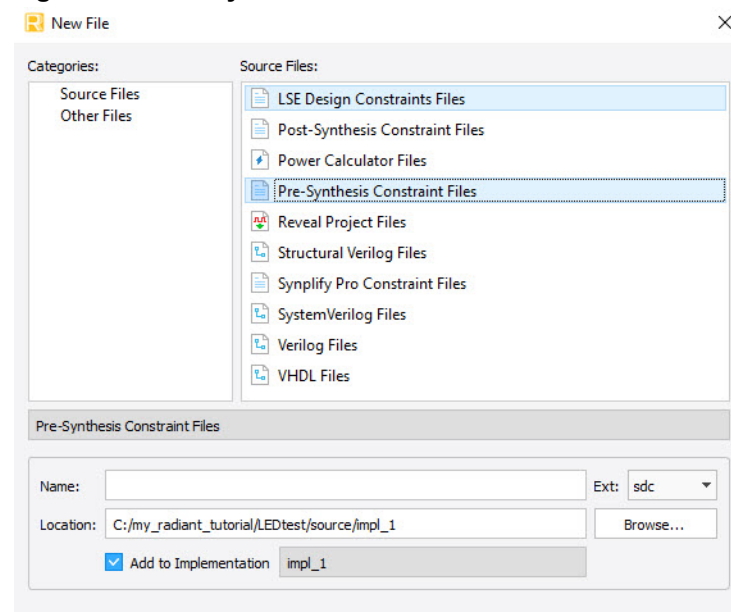
Synthesis constraints can be added to a design implementation in the format of the Synopsys Design Constraint language, while constraints can be added in the Synopsys standard timing constraints format in the form of FPGA Design Constraint (LDC, PDC, or FDC format) files.

If you are using the Lattice Synthesis Engine, the synthesis constraints will be included in an .ldc file. If you are using Synplify Pro for synthesis, the constraints will be included in an .fdc file. The older .sdc file format is also supported for constraints.

To create a new synthesis constraint file, right-click the Synthesis Constraint Files folder in the File List pane and choose **Add > New File**. In the New File dialog box, select one of the following and give the file a name:

- ▶ Pre-Synthesis Constraint Files (.ldc)
- ▶ Post-Synthesis Constraint Files (.pdc)
- ▶ Synplify Pro Constraint Files (.fdc)

Figure 59: New Synthesis Constraint Files



The .ldc, .pdc, or .fdc file will open in the Source Editor to allow you to manually add the constraints. You can use the Pre-Synthesis Constraint Editor tool to add pre-synthesis constraints to the .ldc file and the Post-Synthesis Timing Constraint Editor tool to add logic view level post-synthesis timing constraints to .pdc files. You can also use the Device Constraint Editor and Floorplan View to add physical constraints to .pdc files. For detailed information about setting constraints, see *Applying Design Constraints* and the *Constraints Reference Guide* in the Radiant Software Help.

An alternative way of adding constraint files is through Source Template. To view a constraint template, click on the Source Template tab on the left-hand side of the Project Navigator pane. If not selected, make sure it is enabled in **View > Show Views > Source Template**. The list of constraint templates includes the timing constraints, physical constraints, and user templates. Select a template and copy and paste it into your active design.

Constraint Creation

LDC (pre-synthesis) and PDC (post-synthesis) files are used to input timing and physical constraints. The following steps illustrate how to assign and edit constraints in the Radiant software and implement them at each stage of the design flow.

1. If desired, define some constraints at the HDL level using HDL attributes. These source file attributes are included in the Unified Database (UDB), and will be displayed in the Radiant software after the Map Design process is run. The following is an example of applying the LOC attribute in Verilog source code:

```
module top (
    input clk1,
    input datain /* synthesis loc = B12 */,
    output ff_clk1out
)
```

For more information on HDL Attributes, see the topic “HDL Attributes” in the *Constraints Reference Guide* section of the Radiant Software Help.

2. Open one or more of the following tools to create new constraints or to modify existing constraints from the source files:
 - ▶ Device Constraint Editor, which consists of:
 - ▶ Spreadsheet View – the primary view for setting constraints.
 - ▶ Package View – examines the pin layout of the design, modifies signal assignments, reserves pin sites that should be excluded from placement and routing, and runs PIO design rule check to verify legal placement of signals to pins.
 - ▶ Device View – examines FPGA device resources.
 - ▶ Netlist View - shows port types (Input, Output) and groups.
 - ▶ Timing Constraint Editor. Timing/Physical constraints are entered through:
 - ▶ Pre-Synthesis Constraint Editor - used to enter pre-synthesis constraints such as clocks, clock latency/uncertainty/Group, Input/Output delays, timing exceptions, and attributes.
 - ▶ Post-Synthesis Timing Constraint Editor - This post-synthesis version of the Timing Constraint Editor is used to enter logic view level timing constraints and physical constraints.
 - ▶ Floorplan View examines the device layout of the design, draws bounding boxes for GROUPs, draws REGIONs for the assignment of groups or to reserve an area, and reserves sites and REGIONs that should be excluded from placement.
3. Save the constraints to the pre-synthesis constraint file (.ldc) or post-synthesis constraint file (.pdc).
4. Run the Map Design process (Map).
5. Run the Map Timing Analysis process and examine the timing analysis report. This is an optional step, but it can be a quick and useful way to

identify serious timing issues in the design and constraint errors (syntax and semantic). Modify constraints as needed and save them.

6. Run the Place & Route Design process.
7. Open views directly or by cross-probing to examine timing and placement and create new GROUPs. Also examine the Place & Route Timing report.
8. Modify constraints or create new ones using the appropriate constraint tool. Save the constraint changes and rerun the Place & Route Design process.

Simulation Flow

The simulation flow in the Radiant software supports source files that can be set in the File List view to be used for the following purposes:

- ▶ Simulation and Synthesis (default)
- ▶ Simulation
- ▶ Synthesis

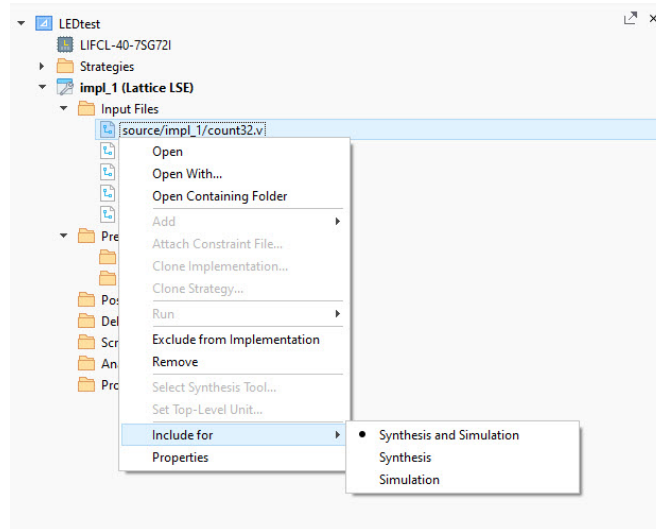
This allows the use of test benches, including multiple file test benches. Additionally, multiple representations of the same module can be supported, such as one for simulation only and one for synthesis only.

You can add top level signals to the waveform display in the simulator and to automatically start the simulator running.

The Simulation Wizard automatically includes any files that have been set for simulation only or for both simulation and synthesis. You can select the top of the design for simulation independent of the implementation design top. This allows easy support for test bench files, which are normally at the top of the design for simulation but not included for implementation. The implementation wizard exports the design top to the simulator, along with source files, and set the correct top for the .spf file if running timing simulation.

After you add a module, use the **Include for** menu to specify how the module file is to be used in the design.

Figure 60: “Include For” Input Files



Simulation Wizard Flow

When you are ready to simulate, export the design using the Simulation Wizard. Aside from the RTL simulation, you can perform the Post-Synthesis simulation and Post-Place & Route back annotated netlist simulation.

Post-Synthesis Simulation File generates a Verilog netlist (_syn.vo) file. Similarly, the Gate-Level Simulation File generates a Verilog netlist of the routed design (.vo file) that is back annotated with timing information (.sdf file). The generated file enables you to run a timing simulation of your design. For more details on how to generate these files, see [“Task Detail View” on page 31](#).

1. In the Radiant software, click **Tools > Simulation Wizard** or click the  icon in the toolbar. The Simulation Wizard opens.
2. In the Preparing the Simulator Interface page click **Next**.
3. In the Project box, enter the name of your project in the Project Name text box. In the Project Location box browse to the file path location where you want to save your simulation project.

When you designate a project name in this wizard page, a corresponding folder will be created in the file path you choose. If you add a subdirectory to the default file path the wizard will prompt you to create a new directory. Click **Yes** in the popup dialog box that appears.

- In the Simulator box, click either the **ModelSim** or **Active-HDL** simulator check box.

Notes

- ▶ If Aldec-HDL is not installed, it will not be selectable.
- ▶ If you change simulation tool, project name will not change. Any previous Simulation Wizard results will be deleted, and a new simulation will be started using new simulation files.

- In the Process Stage box, choose what type of Process Stage of simulation project you wish to create. Valid types are **RTL**, **Post-Synthesis**, **Post-Route Gate-Level**, and **Post-Route Gate-Level+Timing**. Only those process stages that are available are activated. Click **Next**.

Note

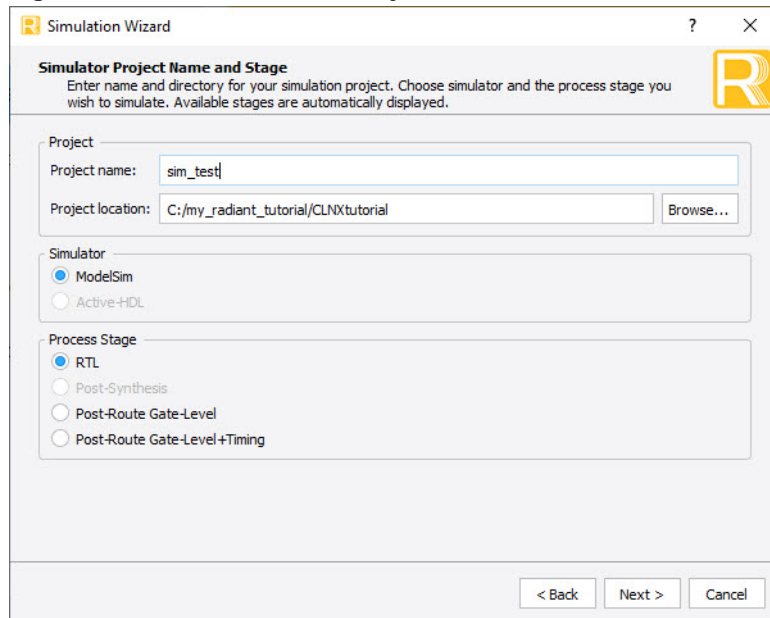
If you want to run a post-synthesis simulation, you must first generate a post-synthesis simulation file (*<file_name>_syn.vo*) as follows:

- ▶ In the Radiant Software, click Task Detail View  and check **Post-Synthesis Simulation File**.
- ▶ In the Process Toolbar, click **Synthesize Design**.

If you want to run a post-route gate-level+timing simulation, you must first generate a post-routed, back-annotated simulation file (*<file_name>_vo.vo*) and a timing constraint file (*<file_name>_vo.sdf*) as follows:

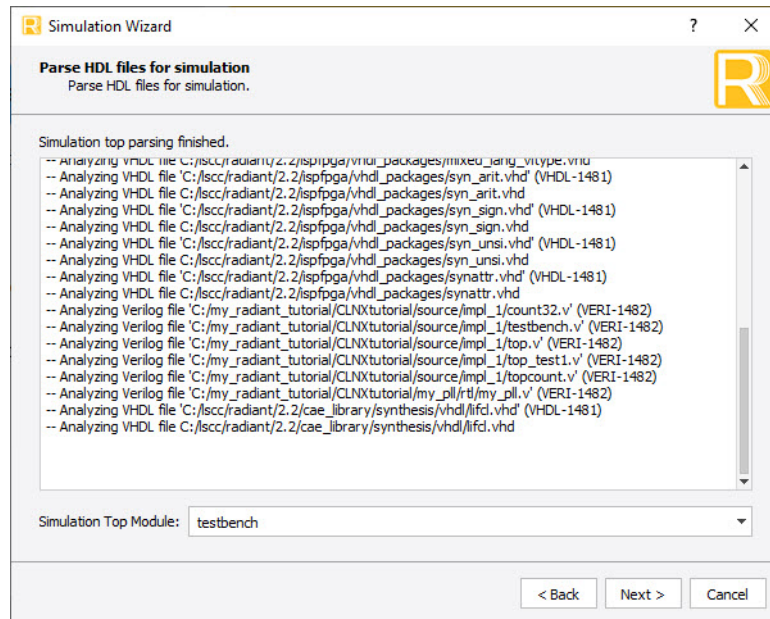
- ▶ In the Process Toolbar, click **Export Files**. Ensure that **Gate-Level Simulation File** is selected in the Task Detail View .

Figure 61: New Simulation Project



6. In the Add and Reorder Source page, select from the source files listed in the Source Files list box or use the browse button to choose a source file not listed.
 - ▶ By default, the **Automatically Set Simulation Compilation File Order** option is checked.
7. Click **Next**. If the **Automatically Set Simulation Compilation File Order** option was checked in the previous step, you'll need to specify a top level simulation test bench in the **Simulation Top Module** box in the Parse HDL Files for Simulation box.

Figure 62: Simulation Top Module



8. Click **Next**. A Summary page appears that provides information on the project selections, including the simulation libraries. By default the **Run simulator**, **Add top-level signals to waveform display**, and **Run simulation** check boxes are enabled. Their functionality is as follows:
 - ▶ The **Run simulator** option launches the simulation tool you chose earlier in the wizard.
 - ▶ The **Add top-level signals to waveform display** option can only be selected if the **Run simulator** box is checked. This option adds top-level signals to the waveform in the simulation tool.
 - ▶ The **Run simulation** option can only be selected if the **Add top-level signals to waveform display** option is checked. This option causes your simulation tool to run the simulation automatically.

If the **Run simulation** option is selected, you can specify the simulation run length. The default run length is set to 100, measured in nanoseconds (ns). You can manually enter an integer or fractional value in the box. A value of 0 will cause the simulator to run the

current simulation continuously, or until it hits a breakpoint or specified break event. You can specify a different time unit in the dropdown. Available time units are fs, ps, ns, us, ms, or sec.

9. Click **Finish**. After the simulation is complete, the Simulation Wizard Project (.spf) file and a simulation script DO file are generated and the simulation is displayed. If you are using Active-HDL, the wizard will generate an .ado file and if you are using ModelSim/Questa, it creates an .mdo file as well.

In some designs, the compile order of the HDL files passed to the simulator might result in compilation warnings. In most cases, these compilation warnings can be safely ignored. The warnings can be eliminated in one of two ways:

- ▶ The correct compilation order for the HDL files can be set manually in the File List view. After the correct order is determined, the files are sent to the simulator, which will eliminate any compilation warnings.
- ▶ The correct compilation order for the HDL files can be set in the Simulation Wizard during the “Add and Reorder” step. After the correct order for the files is set manually, the files will be sent to the simulator, which will eliminate any compilation warnings.

You can also run the simulation directly from the wizard.

Chapter 7

Working with Tools and Views

This chapter covers the tools and views controlled from the Radiant software. Tool descriptions are included and common tasks are described.

Overview

The Radiant design environment streamlines the implementation process for FPGAs by combining the tool and data views into one common location. Two main features of this design environment make it easy to keep track of unsaved changes in your design and examine data objects in different views.

Shared Memory The Radiant software uses shared memory that is accessed by all tools and views. As soon as design data has been changed, an asterisk * appears in the tab of the open views, notifying you that unsaved changes are in memory.

Cross-Probing Shared design data in the Radiant software enables you to select a data object in one view and display it in another. This cross-probing capability is especially useful for displaying the physical location of a component or net after it has been implemented. You can click on a hyperlink icon to cross-probe into the specific tool. The Radiant software supports:

- ▶ Post-Synthesis timing report links to Netlist Analyzer
- ▶ Map & PAR timing reports link to Physical Designer's Placement and Routing views

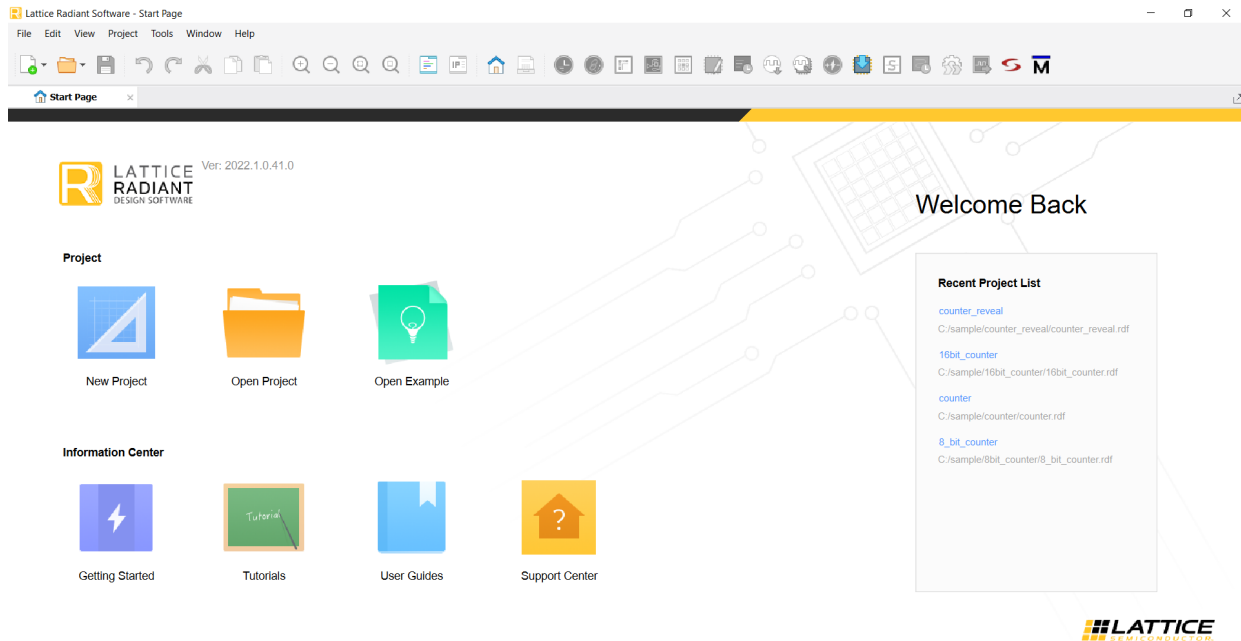
View Menu Highlights

The View menu and toolbar control the display of all toolbars, project views, and display control. Also included in the View menu are the important project-level features: Start Page and Reports.

Start Page

The Start Page is displayed by default when you run the Radiant software. The Start Page enables you to open projects, read product documentation, and view the software version and updates. You can modify startup behavior by choosing **Tools > Options > General > Startup**.

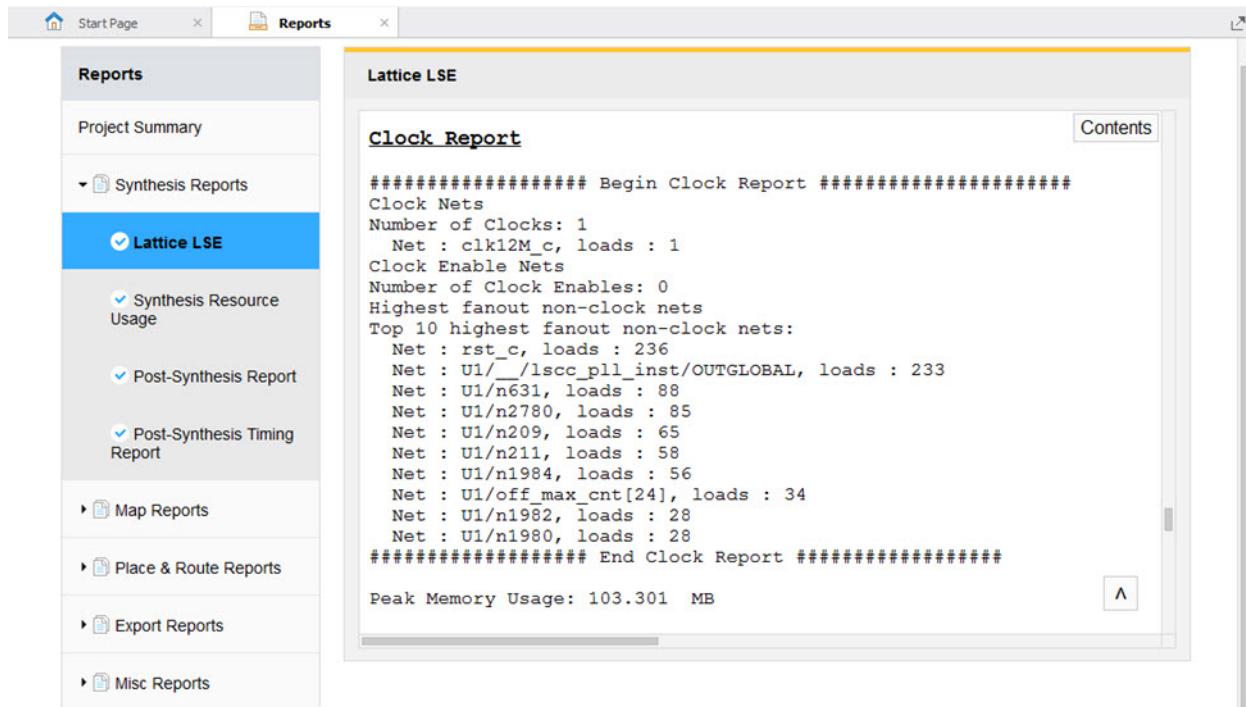
Figure 63: Default Start Page



The Start Page gives you quick access to recent projects and to product documentation.

Reports

The Reports view provides one central location for the project summary and design processing reports. It is displayed by default when a project is open. Alternatively, click on the Reports  icon in the toolbar.

Figure 64: Reports View Showing Last Process Run

The Reports view is organized into Project Summary, Synthesis Reports, Map Reports, Place & Route Reports, Export Reports, and Misc Reports.

The different file icons indicate:

- ▶ A report has been completed  (blue check mark).
- ▶ A report has never been generated  (gray circle).
- ▶ A report is out of date  (orange question mark).

Select any item to view its report.

The Reports view also supports path cross-probing through the timing or analysis reports. You can view a specific clock or data path in Netlist Analyzer and Physical Designer by clicking the icon next to the path.

Figure 65: Cross-Probing

The screenshot shows the 'Place & Route Timing Analysis' report. On the left, a sidebar lists various reports, with 'Place & Route Timing Analysis' selected. A red arrow points from the 'Source Clock Path' section of the report to the 'Physical View' of the design, illustrating the cross-probing functionality.

Source Clock Path

Shown in: Netlist Analyzer Floor Planner Physical View

Name	Cell/Site Name	Delay Name
OSCInst0/CLKHF	HFOSC_HFOSC_R1C32	CLOCK LATENCY
oclk		NET DELAY

Data path

Shown in: Netlist Analyzer Floor Planner Physical View

Name	Cell/Site Name	Delay Name
{count_15__i19/CK count_15__i20/CK}->count_15__i20/Q	SLICE_R8C5C	CLK_TO_Q1_DELAY
count[20]		NET DELAY
i15_4_lut_adj_7/A->i15_4_lut_adj_7/Z	SLICE_R9C3B	A0_TO_F0_DELAY
n39_adj_5		NET DELAY
i22_4_lut_adj_5/D->i22_4_lut_adj_5/Z	SLICE_R9C5A	D1_TO_F1_DELAY
n46_adj_1		NET DELAY
i103_4_lut/B->i103_4_lut/Z	SLICE_R9C5B	B1_TO_F1_DELAY
n158		NET DELAY
i53_4_lut/B->i53_4_lut/Z	SLICE_R9C6A	B1_TO_F1_DELAY
n159		NET DELAY

To learn more about cross-probing, view [“Cross-Probing”](#) on page 37.

Tools

The entire FPGA implementation process tool set is contained in the Radiant software. You can run a tool by selecting it from the Tools menu or toolbar.

This section provides an overview of each of these tools. More detailed information is available in their respective user guides, which you can access from the Start Page or from the Radiant Software Help. Detailed descriptions of external tools can be found in their product documentation as well.

If you are viewing an encrypted design, some secured objects may not be visible in the selected tool. To learn more, see [“Secure Objects in the Design”](#) in the Help.

Timing Constraint Editor

The Timing Constraint Editor is used to edit pre-synthesis (.ldc) constraints and post-synthesis constraints stored in a .pdc file. The Timing Constraint Editor consists of two tools:

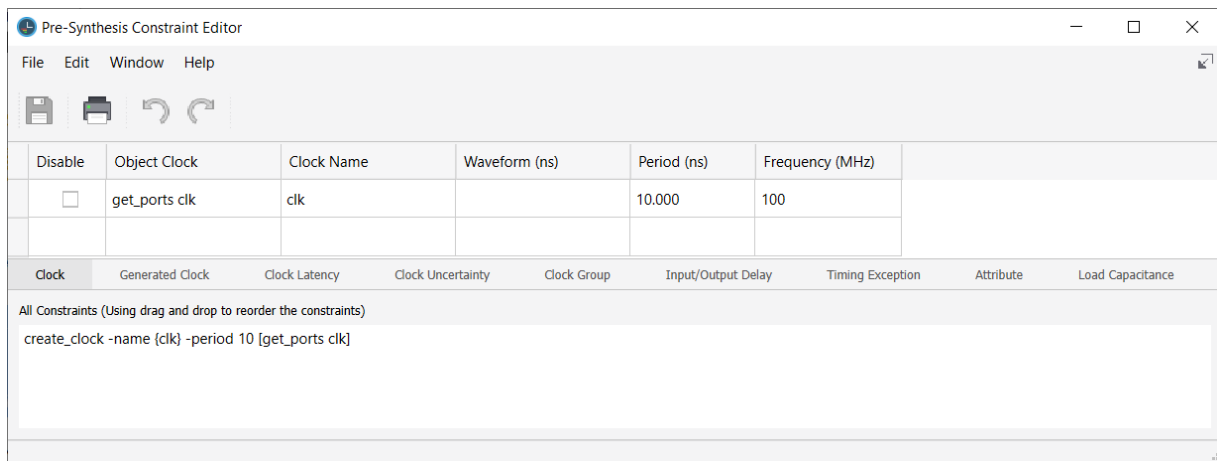
- ▶ Pre-Synthesis Constraint Editor
- ▶ Post-Synthesis Timing Constraint Editor

Both tools have identical interfaces and the entry mechanisms of the constraints are also the same. The key differences are that pre-synthesis constraints are entered pre-synthesis and are synthesized by the chosen synthesis tool. The post-synthesis constraints are already synthesized and populated in each of the different constraints tabs. You cannot modify the post-synthesis constraints that were populated from pre-synthesis, but can supply new constraints (physical and timing) to either override the existing one already populated or supply new constraints to better constrain the design for improved performance upon analysis.

The different constraint types are entered through these tabs:

- ▶ Clock - used to define the clocking scheme of the design.
- ▶ Generated Clock - used to define generated clocks such as from PLLs.
- ▶ Clock Latency - is the delay between the clock source and clock pin. Used to define the latency in terms of rise and fall times.
- ▶ Clock Uncertainty - is the jitter difference of two signals, possibly caused by clocks. Used to define the amount of uncertainty of a clock or during clock domain transfer.
- ▶ Clock Group - used to specify which clocks are not related in terms of being logically/physically exclusive and asynchronous.
- ▶ Load Capacitance - used to define the load capacitance of ports.
- ▶ Input/Output Delay - used for setting input and output delays.
- ▶ Timing Exception - used to set min/max, false, and multicycle path constraints.
- ▶ Attribute - used to set synthesis attributes.

Figure 66: Pre-Synthesis Constraint Editor



Unified Constraints Flow

The goal of the unified constraints flow is to have one constraint file that can be used by both the synthesis tools (LSE & Synplify Pro). From Radiant 3.0 onwards, pre-synthesis TCE can be launched with Synplify Pro, and constraints can be saved into an .sdc file, which can be used by both LSE and Synplify Pro, along with .ldc (for LSE) and .fdc (Synplify Pro).

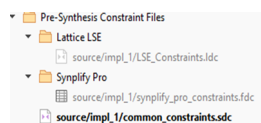
Table 1: Supported Constraints File Type

Constraint Type	Synthesis Tool	TCE Support
.ldc	LSE	Yes
.sdc	LSE/Synplify Pro	Yes
.fdc	Synplify Pro	No

Table 2: Difference between Radiant 3.0 (or later) and previous versions

	Radiant 1.x/2.x	Radiant 3.0 or later
.sdc file	For Synplify Pro only	For both LSE and Synplify Pro

The following figure shows the Radiant folder structure. The .ldc constraints are stored under the Lattice LSE folder, the .fdc constraints are stored under Synplify Pro, and the newly created .sdc constraints are stored outside these two folders, which can be commonly used by both synthesis tools.



Notes:

1. If the previous project includes an .sdc file, Radiant 3.0 or later will use it in the LSE flow, but it can cause unexpected syntax errors.
2. If the previous project includes both .ldc and .sdc files, Radiant 3.0 or later will force one of them to be inactive. Only one constraint file can be active. The Timing Constraint Editor can only be launched with the active constraints file.

Known Limitations

- ▶ Constraints on internal objects from Synplify Pro may not be fully compatible. The pre-synthesis TCE uses Verific parser for compilation whereas Synplify Pro uses their own parser. This may cause some name changes on the internal objects when Synplify Pro is used.
- ▶ Pre-Synthesis TCE "Attribute" tab is only supported by LSE. The attributes are passed using a Lattice Proprietary constraint "Idc_define_attribute," which is not supported by Synplify Pro.

See Also

- ▶ [“Constraints Reference Guide” on page 576](#)
- ▶ [“Using Radiant Software Pre-Synthesis Constraints” on page 170](#)
- ▶ [“Device Constraint Editor” on page 99](#)

Constraint Propagation

Constraint propagation is not a stand-alone tool, nor do you have to provide any input to take advantage of this feature.

You usually define constraints for the top-level design as well as include other constraint files for any IP that are generated with Radiant tools such as IP Catalog. Constraints defined at the IP level may not contain the correct hierarchical names and so will not be applied correctly when synthesized. To help honor as much of the supplied constraints as possible, this feature runs a design rule check (DRC) on all the input constraints and writes out a new constraint file to support hierarchical constraints such as soft-IP constraints and honor top-level constraints.

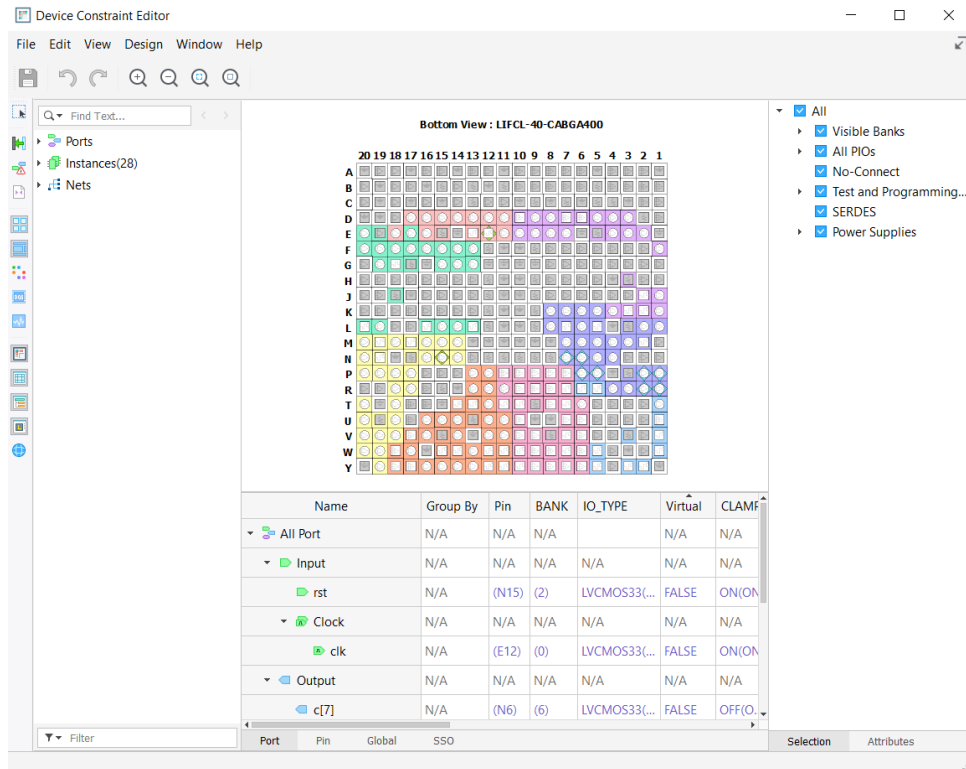
For more information about Constraint Propagation, refer to *Constraint Propagation* in the Radiant Software Help.

Device Constraint Editor

The Device Constraint Editor is used to edit post-synthesis constraints. These constraint editing views are available from the Radiant toolbar and Tools menu.

All modified constraints are saved to a .pdc file and the flow returned to Map. The Device Constraint Editor shows the pin layout of the device and displays the assignments of signals to device pins. This view allows you to edit these assignments, and reserve sites on the layout to exclude from placement and routing. The Device Constraint Editor is also the entry mechanism for physical constraints.

Device Constraint Editor views, shown in Figure 67, enable you to develop constraints that will shorten turn-around time and achieve a design that conforms to critical circuit performance requirements.

Figure 67: Device Constraint Editor

Global Settings

In the Device Constraint Editor tool at the bottom of the tool, there are a series of tabs to allow you to set many of the device settings such as Junction Temperature, Voltage, sysCONFIG, User Code, Derating, Bank VCCIO, Global Set/Reset, Use Primary Net, and Vref Locate constraints.

For more information on how to do this, in addition to detailed information about Device Constraint Editor, see *Setting Global Settings* in the Radiant Software Help.

sysCONFIG Settings

Defines system configuration option settings for the sysCONFIG feature. If you do not specify these settings in the .pdc file, using the Global tab in Device Constraint Editor or manually, some default sysCONFIG constraints will automatically be generated based on the device selection.

The SYSCONFIG constraint is available for all Lattice FPGA devices that support the sysCONFIG configuration port. The sysCONFIG port can be a single data line or byte-wide (multiple byte) data port that can support serial and parallel configurations streams. The devices also support daisy chaining.

To set sysCONFIG parameter values:

1. After opening the design project, choose **Tools** >  **Device Constraint Editor**.
2. Select the Global tab at the bottom of the view.
3. In the sysCONFIG drop down setting, for each parameter in the value column, double click to bring up the drop down to select a value or, type in the actual value.

SYSCONFIG Keyword Settings

The following table shows the default settings in bold type and the selectable settings for all of the keyword values for the SYSCONFIG constraint. They are ordered as they appear in the Global tab in the Device Constraint Editor.

Device Support LAV-AT-E, LFCPNX, LFD2NX, LFMXO5, LIFCL, UT24C, UT24CP

Syntax

```
ldc_set_sysconfig <keyword>=<value>+
```

You should set the constraint configuration using the Device Constraint Editor or by editing the .pdc file directly. If you do not set sysCONFIG options, default SYSCONFIG constraints are automatically set.

Examples

The SYSCONFIG constraint allows you to set a series of keyword values in succession after the ldc_set_sysconfig tcl command.

Figure 68: Example of a SYSCONFIG in the .pdc constraint file

```
ldc_set_sysconfig {JTAG_PORT=ENABLE PROGRAMN_PORT=ENABLE
MCCLK_FREQ=56.2}
```

Table 3: sysCONFIG Values

Keyword	Default and Selectable Values	Supported Devices
SLAVE_SPI_PORT	DISABLE [disable, serial, dual, quad]	LAV-AT-E, LFCPNX, LFD2NX, LFMXO5, LIFCL, UT24C, UT24CP
MASTER_SPI_PORT	DISABLE [disable, serial, dual, quad]	LAV-AT-E, LFCPNX, LFD2NX, LFMXO5, LIFCL, UT24C, UT24CP
SLAVE_I2C_PORT	DISABLE [disable, enable]	LFCPNX, LFD2NX, LFMXO5, LIFCL, UT24C, UT24CP

Keyword	Default and Selectable Values	Supported Devices
SLAVE_I3C_PORT	DISABLE [disable, enable]	LFCPNX, LFD2NX, LFMXO5, LIFCL, UT24C, UT24CP
JTAG_PORT	ENABLE [enable, disable]	LFCPNX, LFD2NX, LFMXO5, LIFCL, UT24C, UT24CP
DONE_PORT	ENABLE [enable, disable]	LFCPNX, LFD2NX, LFMXO5, LIFCL, UT24C, UT24CP
INITN_PORT	ENABLE [enable, disable]	LFCPNX, LFD2NX, LFMXO5, LIFCL, UT24C, UT24CP
PROGRAMN_PORT	ENABLE [enable, disable]	LFCPNX, LFD2NX, LFMXO5, LIFCL, UT24C, UT24CP
BACKGROUND_RECONFIG	OFF [off, on, sram_ebr, sram_only]	LAV-AT-E, LFCPNX, LFD2NX, LFMXO5, LIFCL, UT24C, UT24CP
DONE_EX	OFF [off, on]	LFCPNX, LFD2NX, LFMXO5, LIFCL, UT24C, UT24CP
DONE_OD	ON [off, on]	LFCPNX, LFD2NX, LFMXO5, LIFCL, UT24C, UT24CP
MCCLK_FREQ	See MCCLK section below	LAV-AT-E, LFCPNX, LFD2NX, LFMXO5, LIFCL, UT24C, UT24CP
TRANSFR	OFF [off] for LFD2NX, LIFCL, and UT24C ON [off, on] for LFCPNX and UT24CP	LFCPNX, LFD2NX, LFMXO5, LIFCL, UT24C, UT24CP
CONFIG_IOSLEW	SLOW [slow, medium, fast]	LFCPNX, LFD2NX, LFMXO5, LIFCL, UT24C, UT24CP
CONFIGIO_VOLTAGE_BANK0/1	NOT_SPECIFIED [3.3, 2.5, 1.8, 1.5, 1.2]	LAV-AT-E, LFCPNX, LFD2NX, LFMXO5, LIFCL, UT24C, UT24CP
CONFIG_SECURE	OFF [off, on]	LFCPNX, LFD2NX, LFMXO5, LIFCL, UT24C, UT24CP
WAKE_UP	ENABLE [disable] _DONE_SYNC DONE_EX=OFF [on]	LFCPNX, LFD2NX, LFMXO5, LIFCL, UT24C, UT24CP
COMPRESS_CONFIG	OFF [off, on]	LFCPNX, LFD2NX, LFMXO5, LIFCL, UT24C, UT24CP

Keyword	Default and Selectable Values	Supported Devices
EARLY_IO_RELEASE	OFF [off, on]	LFCPNX, LFD2NX, LFMXO5, LIFCL, UT24C, UT24CP
BOOTMODE	DUAL [dual, single, none]	LFCPNX, LFD2NX, LFMXO5, LIFCL, UT24C, UT24CP
MASTER_PREAMBLE_TIMER_CYCLES	600000	LFCPNX, LFD2NX, LFMXO5, LIFCL, UT24C, UT24CP

Table 4: Avant sysCONFIG Values

Keyword	Default and Selectable Values	Supported Devices
USERCODE Format	BINARY [HEX, ASCII, AUTO]	LAV-AT-E
USERCODE	32'b00000000000000000000000000000000 [32'b11111111111111111111111111111111]	LAV-AT-E
BITSTREAM_REV Format	BINARY [HEX, ASCII, TIMESTAMP]	LAV-AT-E
BITSTREAM_REV	32'b00000000000000000000000000000000 [32'b11111111111111111111111111111111]	LAV-AT-E
UNIQUE_ID	8'b00000000 [8'b11111111]	LAV-AT-E
BOOT_SEL	DUAL [SINGLE]	LAV-AT-E
PROGRAMN_RECOVERY	DISABLE	LAV-AT-E
MSPI_RESET	DISABLE	LAV-AT-E
MULTI_BOOT_MODE	DISABLE	LAV-AT-E
DAISY_CHAIN	DISABLE	LAV-AT-E
DAISY_CHAIN_WAIT_DONE	DISABLE	LAV-AT-E
MSPI_SIGNATURE_TIMER	200MS [100MS, 50MS, 40MS, 20MS, 1MS, 500US, 100US, INFINITE]	LAV-AT-E
MSPI_CLK_PHASE	RX_RISING_TX_FALLING	LAV-AT-E
MSPI_CPOL	IDLE_CLOCK_LOW	LAV-AT-E
MSPI_SHIFT_ORDER	MSB_FIRST	LAV-AT-E
SSPI_CLK_PHASE	RX_RISING_TX_FALLING	LAV-AT-E
SSPI_CPOL	IDLE_CLOCK_LOW	LAV-AT-E
SSPI_SHIFT_ORDER	MSB_FIRST	LAV-AT-E
SSPI_DAIYN_CHAIN_MODE	DISABLE [SCM, AUTO]	LAV-AT-E

DONE_PORT =ENABLE (default) | DISABLE

When set to ENABLE, the pin is not available in user mode.

INITN_PORT =ENABLE (default) | DISABLE

When set to ENABLE, the pin is not available in user mode.

PROGRAMN_PORT =ENABLE (default) | DISABLE

When set to ENABLE, the pin is not available in user mode.

BACKGROUND_RECONFIG =OFF (default) | ON | SRAM_EBR | RAM_ONLY

ON turns on background reconfig feature or can be done in embedded blockram or SRAM.

DONE_EX = OFF (default) | ON

The customer can select if the device should wake up on its own after the DONE bit is set or wait for an external DONE signal to drive the DONE pin high. The DONE_EX attribute will determine if the wake up sequence is driven by an external DONE signal. If set to ON, the user wishes to delay wake up until the DONE pin is driven high by an external signal and synchronous to the clock. The user will select OFF if they want to synchronously wake up when the DONE bit is set and ignore any external driving of the DONE pin.

DONE_OD = ON (default) | OFF

Done Pin Open Drain. Enables you to configure the DONE pin as an open drain pin. By default, the pullup on the DONE pin is active.

The DONE pin used in sysCONFIG to indicate that configuration is done. The DONE pin can be linked to other DONE pins and used to initiate the wake up process. The DONE pin can be open collector or active drive. Default is ON and insures the user needs to make a conscious decision to drive the pin.

MCCLK_FREQ = <frequency_value> (MHz)

Controls the Master Clock frequency. When a master configuration mode is used, MCLK will become an output clock with the frequency set by the user. Until the device is configured, the default Master Clock frequency is the lowest frequency support by the device. For LFCPNX, LFD2NX, LFMXO5, LIFCL, UT24C, and UT24CP a 450 Mhz oscillator is divided and made available for configuration.

Valid options are 3.5 MHz, 7.0 MHz, 14.1 MHz, 28.1 MHz, 56.2 MHz, 90.0 MHz, 112.5 MHz, and 150.0 MHz. The default is the lowest frequency, 3.5 MHz.

TRANSFR =OFF (default) for LFD2NX, LIFCL, and UT24C. ON [default] for LFCPNX and UT24CP

For the current supported devices, only OFF is available. When ON is selected for a future supported device, it sets the CR0 TransFR option. This will also enable the Hold IO type.

CONFIG_IOSLEW = SLOW (default) | MEDIUM | FAST

Controls config output pin slew rate. Sets CR0[7:6].

CONFIGIO_VOLTAGE_BANK0/1 = NOT_SPECIFIED (default) | 2.5 | 1.2 | 1.5 | 1.8 | 3.3

Setting this attribute informs the software which voltage is required in Bank0/1 to satisfy the user's sysCONFIG requirements. The sysCONFIG pins used for configuration may or may not be used in the design and hence the user shall be able to declare the voltage interface for this bank. DRC errors can then be generated based on CONFIGIO_VOLTAGE and usage of the dual-purpose sysCONFIG pins. BANK0 and 1 values can be different and a combination of the available values including NOT_SPECIFIED.

CONFIGIO_VOLTAGE_BANK1/2 NOT_SPECIFIED (default) | 2.5 | 1.2 | 1.8 | 3.3

The CONFIGIO_VOLTAGE_BANK0/1 attribute supports all devices except LAV-AT-E and ICE40UP.

The CONFIGIO_VOLTAGE_BANK1/2 attribute only supports the LAV-AT-E device

CONFIG_SECURE = OFF (default) | ON

Configuration Security. When set to ON, no readback operation is supported through the sysCONFIG port or ispJTAG port of the general contents. The USERCODE area is readable and not considered securable. The OFF setting indicates that readback is enabled through any port. Prevents readback of the configuration memory contents.

WAKE_UP = ENABLE/DISABLE_DONE_SYNC (DONE_EX = OFF) / (DONE_EX = ON)

Wake Up. Wake Up is a controlled event after a part has been configured. External control of the DONE pin can be selected to either delay wake up or be ignored. See [DONE_EX](#).

The Wake Up sequence controls three internal signals, and the DONE Pin will be driven after configuration and prior to user mode. If DONE_EX = ON, the WAKE_UP keyword will take your selected option (1-7), and then the software will set wake-up signal bits accordingly. If you do not select a wake-up sequence, the default wake-up sequence will be 4. If DONE_EX = OFF (default), the WAKE_UP keyword will take your selected option (1-25), and then the software will set wake-up signal bits accordingly. If you do not select a wake-up sequence, the default wake-up sequence will be 21.

COMPRESS_CONFIG = OFF (default) | ON

Bitstream Compression. When set to ON, for those devices that support bitstream compression, the software generates a compressed version of the bitstream file.

EARLY_IO_RELEASE =OFF (default) | ON [once]

Program and wakeup the left/right I/O banks early, before the rest of the device is programmed.

BOOTMODE =DUAL (default) | SINGLE | NONE

This setting enables a user to select the booting mode at power up.

DUAL - Perform the dual boot for the booting event. In case the primary booting image (bitstream) fails, the secondary (golden) boot image is invoked to boot device.

SINGLE - Perform a single boot only. If failure occurs, device will move to unprogrammed mode directly.

NONE - No master SPI booting at startup, waits for slave configuration port to configure the device.

MASTER_PREAMBLE_TIMER_CYCLES =600000 (default)

This attribute sets the master preamble timer count value.

The preamble_count value below is the number of clock cycles to get a valid preamble before an error is declared.

000 -> 600000

001 -> 400000

010 -> 200000

011 -> 40000

100 -> 20000

101 -> 4000

110 -> 2000

111 -> 400

USERCODE Format BINARY (default) | HEX | ASCII | AUTO

UserCODE file format.

USERCODE 32'b00000000000000000000000000000000 (default) |
32'b11111111111111111111111111111111

USERCODE is formatted to a 32-bit binary code in the bitstream but you can enter it based on your USERCODE Format selection.

BITSTREAM_REV Format BINARY (default) | HEX | ASCII | TIMESTAMP

BITSTREAM_REV 32'b00000000000000000000000000000000 (default) | 32'b11111111111111111111111111111111

UNIQUE_ID 8'b00000000 (default) | 8'b11111111

BOOT_SEL DUAL (default) | SINGLE

PROGRAMN_RECOVERY DISABLE (default)

MSPI_RESET DISABLE (default)

MULTI_BOOT_MODE DISABLE (default)

DAISY_CHAIN DISABLE (default)

DAISY_CHAIN_WAIT_DONE DISABLE (default)

MSPI_SIGNATURE_TIMER 200MS (default) | 100MS | 50MS | 40MS | 20MS | 1MS | 500US | 100US | INFINITE

MSPI_CLK_PHASE RX_RISING_TX_FALLING (default)

MSPI_CPOL IDLE_CLOCK_LOW (default)

MSPI_SHIFT_ORDER MSB_FIRST (default)

```
48'b0000000000000000000000000000000000000000000000000000000 (default) |  
48'b11111111111111111111111111111111111111111111111111
```

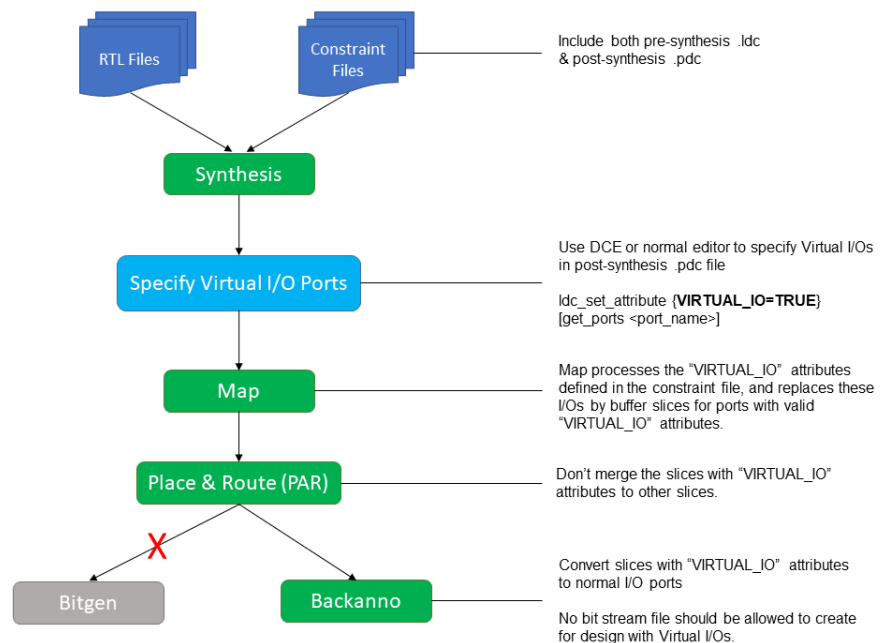
SSO Analysis

The Radiant software enables you to run an analysis of simultaneous switching outputs (SSO). SSO analysis describes the noise on signals caused by a large number of output drivers that are switching at the same time. Analysis of SSO helps ensure that your I/O standards and power integrity meet requirements of the PCB design. This tool can be accessed in the Device Constraint Editor in the SSO tab at the bottom. From there you can set output loads, ground plane PCB noise, SSO Allowance %, and power plane PCB noise values. Once the flow is re-run, an SSO report is generated.

Specifying Virtual I/O Ports

The Radiant Virtual I/O flow is useful in checking the design timing, resource utilization, or performance of a selected device to determine if your modules exceeded the number of physical pins present in the FPGA (e.g., during an IP design). Without the Virtual I/O flow, the implementation will result in an error due to the number of I/Os exceeding the device capacity. You can use the Virtual I/O flow to specify certain ports as "virtual" ports at the top level.

Refer to the following flow chart.



A Virtual I/O is terminated in the LUT instead of a PIO. This is reported as the "Number of feedthrough LUT4s" in the MAP report, and is not included in the number of LUTs in the design implementation.

If the total number of real ports in the design is less than or equal to the total number of I/O ports in the device, you can use the regular Radiant flow for such IP without the Virtual I/O flow.

To specify the Virtual I/O ports in the selected I/Os:

- ▶ In the .pdc file using ldc_set_attribute.

Syntax: ldc_set_attribute {VIRTUAL_IO=TRUE} [get_ports
 <port_name>]
 ldc_set_attribute {VIRTUAL_IO = TRUE} [get_ports ch0_sdrxp]

Notes:

Wildcard "*" is supported for the port names:

```
ldc_set_attribute {VIRTUAL_IO = TRUE} [get_ports datain*]
```

- ▶ Using the DCE Spreadsheet View Virtual column, set the required ports value to "TRUE." By default, the value is set to "FALSE."

See Also

- ▶ [“Checking the Virtual I/O Ports” on page 111](#)
- ▶ [“Verifying Virtual I/O Ports” on page 112](#)

Checking the Virtual I/O Ports

The Virtual I/O attribute for a port is ignored if one of the following conditions is met:

- ▶ The I/O port is with tristate signal.
e.g. tristate output or bi-directional I/O. Tristate support is only in I/O port.
- ▶ The I/O port is with LOCATION constraint.
When user specifies the location (either to pin or bank), it is a real external I/O.
- ▶ The I/O port is with IO_TYPE attribute in RTL or constraint.
When user specifies IO_TYPE, it is a real external I/O. Virtual I/O should not be specified for the ports with IO_TYPE attribute.
- ▶ The I/O port is connected with DELAY module.
- ▶ The I/O port is not a simple input buffer (IB) or output buffer (OB) but a special I/O buffer, either from an IP or RTL instantiation.
e.g. MIPI, SEIO18_CORE, DIFFIO18_CORE, etc.
Its special usage means it is a real external I/O.
- ▶ The I/O port drives a clock signal or is driven by a clock signal.
- ▶ The I/O port is connected to an external I/O directly in a hardware device, such as:
 - ▶ APIO pin of a component (e.g. High speed I/O in PCIE)
 - ▶ JTAG ports

- ▶ Some ports of ADC/PMU/I2C etc. components
- ▶ IDDR's D input
- ▶ ODDR's Q output
- ▶ DQS's DQSI input

Notes:

I/O registers are treated differently in the LFCPNX (CertusPro-NX) and UT24CP (CertusPro-NX-RT) device versus the iCE40UP (iCE40 UltraPlus) device. For the LFCPNX and UT24CP device, IOLOGIC is a whitebox cell model. I/O registers are mapped into IOLOGIC. So, when IREG's D input or OREG's Q output is a Virtual I/O port, the I/O REG will be pushed into PLC (SLICE).

However, for the iCE40UP device, I/O register is in IOLOGIC black box. It is packed by the synthesis tool. As a result, the Virtual I/O constraints will be ignored for ports associated with I/O registers.

See Also ▶ [“Specifying Virtual I/O Ports” on page 110](#)

▶ [“Verifying Virtual I/O Ports” on page 112](#)

Verifying Virtual I/O Ports

There are several ways to verify Virtual I/Os.

Viewing Virtual I/Os in DCE If you have specified the Virtual I/O attributes for some I/O ports in the Post-Synthesis constraint file (.pdc), you can see them in the "Virtual" column. As shown in the following figure, the ports "datain[0]" to "datain[7]" are virtual I/Os.

Name	Group By	Pin	BANK	IO_TYPE	Virtual	CLAMP
clk	N/A	(E12)	(0)	LVCMOS33(LVCM...	FALSE	ON(ON)
datain[0]	N/A	N/A	N/A	N/A	TRUE	N/A
datain[1]	N/A	N/A	N/A	N/A	TRUE	N/A
datain[2]	N/A	N/A	N/A	N/A	TRUE	N/A
datain[3]	N/A	N/A	N/A	N/A	TRUE	N/A
datain[4]	N/A	N/A	N/A	N/A	TRUE	N/A
datain[5]	N/A	N/A	N/A	N/A	TRUE	N/A
datain[6]	N/A	N/A	N/A	N/A	TRUE	N/A
datain[7]	N/A	N/A	N/A	N/A	TRUE	N/A

Changing Virtual I/O attributes in DCE

If you save the result, the Radiant software will ask you to rerun the Map Design and Place & Route Design processes, since the .pdc file has been changed.

Viewing Virtual I/Os in the Map Report file

The number of Virtual I/Os are reported in the Map Report (.mrp) file. Here is an example:

Number of LUT4s:	8 out of 32256 (<1%)
Number used as logic LUT4s:	0
Number used as distributed RAM:	0 (6 per 16X4 RAM)
Number used as ripple logic:	0 (2 per CCU2)
Number of inserted feedthrough LUT4s:	8
Number for virtual IO:	8
Number of Virtual IOs:	8

The report shows that the "Number of Virtual I/Os" is 8, and it used 8 LUTs to replace the specified Virtual I/O ports. These eight LUTs are part of "Number of inserted feedthrough LUT4s."

You can find the individual Virtual I/Os in the "I/O (PIO) Attributes" section in the same .mrp file. If a port is a Virtual I/O, the value is marked as "VIRTUAL" in column "Levelmode IO_TYPE."

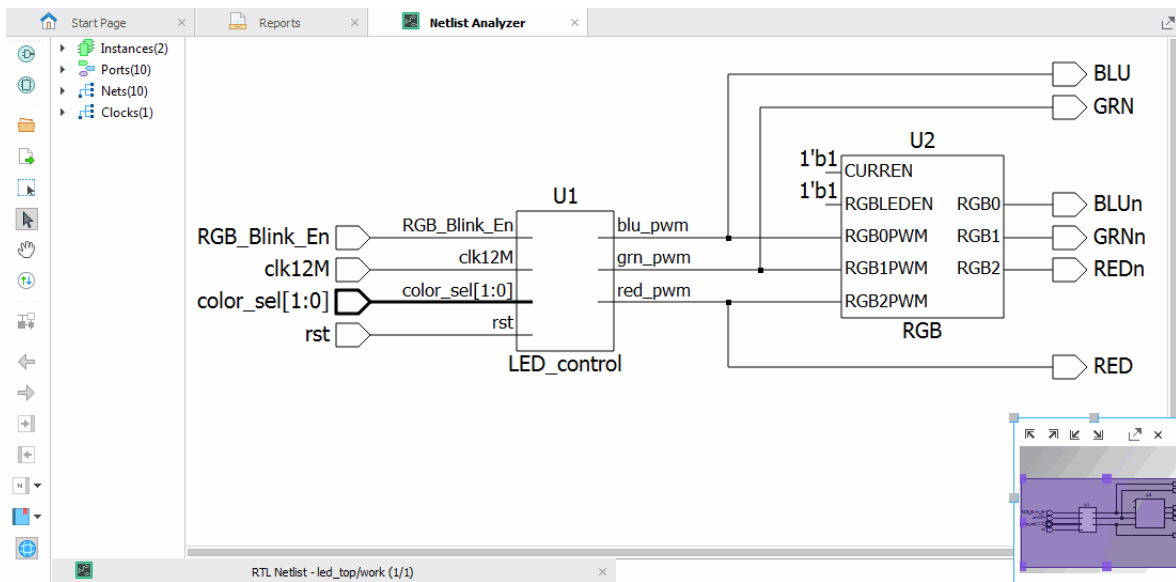
I/O Name	Direction	Levelmode IO_TYPE	I/O REG	I/O DDR	Special I/O Buffer
clk	INPUT	LVCMS33			
datain[7]	INPUT	VIRTUAL			
datain[6]	INPUT	VIRTUAL			
datain[5]	INPUT	VIRTUAL			

See Also

- ▶ ["Specifying Virtual I/O Ports" on page 110](#)
- ▶ ["Checking the Virtual I/O Ports" on page 111](#)

Netlist Analyzer

Netlist Analyzer works with Lattice Synthesis Engine (LSE) and Synplify Pro to produce schematic views of your design while it is being implemented. Use the schematic views to better understand the hierarchy of the design and how the design is being implemented. The Netlist Analyzer window has four parts, as shown in Figure 69.

Figure 69: Netlist Analyzer

- ▶ Tool bar provides buttons for various functions.
- ▶ Browser provides nested lists of module instances, ports, and nets.
- ▶ Schematic view shows a schematic of the design. Depending on the size of the design, the schematic may be made of multiple sheets.
- ▶ World View, which is a miniature view of the sheet, helps you pan and zoom in the schematic view.

Netlist Analyzer can have multiple schematics open. The open schematics are shown on tabs along the bottom of the window.

There are several ways to adjust the view of a schematic and to navigate through the hierarchy. For more information on how to do this, in addition to detailed information about Netlist Analyzer, see “Analyzing a Design” in *Managing Projects* in the Radiant Software Help.

Physical Designer

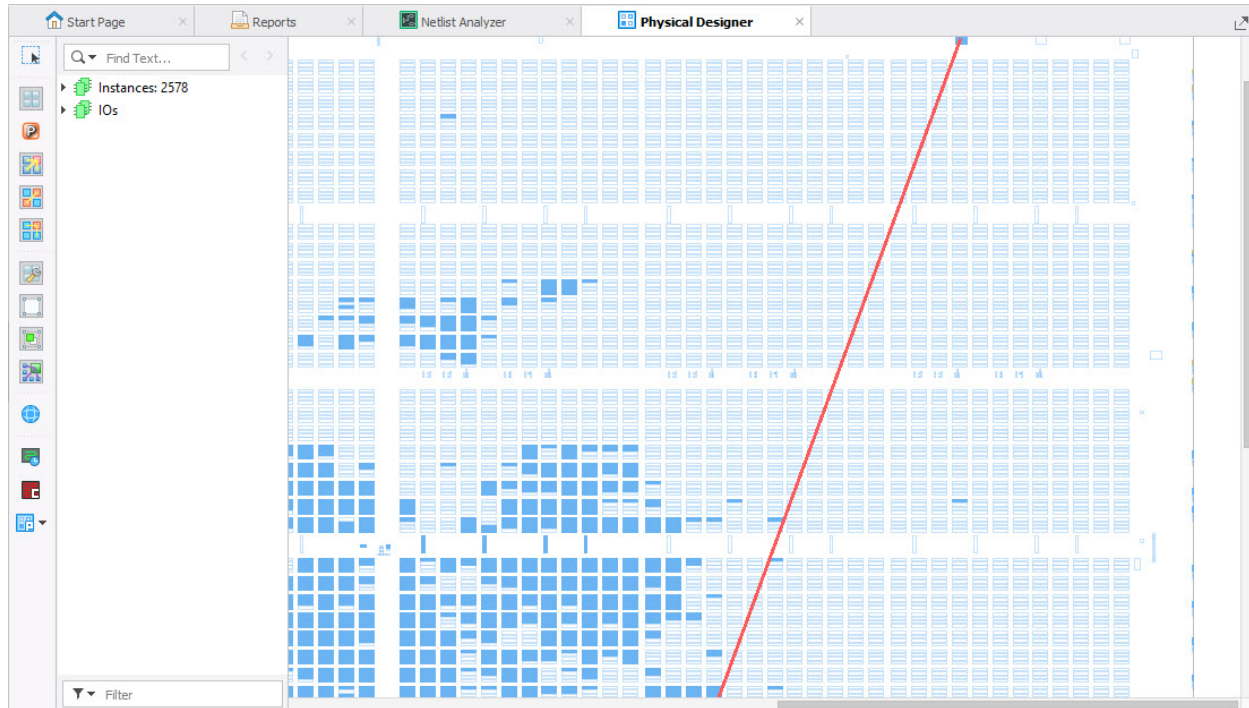
Physical Designer provides one central location where you can do all the floorplanning and be able to view the physical layout of the design. Physical Designer does this with Placement, IO, and Routing modes.

Placement Mode

Placement Mode provides a large-component layout of your design, and is available as soon as the target device has been specified. Placement Mode displays user constraints, and placement and routing information. All connections are displayed as fly-lines. Placement Mode allows you to create REGION and GROUP constraints to keep components together. You can specify the types of components and connections to be displayed. As you move your mouse pointer over the floorplan layout, details are displayed in tool-tips and in the status bar, including:

- ▶ Number of resources for each GROUP and REGION
- ▶ Number of utilized slices for each PLC component
- ▶ Name and location of each component, port, net, and site

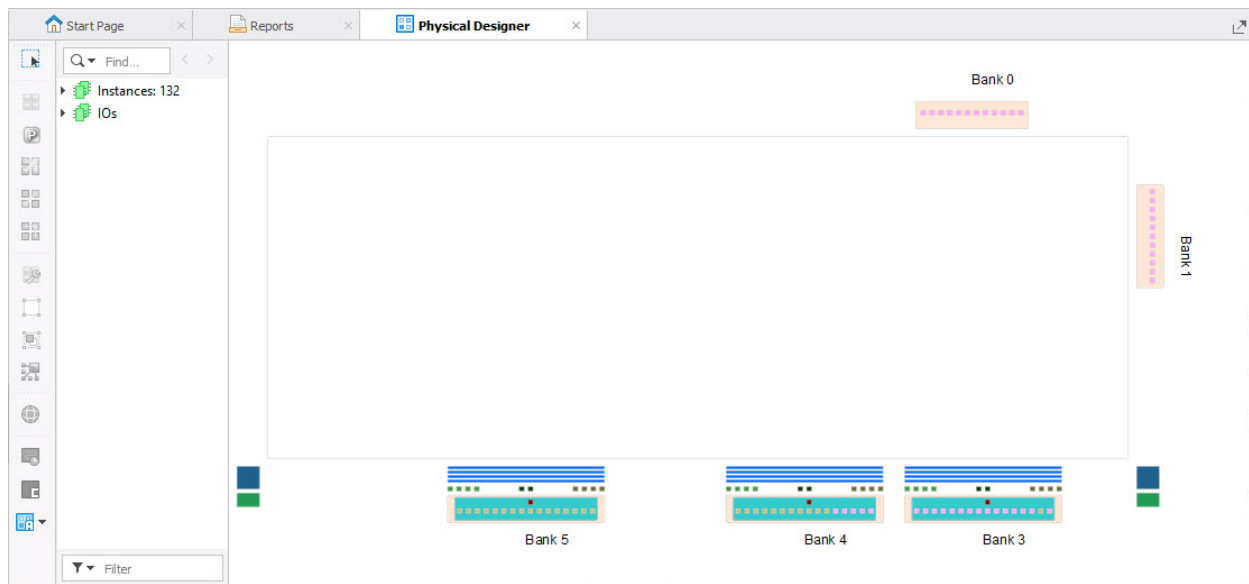
Figure 70: Placement Mode



Placement Mode has a Congestion Timing Map view allowing you to debug timing congested areas of your design. This view gives the timing of the most critical paths within the slack threshold input. There is also the Congestion View, which is a read-only view showing the most congested regions based on wire length and pins selected.

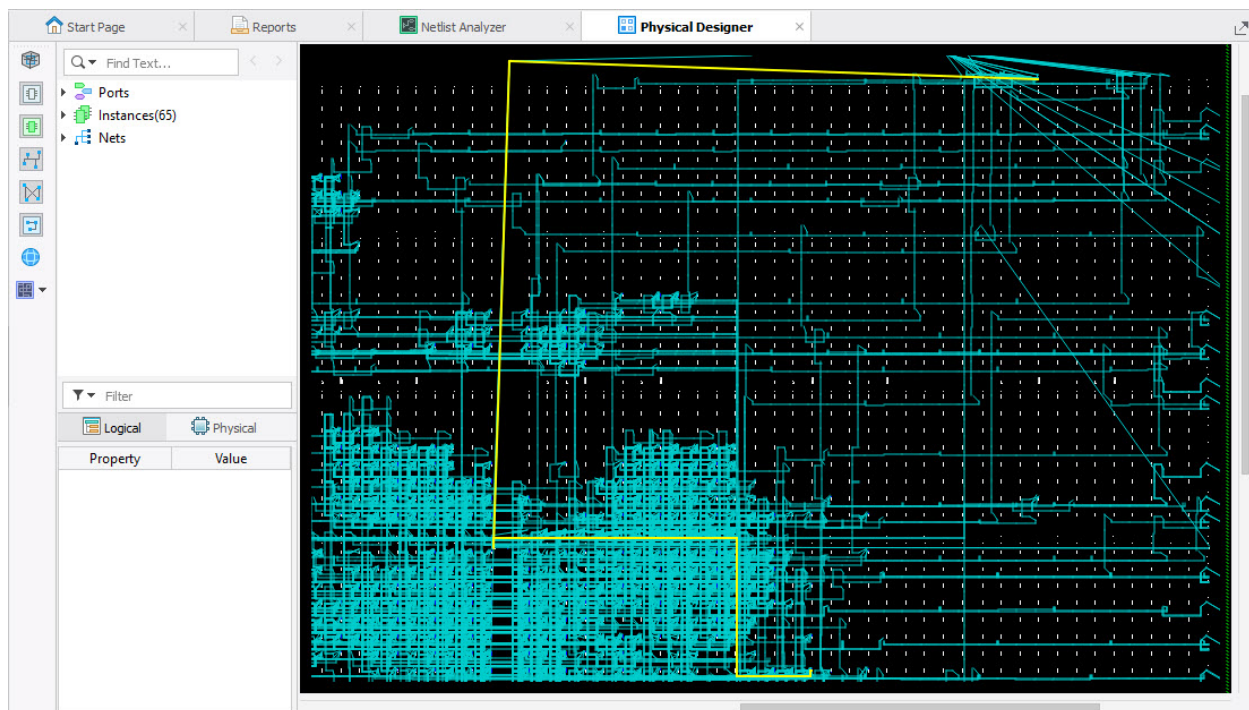
IO Mode

IO Mode is used for I/O assignments such as DDR Interface, DQS, and clock assignments. I/O resource utilization can be displayed by hovering and displayed in tool tips.

Figure 71: IO Mode

Routing Mode

Routing Mode provides a read-only detailed layout of your design that includes physical wire connections. Routed connections are displayed as Manhattan-style lines, and unrouted connections are displayed as fly-lines.

Figure 72: Routing Mode

As you move your mouse pointer slowly over the layout, the name and location of each REGION, group, component, port, net, and site are displayed as tool tips and also appear in the status bar. The tool tips and status bar also display the group name for components that are members of a group.

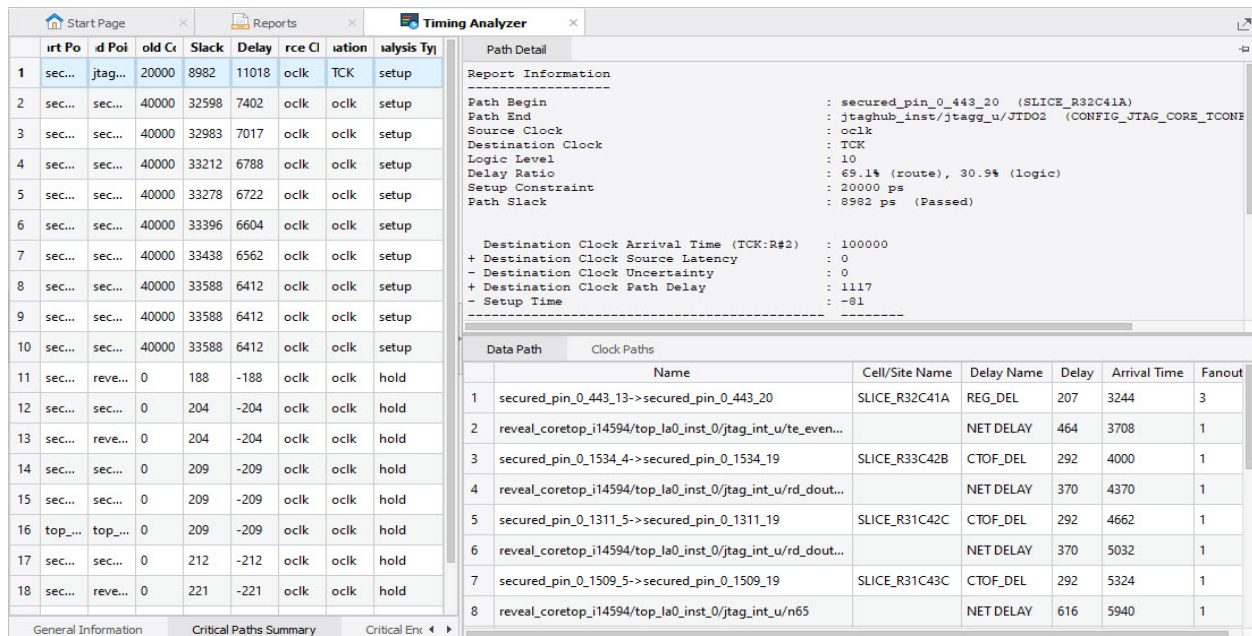
The Routing Mode toolbar allows you to select the types of elements that will be displayed on the layout, including components, empty sites, pin wires, routes, and timing paths. Routing Mode is available after placement and routing.

Timing Analyzer

Timing Analyzer analyzes timing constraints that are present in the .ldc and .pdc files. These timing constraints are defined in the Timing Constraint Editor or in a text editor before the design is mapped. A Timing Analysis report file, which shows the results of timing constraints, is generated each time you run the LSE, Map Timing Analysis process, or the Place & Route Timing Analysis (PAR) process. Place & Routing Timing Analysis results can then be viewed in the Timing Analyzer windows.

The Map Timing Analysis report (.tw1) contains estimated routing that can be used to verify the expected paths and to provide an estimate of the delays before you run Place & Route. The PAR Timing Analysis report (.twr) contains delays based on the actual placement and routing and is a more realistic estimate of the actual timing.

Figure 73: Timing Analyzer



Timing Analyzer consists of five tabs of information:

- General Information

- ▶ Critical Paths Summary
- ▶ Critical Endpoint Summary
- ▶ Unconstrained Endpoint Summary
- ▶ Query

Each tab can be detached from the main window, rearranged, and resized. When you select a constraint from the Constraint pane, you can view the path table details on one pane, and Timing Analyzer report in the other. For detailed information about Timing Analyzer see *Analyzing Static Timing* in the Radiant Software Help.

Using Standalone Timing Analyzer

Standalone Timing Analyzer can be run on designs using post-synthesis, post-map, post-PAR Unified Design Database (.udb), and associated timing constraints specified in the .pdc file of the design. Using these input files, it provides static timing analysis, path tables, and a report of each timing constraint like the Timing Analyzer in Radiant.

Running Standalone Timing Analyzer from Radiant Timing Analyzer

When you open the Standalone Analyzer tool, it will automatically create the Unified Design Database, which includes logic and physical data (standalone_tav_*.udb). This file is saved to the current project. You can use this database to run tavmain.exe.

To open and run a timing analysis in Standalone Timing Analyzer:

- ▶ In the Radiant software main window, click the  button in the upper left. You can select a different .udb file.
- ▶ If you wish to open the Standalone Timing Analyzer directly, click the  button in the upper left or choose **Tools > Standalone Timing Analyzer**.

See Also

- ▶ [Running Standalone Timing Analyzer from the Command Line](#)

Loading Timing Constraint File and Running Timing Analysis in Standalone Timing Analyzer

To load a timing constraint file and run timing analysis:

1. In the Timing Analyzer main window, choose **File > Load Timing Constraint File**. This menu can be used after opening the design.

2. After selecting the constraint file, you can load the timing constraint and do the timing analysis based on the selected constraint file.
3. If you select Load Timing Constraint File (Incremental Flow), do the timing analysis based on the constraints in the .udb file and the selected constraint file.

Exporting Timing Report and Exporting Timing Analysis Reports in Standalone Timing Analyzer

To export timing report and timing analysis report:

- ▶ In the Timing Analyzer main window, choose **File > Export > Timing Report For Setup/Hold**. This menu can export timing analysis reports for setup or hold (.twr).

Exporting Timing Constraints to PDC File in Standalone Timing Analyzer

To export timing constraints to PDC file:

- ▶ In the Timing Analyzer main window, choose **File > Export > Export Timing Constraints To PDC file**.
This menu can be used after opening the design.
- ▶ You can export timing constraints in the current .udb file to a .pdc file.

Saving Design Files in Standalone Timing Analyzer

To save design files in Standalone Timing Analyzer:

- ▶ You can save changes to design data. In the Timing Analyzer main window, choose **File > Save Design As**.

Setting Options in Standalone Timing Analyzer

Setting options in Standalone Timing Analyzer is the same as setting options in Radiant Timing Analyzer." Refer to ["Setting Options in Timing Analyzer" on page 292](#).

To set options in Timing Analyzer:

- ▶ In the Timing Analyzer main window, choose **Edit > Timing Option Setting**, or click the  button in the upper left.
This menu can be used after opening the design.

- ▶ When loading the post-synthesis .udb file, you cannot change the speed in the Timing Option Setting dialog box. This feature is the same as the timing report.
- ▶ The default for the Report Format is Lattice Standard. You can change the Report Format in the Timing Option Setting dialog box.

Setting Operating Conditions in Standalone Timing Analyzer

To set operating conditions in Standalone Timing Analyzer:

1. In the Timing Analyzer main window, choose **Edit > Operating Conditions Setting**.
You can change the voltage and temperature based on the current design data.
2. When you click the **OK** button, the timing analysis process starts based on the edited design data.
3. When loading the post-synthesis .udb file, you cannot change the settings in the Operating Conditions Setting dialog box. This feature is the same as timing report.

Running Timing Constraint Editor in Standalone Timing Analyzer

To run timing constraint editor in Standalone Timing Analyzer:

1. In the Timing Analyzer main window, choose **Edit > Timing Constraint Editor**. After selecting the Timing Constraint Editor, you can edit the timing constraint design based on the current design data.
2. When you close the Post-Synthesis Timing Constraint Editor window, a dialog box appears. If you click yes, the timing analysis process starts based on the edited design data.

Refreshing Timing in Standalone Timing Analyzer

To refresh timing in Standalone Timing Analyzer:

- ▶ In the Timing Analyzer main window, choose **Edit > Refresh**. Based on the current timing constraint in the .pdc file, you can do the timing analysis to check if the new constraint works as expected.

To edit the location of the current .pdc file: ▶ You can add the new constraint in the current .pdc file.

Click the **Refresh** button to start the timing analysis process.

- ▶ To add the new constraint in the .pdc file, you can copy the text from the timing table.
- ▶ If the location of the current .pdc file is not specified, you can also choose **File > Load Timing Constraint File** to specify the .pdc file and start the timing analysis process.

Reveal Inserter

Reveal Inserter lets you add debug information to your design to allow hardware debugging using Reveal Analyzer. Reveal Inserter enables you to select the design signals to use for tracing, triggering, and clocking. Reveal Inserter will automatically generate the debug core, and insert it into a modified design with the necessary debug connections and signals. Reveal Inserter supports VHDL and Verilog sources. After the design has been modified for debug, it is mapped, placed and routed with the normal design flow in the Radiant software.

For more information about Reveal Inserter, refer to the *Reveal User Guide for Radiant Software*. Also, refer to *Testing and Debugging On-Chip* in the Radiant Software Help.

Using the Reveal Controller Simulation Model

The Reveal Controller Simulation Model replicates the functionality of Reveal Controller in simulation.

To set up the Reveal Controller Simulation Model:

1. Choose **Tools > Reveal Inserter** to insert a reveal module into the design.
2. Run **Synthesize Design** to generate the reveal_workspace file.
 - ▶ If you insert Reveal Controller only, the tool will create the sim_src folder with the pre-synthesis Reveal generated source file under reveal_workspace.
 - ▶ If you insert Reveal Controller and Analyzer, the Reveal engine will not create the folder and no source files will be provided.
3. Click **Tools > Simulation Wizard** or click the  icon in the toolbar.
4. In the Simulator Project Name and Stage dialog box, enter the project name.
Click **Next**.
5. In the Add and Reorder Source dialog box, select and add the source files from the sim_src folder.
Click **Next**.

The following compiler directives are all enabled in the source file. You can use the OEM simulator or any stand-alone simulator to compile and simulate the Controller:

- ▶ ``define en_sw`
- ▶ ``define en_led`
- ▶ ``define en_user_mem`
- ▶ ``define en_user_creg`
- ▶ ``define en_user_sreg`

Data Capture Mode

The data capture mode specifies whether Reveal Analyzer can look for one trigger or multiple triggers in a test run. Multiple Trigger Capture mode provides the greatest flexibility during test runs. Single Trigger Capture mode slightly reduces the amount of FPGA resources needed.

To set the data capture mode:

1. In the Data Capture Mode section, select one of the following:
 - ▶ **Single Trigger Capture.** Reveal Analyzer captures the data for only one trigger.
 - ▶ **Multiple Trigger Capture.** Reveal Analyzer captures the data for multiple triggers. The number of triggers is specified in Reveal Analyzer unless you are creating a module to monitor the power-on reset (POR) functions (see below).
2. If you select Multiple Trigger Capture, choose the “Minimum samples per trigger.” The number of samples collected for each trigger is set in Reveal Analyzer but cannot be smaller than this value.
3. If you select Multiple Trigger Capture and are creating a POR module, choose the “Number of triggers for POR.” For an explanation of POR modules, see [“Power-on Reset \(POR\) Debug” on page 122](#).

Power-on Reset (POR) Debug

An automatic “trigger enable” signal must be built into the Reveal module to monitor POR functions. Before Reveal Analyzer starts, these functions happen immediately after the power-on of the test board. When the trigger enables signal transitions to “active,” the module will watch for the trigger and collect samples. It is similar to clicking the Run  button in Reveal Analyzer. When Reveal Analyzer starts, it loads and displays any data collected by the POR modules.

POR modules are not supported in iCE40UP devices.

To set a POR trigger enable:

1. In the POR Debug section, select **Trigger Enable**.
2. Find the POR trigger signal in the Design Tree view and drag it to the text box in the POR Debug section.
3. Choose whether the signal is **Active High** or **Active Low**.

Configuring User Memory Setup

Setting up the User Memory setting consists of assigning six mandatory ports. (which must be defined in user RTL design) The maximum data width is 32 bits.

The user design must include a memory block (EBR, Distributed Memory or PMI Memory) to be able setup User Memory.

The mandatory signals in the user design to setup User Memory are:

- ▶ Clock, Clock_enable, Wr_Rdn, Address, WData and RData.

Note:

In future Reveal Inserter releases, each entry width up to 8 bits will be allowed to support extended address range. If you have a Reveal Inserter design that is intended for future use, limit the bus widths to 8 bits.

To set up User Memory signals:

1. Select the top_Controller core in the Dataset view. (There can only be one Controller core.)

When you select Add Controller, it displays the control panel. You can enable User Memory by choosing the corresponding memory port names. The Address and Data Width will be updated automatically.

2. Click the **User Memory Setup** tab.

☒ User Memory Setting

Address: 0x90000000-0x9000FFFF

Address Width: 9 (4-16)

Data Width: 32 (4-32)

	Memory Port
Clock	top_test1_inst/pmi_ram_dp_inst/Clock
Clock_enable	top_test1_inst/pmi_ram_dp_inst/ClockEn
Wr_Rdn	top_test1_inst/pmi_ram_dp_inst/WE_RDN
Address[8:0]	top_test1_inst/pmi_ram_dp_inst/Address[8:0]
WData[31:0]	top_test1_inst/pmi_ram_dp_inst/WData[31:0]
RData[31:0]	top_test1_inst/pmi_ram_dp_inst/Q[31:0]

Virtual Switch & LED Setup **User Memory Setup** Hard IP Setup User Status Register Setup User Control Register Setup

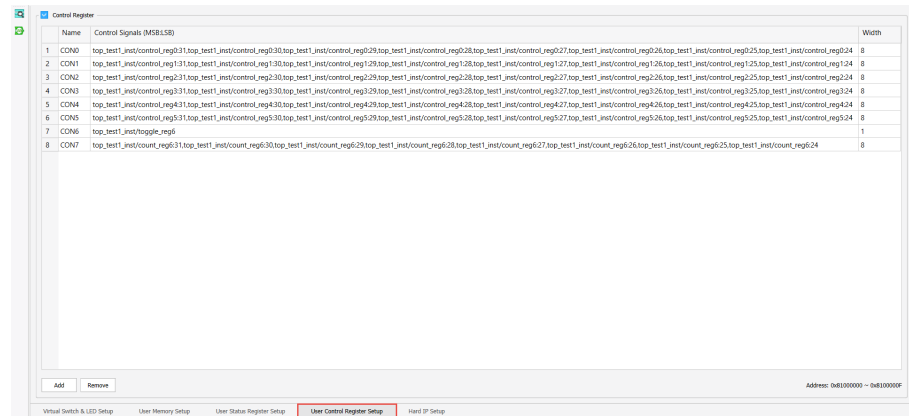
3. Select signals in the Design Tree and drag the signals to the desired position in the Setting List in the User Memory Setup tab. For help finding signals, see [“Searching for Signals” on page 467](#).
4. Drag the desired signals in to the User Memory data panel to the right. The width of the signals will automatically update between 4-32 entries at the top.

5. Beside the title User Memory Setting, there is a check box in which Controller function for User Memory signals can be enabled/disabled.

Configuring User Control Register Setup

The User Control Register Setup tab displays a table-like panel where you can enable the Control Register. The Control Register address ranges from 0x81000000 up to 0x81000007.

1. Click the **User Control Register Setup** tab.



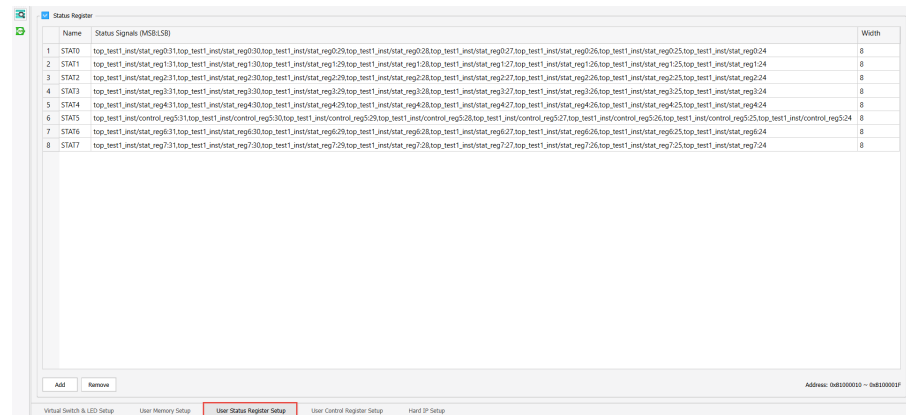
2. Click the **Add** button to drag-and-drop the corresponding signals from Design View to the Control Signals column of the User Control Register Setup tab.
3. If you wish to delete a row from the Control Signals column, you can click the **Remove** button to delete the selected row.
4. You can also double-click the Control Signals column to drag-and-drop signals and change the order of signals.

Configuring User Status Register Setup

The User Status Register Setup tab displays a table-like panel where you can enable the Status Register. The Status Register address ranges from 0x81000010 up to 0x81000017.

To set up User Status Register:

1. Click the **User Status Register Setup** tab.



- Click the **Add** button to drag-and-drop the corresponding signals from Design View to the Status Signals column.
- If you wish to delete a row from the Status Signals column, you can click the **Remove** button to delete the selected row.
- You can also double-click the Status Signals column to drag-and-drop signals and change the order of signals.

Reveal Analyzer

After you generate the bitstream, you can use Reveal Analyzer to debug your FPGA circuitry. Reveal Analyzer gives you access to internal nodes inside the device so that you can observe their behavior. It enables you to set and change various values and combinations of trigger signals. After the specified trigger condition is reached, the data values of the trace signals are saved in the trace buffer. After the data is captured, it is transferred from the FPGA through the JTAG ports to the PC.

For more information about Reveal Analyzer, refer to the *Reveal User Guide for Radiant Software*. Also, refer to *Testing and Debugging On-Chip* in the Radiant Software Help.

User Memory Analysis

The second tab is for User Memory Analysis. The title indicates the range of the User Memory map. Analysis follows no particular steps or order.

- ▶ **Default Data:** A value that will initialize all memory locations.
- ▶ **Write Address/Data:** Address and data in hex to be written.
- ▶ **Read Address/Data:** Enter address. The data is read back when 'Read' is selected.
- ▶ **Memory File:** You can dump from/to a range of memory addresses to a .mem file. A user can also **Load MemFile** to load a pre-configured memory file.

Note:

In future Reveal Controller releases, each entry width up to 8 bits will be allowed to support extended address range. If you have a Reveal Controller design that is intended for future use, limit the bus widths to 8 bits.

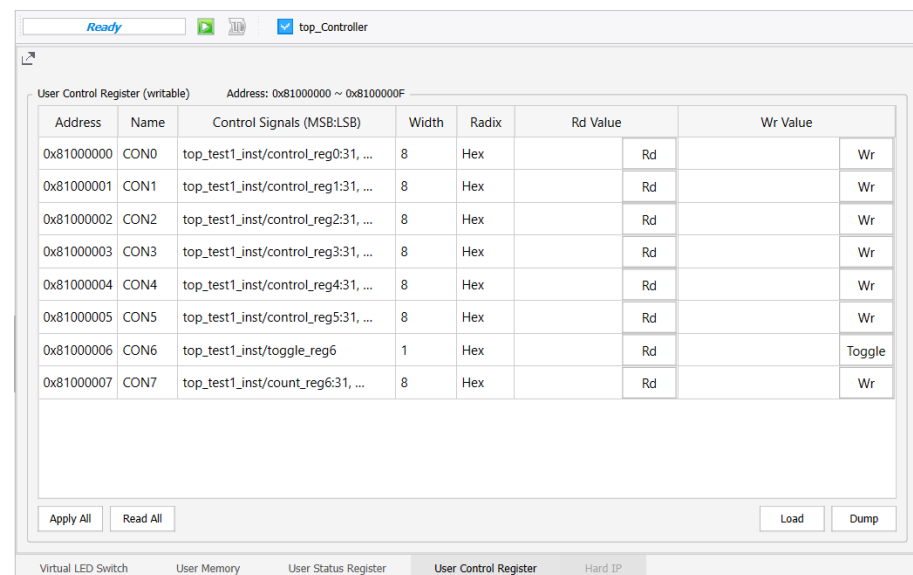
Running User Control Register

1. In the **User Control Register** tab, click the **Rd** button.

It will read the value of the corresponding Name and Address by executing the TCL command.

2. Click the **Wr** button.

It will write the value of the corresponding Name and Address by executing the TCL command.



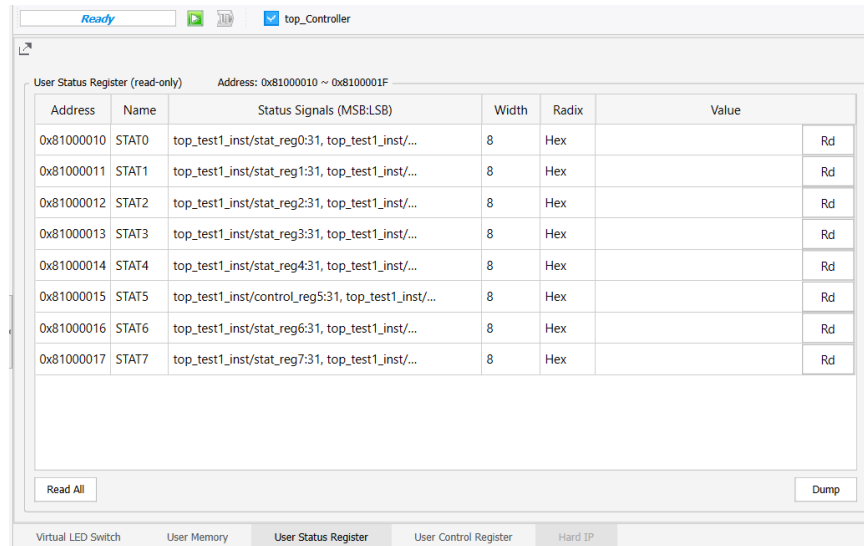
You can click the Toggle button to read and reverse the value of the corresponding Name and Address by executing the TCL command.

3. Double-click a row in the Control Signals column to display the list of signals with their corresponding read value.
4. In the User Control Register tab, you can also click the following buttons:
 - ▶ **Apply All:** Applies the value of all Name and Address by executing multiple TCL commands sequentially.
 - ▶ **Read All:** Reads the value of all Name and Address by executing multiple TCL commands sequentially.
 - ▶ **Load:** Loads the value of the register file to Control Register.
 - ▶ **Dump:** Saves the value of Control Register to a .txt file.

Running User Status Register

1. In the **User Status Register** tab, click the **Rd** button.

It will read the value of the corresponding Name and Address by executing the TCL command.



2. On the bottom left corner, click the **Read All** button.

The Read All button reads the value of all Name and Address by executing multiple TCL commands sequentially.

3. On the bottom right corner, click the **Dump** button.

The Dump button saves the value of Status Register as a .txt file.

4. Double-click a row in the Status Signals column to display the list of signals with their corresponding read value.

Running Reveal Eye-Opening Monitor

The Reveal Eye-Opening Monitor can be launched from the Reveal Analyzer/Controller. You need to be connected to a board to be able to run the Eye-Opening Monitor.

To launch the Eye-Opening Monitor:

1. In the Radiant software main window choose **Tools > Reveal Analyzer/Controller**

In the dropdown menu, choose **top_Controller** to see the **Hard IP** tab. Scroll down to the PCS channels to see the **Eye-Opening Monitor** button.

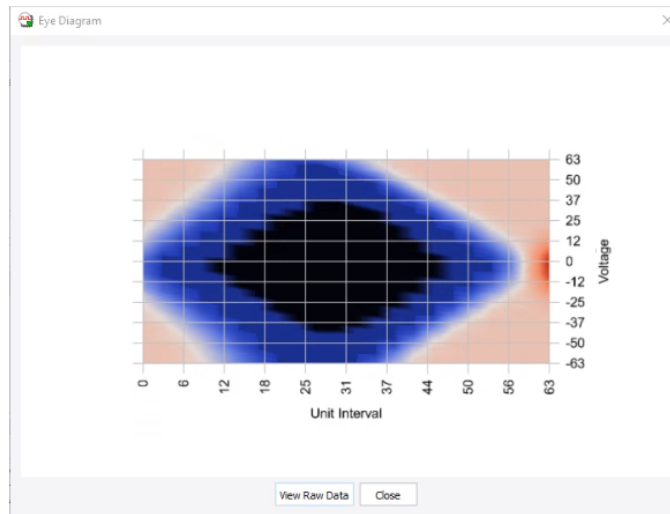
2. Click the **Eye-Opening Monitor** button in the upper right of the PCS channel.

The Eye-Opening Monitor dialog box appears.

3. In the Eye-Opening Monitor dialog box, select the quality of the eye diagram from the dropdown menu.

It is recommended to select normal or low quality for a faster result, and select high quality for the final result. The default is normal quality.

4. Click the **Run** button to start the EOM process. After the EOM has finished running, it will display the eye diagram for the current PCS channel. The diagram below is an example of the eye diagram.



5. At the bottom of eye diagram, click the **View Raw Data** button to open the raw data file containing the number of samples and errors used in calculating the density of the eye diagram.

	x	y	nsamples_cur	nsamples_pre	errors_cur	errors_pre
1	0	-63	0x1e0000	0x1e0000	0x000000	0x0f3b22
2	0	-60	0x1e0000	0x1e0000	0x000000	0x0f39bc
3	0	-57	0x1e0000	0x1e0000	0x000000	0x0f3ac3
4	0	-54	0x1e0000	0x1e0000	0x000000	0x0f3a35
5	0	-51	0x1e0000	0x1e0000	0x000000	0x0f3a70
6	0	-48	0x1e0000	0x1e0000	0x000000	0x0f39f0
7	0	-45	0x1e0000	0x1e0000	0x000000	0x0f3bf7
8	0	-42	0x1e0000	0x1e0000	0x000000	0x0f3819
9	0	-39	0x1e0000	0x1e0000	0x000000	0x0f338f
10	0	-36	0x1e0000	0x1e0000	0x000000	0x0f26f7
11	0	-33	0x1e0000	0x1e0000	0x000000	0x0f0e98
12	0	-30	0x1e0000	0x1e0000	0x000000	0x0dedb7f
13	0	-27	0x1e0000	0x1e0000	0x000004	0x0e6928
14	0	-24	0x1e0000	0x1e0000	0x000029	0x0dc442
15	0	-21	0x1e0000	0x1e0000	0x0000f4	0x0d0d33
16	0	-18	0x1e0000	0x1e0000	0x000712	0x0b6720
17	0	-15	0x1e0000	0x1e0000	0x0013cf	0x0a2d8f

- ▶ The raw data comes from the eom register.
- ▶ The numbers from nsample_cur, nsample_pre, errors_cur, and errors_pre columns calculate the result.

- ▶ The numbers are collected by reading the eom register of the PCS IP.

If you change the Reveal Controller options, you can re-run the EOM to display the new eye diagram and view the new raw data until the desired result is achieved.

Reveal Controller

Reveal Controller allows you to emulate an otherwise unavailable environment for power debug. For example, your evaluation board would only have a limited number of LEDs or switches but the virtual environment enables up to 32 bits. Register memory mapping and dumping of values is also easily manifested while visibility into Hard IP is also enabled.

For more information about Reveal Controller, refer to the *Reveal User Guide for Radiant Software*. Also, refer to *Testing and Debugging On-Chip* in the Radiant Software Help.

Power Calculator

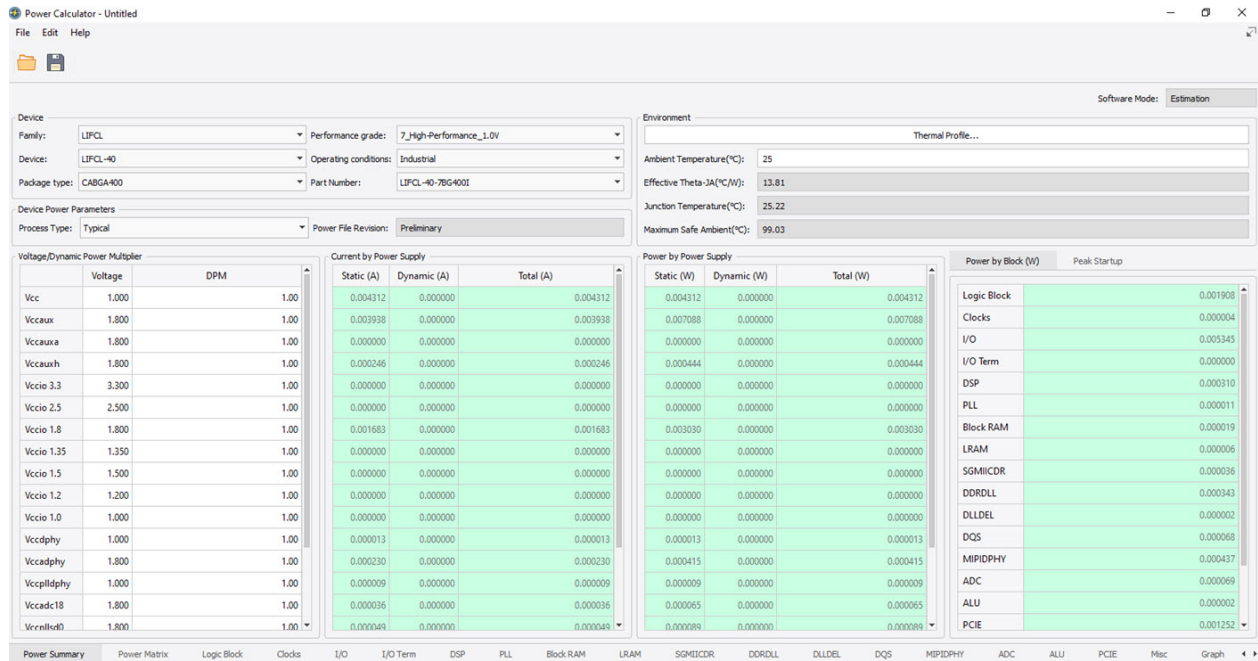
Power Calculator estimates the power dissipation for your design. It uses parameters such as voltage, temperature, process variations, air flow, heat sink, resource utilization, activity and frequency to calculate the device power consumption. It reports both static and dynamic power consumption.

To launch Power Calculator from the Radiant software, choose **Tools > Power Calculator** or click the Power Calculator button  on the toolbar.

Power Calculation Modes Power Calculator opens in estimation mode or calculation mode, depending on the status of the selected .pcf file. If it opens in calculation mode, the Bank settings will be from background power database, not from the constraint file. When you make certain design changes in calculation mode, Power Calculator reverts to estimation mode.

Power Calculator Pages When Power Calculator opens, it displays the Power Summary page, which enables you to change the targeted device, operating conditions, voltage, and other basic parameters. Updated estimates of power consumption are then displayed based on these changes. Tabs for other pages, including Power Matrix, Logic Block, Clocks, I/O, I/O Term, Block RAM, Graph, and Report, are arranged across the bottom. The number and types of these pages depends on the target device.

Figure 74: Power Calculator

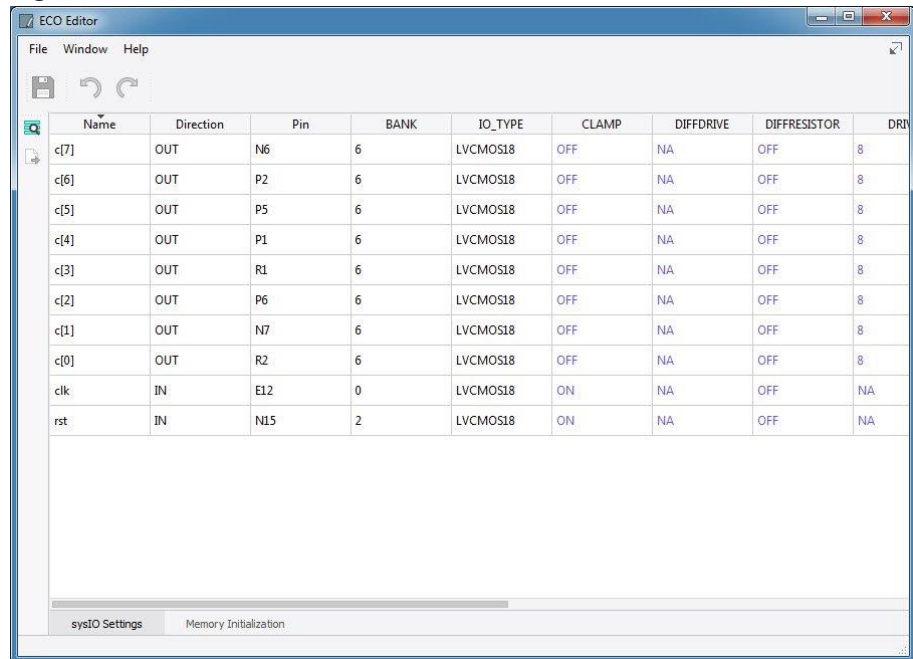


Power Calculator is also available as a non-integrated tool, which you can launch without opening the Radiant software. The non-integrated Power Calculator provides all the same functionality as the integrated version. To open the non-integrated Power Calculator from the Windows Start menu, select **Lattice Radiant Software > Accessories > Power Calculator**. The Startup Wizard enables you to create a new Power Calculator project, based on a selected device or a processed design, or to open an existing Power Calculator project file (.pcf).

For more information on Power Calculator see *Analyzing Power Consumption* in the Radiant Software Help.

ECO Editor

The Engineering Change Order (ECO) Editor enables you to safely make changes to an implemented design without having to rerun the entire process flow. Choose **Tools > ECO Editor** or click the ECO Editor button  on the toolbar.

Figure 75: ECO Editor

ECOs are requests for small changes to be made to your design after it has been placed and routed. The changes are directly written into the Unified Database (.udb) file without requiring that you go through the entire design implementation process.

ECOs are usually intended to correct errors found in the hardware model during debugging. They are also used to facilitate changes that had to be made to the design specification because of problems encountered when other FPGAs or components of the PC board design were integrated.

The ECO Editor includes windows for editing I/O preferences, and memory initialization values. It also provides a Change Log window that enables you to track the changes between the modified .udb file and the post-PAR .udb file.

Note

After you edit your post-PAR UDB file, your functional simulation and timing simulation will no longer match.

For more information, see *Applying Engineering Change Orders* in the Radiant Software Help.

Programmer

The Radiant Programmer is a system for programming devices. The software supports serial programming of Lattice devices using PC and Linux environments. The tool also supports embedded microprocessor programming.

For more information about Programmer and related tools, refer to the *Programming Tools User Guide for Radiant Software*. Also, refer to *Programming the FPGA* in the Radiant Software or Stand-Alone Programmer Help.

Run Manager

Run Manager runs the processes for the different implementation/strategy combinations. Choose **Tools > Run Manager** or click the Run Manager button  on the toolbar.

Run Manager takes the design through the entire process flow for each selected implementation. If you are running on a multi-core system, Run Manager will distribute the iterations so that they are executed in parallel. The option “Maximum implementation processes running in Run Manager” is available in the General section of the Options dialog box. Choose **Tools > Options** to access it. This option enables you to set the maximum number of processes to run in parallel. Generally, the maximum number of processes should be the same as the number of cores in your processor; but if the strategy is using the “Multi-Tasking Node List” option for Place & Route Design, this number should be set to one.

You can use the Run Manager list to set an implementation as active. Right-click the implementation/strategy pair and choose **Set as Active**.

For an implementation that uses multiple iterations of place-and-route, you can select the iteration that you want to use as the active netlist for further processes. Expand the implementation list, right-click the desired iteration, and choose **Set as Active**. The active iteration is displayed in italics.

To examine the reports from each process, set an implementation as active, and then select the Reports view.

See the *Managing Projects* section of the Radiant Software Help for more information about using implementations, strategies, and Run Manager.

Synplify Pro for Lattice

Synplify Pro for Lattice is an OEM synthesis tool used in the Radiant software design flow. Synplify Pro runs in batch mode when you run the Synthesize Design step in Process View. To examine the output report, select **Synplify Pro** in the Process Reports folder of Reports View.

You can also run Synplify Pro in interactive mode. Choose **Tools > Synplify Pro for Lattice** or click the Synplify Pro button  on the toolbar.

For more information, see the *Synplify Pro User Guide*, which is available from the Radiant Start Page or the Synplify Pro Help menu.

Mentor ModelSim

The Mentor® ModelSim® tool is an OEM simulation tool that is closely linked to the Radiant software environment. It is not run as part of the Process implementation flow. To run ModelSim, choose **Tools > ModelSim Lattice-Edition** or click the ModelSim button  on the toolbar.

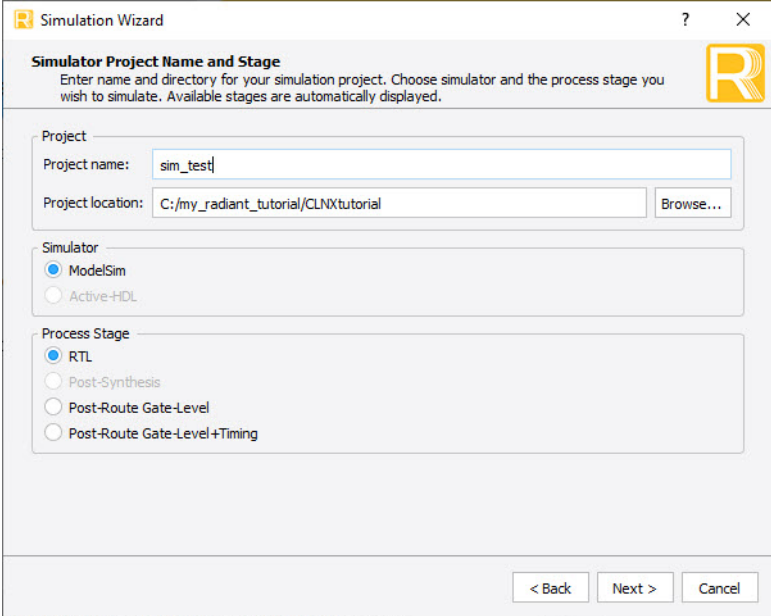
See [“Simulation Flow” on page 88](#) for more information about simulating your design. See [“Simulation Wizard Flow” on page 89](#) for information about creating a simulation project to run in ModelSim.

For complete information about ModelSim, see ModelSim’s online documentation, which is available from the ModelSim Help menu.

Simulation Wizard

The Simulation Wizard enables you to create a simulation project for your design. To open Simulation Wizard, choose **Tools > Simulation Wizard** or click on the icon  in the Radiant software toolbar. The wizard leads you through a series of steps that include selecting a simulation project name, location, specifying the simulator type, selecting the process stage to use, and selecting the source files. To learn more about the flow, view [“Simulation Wizard Flow” on page 89](#).

Figure 76: Simulation Wizard



Simulation Wizard

Simulator Project Name and Stage
Enter name and directory for your simulation project. Choose simulator and the process stage you wish to simulate. Available stages are automatically displayed.

Project
Project name:
Project location:

Simulator
☒ ModelSim
☐ Active-HDL

Process Stage
☒ RTL
☐ Post-Synthesis
☐ Post-Route Gate-Level
☐ Post-Route Gate-Level+Timing

< Back Next > Cancel

Source Template

Source Template provides templates for creating VHDL, Verilog, and constraint files. Templates increase the speed and accuracy of design entry. You can drag and drop a template directly to the source file. You can also create your own templates. For more information, see [“Source Template” on page 28](#).

IP Catalog

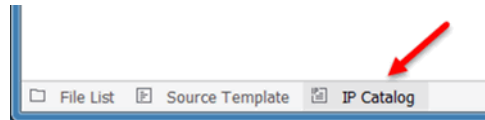
IP Catalog is an easy way to use a collection of functional blocks from Lattice Semiconductor. There are two types of functional blocks available through IP Catalog: modules and IP. IP Catalog enables you to extensively customize these blocks. They can be created as part of a specific project or as a library for multiple projects.

- ▶ **Modules:** These basic, configurable blocks come with IP Catalog. They provide a variety of functions including I/O, arithmetic, memory, and more. Open IP Catalog to see the full list of what’s available.
- ▶ **IP:** Intellectual property (IP) are more complex, configurable blocks. They are accessible through IP Catalog, but they do not come with the tool. They must first be downloaded and installed in a separate step before they can be accessed from IP Catalog.

Overview of the IP Catalog Process

Below are the basic steps of using IP Catalog modules and IP.

1. Open IP Catalog. IP Catalog is accessed via a tab at the lower left of the Radiant software. Click the tab to view the list of available modules and IP.



2. Customize the module/IP. These modules and IP can be extensively customized for your design. The options may range from setting the width of a data bus to selecting features in a communications protocol. At a minimum you need to specify the design language to use for the output.
3. Generate the module/IP and bring its .ipx file into your project. Prior to generating the module/IP, select the option “Insert to project.” This will then automatically bring the .ipx file into your project after the generation step completes. If you do not select this option, add the .ipx file to your project after generation as you would with any other source file (such as a Verilog or VHDL file).
4. Instantiate the module/IP into the project’s design. An HDL instance template is created during generation to simplify this step.
5. IP Catalog modules and IP can be further modified or updated later. After the .ipx file has been added to the Radiant software project, it is visible in the project’s file list. Double-clicking the .ipx file brings up the module/IP’s configuration dialog box where changes can be made and the generation process repeated.

For more information on IP Catalog, see *Designing with Modules* in the Radiant Software Help.

IP Packager

IP Packager is a tool for you to create and IP package easily. You can edit ports, files, parameters, and memory in the IP Packager, and pack a customized IP directly.

See Also

- ▶ [“Running IP Packager and Viewing Documentation” on page 135](#)
- ▶ [“Installing IP Created with IP Packager into IP Catalog” on page 135](#)

Running IP Packager and Viewing Documentation

IP Packager is a stand-alone tool. IP Packager can be run from both Windows and Linux. IP Packager documentation is contained within the IP Packager tool.

To run IP Packager:

- ▶ In Windows, go to the Windows Start menu and choose **Programs > Lattice Radiant Software > Accessories > IP Packager**.
- ▶ In Linux: from the `./<Radiant Software Install Path>/bin/linux64` directory, enter the following on a command line:

```
./ippack.sh
```

The IP Packager tool opens.

To view IP Packager documentation:

- ▶ In the IP Packager main window, choose **Help > Help**.

The IP Packager can also be run from the command line. Refer to [“IP Packager” on page 186](#).

See Also

- ▶ [“IP Packager” on page 135](#)
- ▶ [“Installing IP Created with IP Packager into IP Catalog” on page 135](#)

Installing IP Created with IP Packager into IP Catalog

You can download and add IP created with IP Packager into IP Catalog.

To download and add a User IP:

1. In the Radiant IP Catalog, click on the **Install a User IP**  button.
2. In the **Select user IP package file to install** dialog box, browse to the IP Package (.ipk) file, and click **Open**.
 - ▶ The Soft IP will be installed into a folder in the user's personal directory. For example: C:/Users/<username>/RadiantIPLocal/<IP_name>.
 - ▶ The Soft IP will be added into the IP Catalog.

See Also

- ▶ ["IP Packager" on page 135](#)
- ▶ ["Running IP Packager and Viewing Documentation" on page 135](#)

SEI Editor

SEI (Soft Error Injection) Editor allows you to generate single-bit errors, insert them into a bitstream, and detect them for analysis, simulating the effect of radiation damage on the device's configuration memory.

SEI Editor is available after you have placed and routed your design and generated a bitstream.

Common Tasks

The Radiant software gathers the many FPGA implementation tools into one central design environment. This gives you common controls for active tools, and it provides shared data between views.

Controlling Tool Views

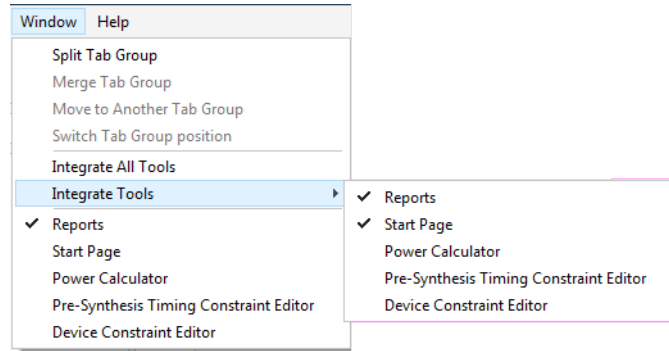
Tool views are highly configurable in the Radiant software environment. You can detach a tool view to work with it as a separate window, or create tab groups to display two views side-by-side.

Detaching and Attaching a Tool View

Each integrated tool view contains a Detach button  in the upper-right corner that allows you to work with the tool view as a separate window.

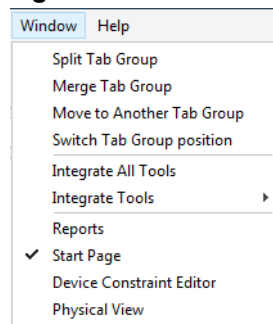
After a tool view is detached, the Detach button changes to an Attach button , which reintegrates the view into the Radiant main window.

You can detach as many tool views as desired. The Window menu keeps track of all open tool views and allows you to reintegrate one or all of them with the main window or detach any of them. Those that are already integrated are displayed with a check mark.

Figure 77: Window Integrate Tools Menu

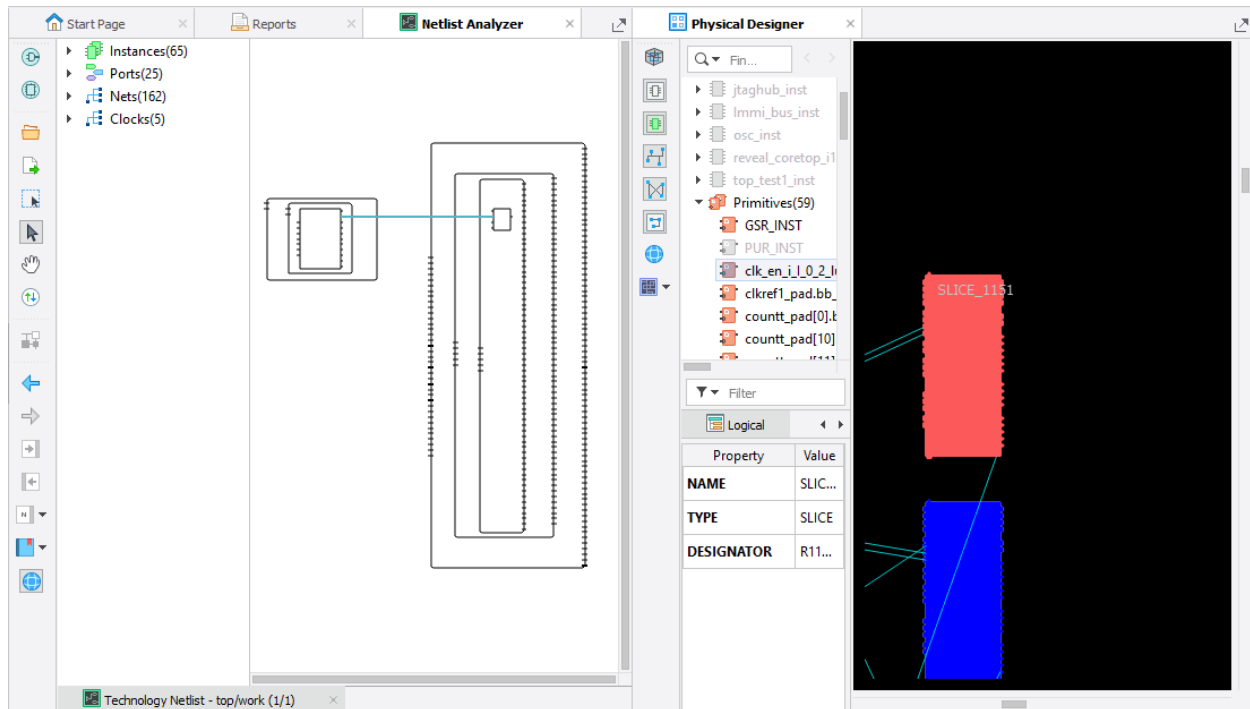
Tab Grouping

The Radiant software allows you to split one or more active tools into a separate tab group. Use the Window menu or the toolbar buttons to create the tab group and control the display.

Figure 78: Tab Controls in View Menu

The Split Tab Group command separates the currently active tool into a separate tab group. Having two separate tab groups enables you to work with two tool views side-by-side. This is especially useful for dragging and dropping to make constraint assignments; for example, dragging a port from Netlist View to Package View in order to assign it to a pin.

Having two separate tab groups is also useful for examining the same data element in two different views, such as the Netlist Analyzer and Physical Designer layouts.

Figure 79: Split Tab Group with Side-by-Side Layout Views

Move an active tool view from one tab group to another by dragging and dropping it, or you can use the Move to Another Tab Group command.

To switch the positions of the two tab groups, click the Switch Tab Group Position command.

To merge the split tab group back into the main group, click the Merge Tab Group command.

Using Zoom Controls

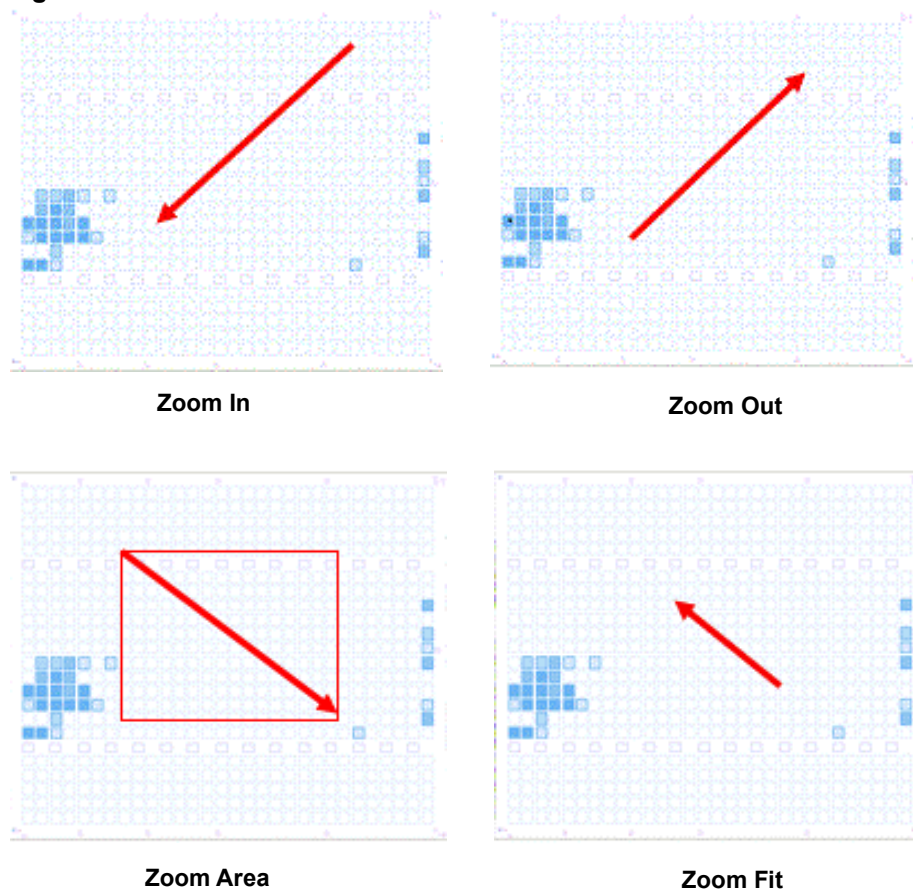
The Radiant software includes display zoom controls in the View toolbar. There are controls for increasing or reducing the scale of the view, fitting the display contents to the window view area, and fitting a selected area or object to the window view area.

Use the mouse to quickly zoom in, out or pan graphical views, such as Placement Mode and Routing Mode, by doing the following, as shown in Figure 80:

- ▶ **Zoom In:** press and hold the left mouse button while dragging the mouse down from upper right to left to zoom in.
- ▶ **Zoom Out:** press and hold the left mouse button while dragging the mouse up from lower left to right to zoom out.
- ▶ **Zoom To:** press and hold the left mouse button while dragging the mouse down from upper left to right to zoom into the box created.

- ▶ **Zoom Fit:** press and hold the left mouse button while dragging the mouse up from lower right to left to reset the diagram so it fits on screen.
- ▶ **Pan:** Click **Pan** () and drag the mouse in any direction to move the diagram, or press and hold **Ctrl** and drag the mouse.

Figure 80: Zoom with Mouse



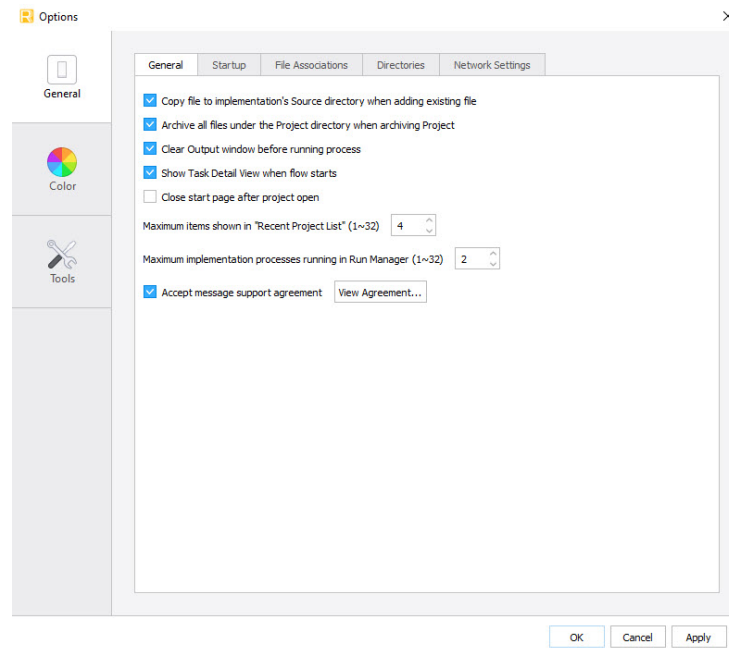
Displaying Tool Tips

When you place the cursor over a graphical element in a tool view, a tool tip appears with information on the element. The same information displayed in the tool tip will also be temporarily shown in the status bar on the lower left of the main window.

Setting Display Options

The Options dialog box, which is available from the Tools menu, enables you to specify general environment options as well as customize the display for the different tools. Tool options include selections for color, font, and other graphic elements.

Tool Options



For more information about Options, refer to [“Environment and Tool Options”](#) on page 234.

Command Line Reference Guide

This help guide contains information necessary for running the core design flow development from the command line. For tools that appear in the Radiant software graphical user interface, use Tcl commands to perform commands that are described in the [“Tcl Command Reference Guide” on page 1295](#).

Command Line Program Overview

Lattice FPGA command line programs can be referred to as the FPGA flow core tools. These are tools necessary to run a complete design flow and are used for tasks such as module generation, design implementation, design verification, and design configuration. This topic provides an overview of those tools, their functions, and provides links to detailed usage information on each tool.

Each command line program provides multiple options for specifying device information, applying special functions using switches, designating desired output file names, and [using command files](#). The programs also have particular default behavior often precludes the need for some syntax, making commands less complex. See [“Command Line General Guidelines” on page 1246](#) and [“Command Line Syntax Conventions” on page 1247](#).

To learn more about the applications, usage, and syntax for each command line program, click on the hyperlink of the command line name in the section below.

Note

Many of the command line programs described in this topic are run in the background when using the tools you run in the Radiant software environment. Please also note that in some cases, command line tools described here are used for earlier FPGA architectures only, are not always recommended for command line use, or are only available from the command line.

Design Implementation Using Command Line Tools The table below shows all of the command line tools used for various design functions, their graphical user interface counterparts, and provides functional descriptions of each.

Table 1: The Radiant Software Core Design and Tool Chart

Design Function	Command Line Tool	Radiant Process	Description
Core Implementation and Auxiliary Tools			
Encryption	Encryption	Encrypt Verilog/VHDL files	Encrypts the input HDL source file with provided encryption key.
Synthesis	SYNTHESIS	Synthesis Design	Runs input source files through synthesis based on Lattice Synthesis Engine options set in Strategies.
Synthesis	Synpwrap	Synthesis Design	Used to manage Synopsys Synplify Pro synthesis programs.
Mapping	MAP	Map Design	Converts a design represented in logical terms into a network of physical components or configurable logic blocks.
Placement and Routing	PAR	Place & Route Design	Assigns physical locations to mapped components and creates physical connections to join components in an electrical network.
Static Timing Analysis	Timing	Post-Synthesis Timing Report, MAP Timing Report, Place & Route Timing Report	Generates reports that can be used for static timing analysis.
Back Annotation	Backanno	Tool does not exist in the Radiant software interface as process but employed in file export.	Distributes the physical design information back to the logical design to generate a timing simulation file.

Table 1: The Radiant Software Core Design and Tool Chart

Design Function	Command Line Tool	Radiant Process	Description
Bitstream Generation	BITGEN	Bitstream	Converts a fully routed physical design into configuration bitstream data.
Device Programming	PGRCMD	Device Programming	Downloads data files to an FPGA device.
IP Packager	IP Packager	IP Packager	IP author select files from disks and pack them into one IPK file.

See Also ▶ [“Command Line Data Flow” on page 1245](#)

▶ [“Command Line General Guidelines” on page 1246](#)

▶ [“Command Line Syntax Conventions” on page 1247](#)

▶ [“Invoking Core Tool Command Line Programs” on page 1249](#)

Command Line Basics

This section provides basic instructions for running any of the core design flow development and tools from the command line.

Topics include:

▶ [“Command Line Data Flow” on page 1245](#)

▶ [“Command Line General Guidelines” on page 1246](#)

▶ [“Command Line Syntax Conventions” on page 1247](#)

▶ [“Setting Up the Environment to Run Command Line” on page 1248](#)

▶ [“Invoking Core Tool Command Line Programs” on page 1249](#)

▶ [“Invoking Core Tool Command Line Tool Help” on page 1249](#)

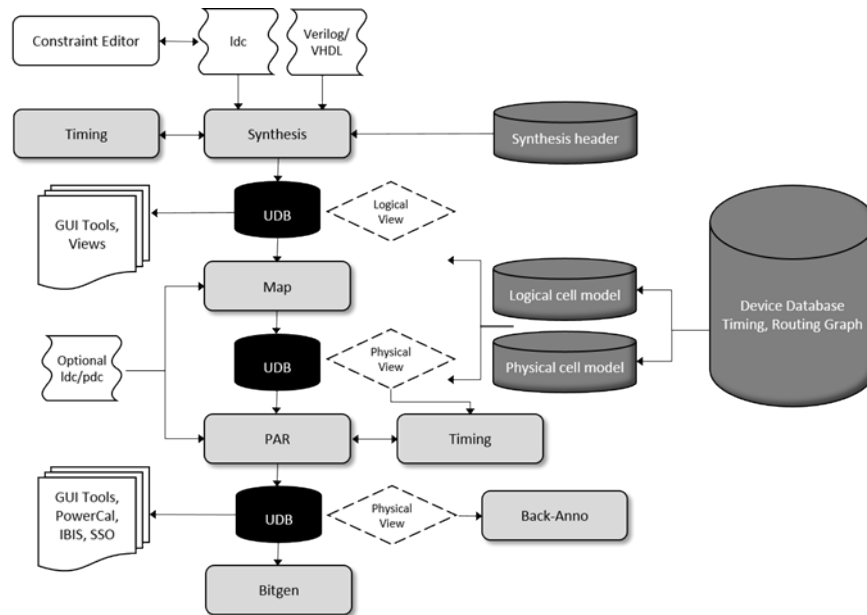
Command Line Data Flow

The following chart illustrates the FPGA command line tool data flow through a typical design cycle.

Command Line Tool Data Flow

See Also ▶ [“Command Line Reference Guide” on page 1243](#)

▶ [“Command Line General Guidelines” on page 1246](#)



Command Line General Guidelines

You can run the FPGA family Radiant software design tools from the command line. The following are general guidelines that apply.

- ▶ Files are position-dependent. Generally, they follow the convention [options] <infile> <outfile> (although order of <outfile> and <infile> are sometimes reversed). Use the **-h** command line option to check the exact syntax; for example, **par -h**.
- ▶ For any Radiant software FPGA command line program, you can invoke help on available options with the **-h** or **-help** command. See [“Invoking Core Tool Command Line Programs” on page 1249](#) for more information
- ▶ Command line options are entered on the command line in any order, preceded by a hyphen (-), and separated by spaces.
- ▶ Most command line options are case-sensitive and must be entered in lowercase letters. When an option requires an additional parameter, the option and the parameter must be separated by spaces or tabs (i.e., **-l 5** is correct, **-l5** is not).
- ▶ Options can appear anywhere on the command line. Arguments that are bound to a particular option must appear after the option (i.e., **-f <command_file>** is legal; **<command_file> -f** is not).
- ▶ For options that may be specified multiple times, in most cases the option letter must precede each parameter. For example, **-b romeo juliet** is not acceptable, while **-b romeo -b juliet** is acceptable.
- ▶ If you enter the FPGA family Radiant software application name on the command line with no arguments and the application requires one or more arguments (**par**, for example), you get a brief usage message consisting of the command line format string.

- ▶ For any Radiant software FPGA command line program, you can store program commands in a command file. Execute an entire batch of arguments by entering the program name, followed by the **-f** option, and the command file name. This is useful if you frequently execute the same arguments each time you execute a program or to avoid typing lengthy command line arguments. See [“Using Command Files” on page 1290](#).

See Also ▶ [“Command Line Reference Guide” on page 1243](#)

- ▶ [“Invoking Core Tool Command Line Tool Help” on page 1249](#)
- ▶ [“Command Line Syntax Conventions” on page 1247](#)
- ▶ [“Using Command Files” on page 1290](#)
- ▶ [“Command Line Data Flow” on page 1245](#)

Command Line Syntax Conventions

The following conventions are used when commands are described:

Table 2: Command Line Syntax Conventions

Convention	Meaning
()	Encloses a logical grouping for a choice between sub-formats.
[]	Encloses items that are optional. (Do not type the brackets.) Note that <infile[.udb]> indicates that the .udb extension is optional but that the extension must be UDB.
{ }	Encloses items that may be repeated zero or more times.
	Logical OR function. You must choose one or a number of options. For example, if the command syntax says pan up down right left you enter pan up or pan down or pan right or pan left .
< >	Encloses a variable name or number for which you must substitute information.
, (comma)	Indicates a range for an integer variable.
- (dash)	Indicates the start of an option name.
:	The bind operator. Binds a variable name to a range.
bold text	Indicates text to be taken literally. You type this text exactly as shown (for example, “Type autoroute -all -i 5 in the command area.”) Bold text is also used to indicate the name of an EPIC command, a Linux command, or a DOS command (for example, “The playback command is used to execute the macro you created.”).

Table 2: Command Line Syntax Conventions

Convention	Meaning
Italic text or text enclosed in angle brackets <>	Indicates text that is not to be taken literally. You must substitute information for the enclosed text. Italic text is also used to show a file directory path, for example, "the file is in the /cd/data/Radiant directory").
Monospace	Indicates text that appears on the screen (for example, "File already exists.") and text from Linux or DOS text files. Monospace text is also used for the names of documents, files, and file extensions (for example, "Edit the autoexec.bat file"

See Also ▶ ["Command Line Reference Guide" on page 1243](#)

▶ ["Command Line General Guidelines" on page 1246](#)

▶ ["Invoking Core Tool Command Line Tool Help" on page 1249](#)

▶ ["Using Command Files" on page 1290](#)

Setting Up the Environment to Run Command Line

For Windows The environments for both the Radiant Tcl Console window or Radiant Standalone Tcl Console window (pnmainc.exe) are already set. You can start entering Tcl tool command or core tool commands in the console and the software will perform them.

When running the Radiant software from the Windows command line (via cmd.exe), you will need to add the following values to the following environment variables:

- ▶ PATH includes, for 64-bit

```
<Install_directory>\bin\nt64;<Install_directory>\ispfpga\bin\nt64
```

Example <Install_directory>:

```
c:\lsc\radiant\1.0\bin\nt64;c:\lsc\radiant\1.0\ispfpga\bin\nt64
```

- ▶ FOUNDRY includes

```
set FOUNDRY= <Install_directory>\ispfpga
```

For Linux On Linux, the Radiant software provides a similar standalone Tcl Console window (radiantc) that sets the environment. The user can enter Tcl commands and core tool commands in it.

If you do not use the Tcl Console window, you need to run "bash" to switch to BASH" first, then run the following command.

- ▶ For BASH (64-bit):

```
export bindir=<Install_directory>/bin/lin64
source $bindir/radiant_env
```

After setting up for either Windows or PC, you can run the Radiant software executable files directly. For example, you can invoke the Place and Route program by:

```
par test_map.ldb test_par.ldb
```

See Also ▶ [“Invoking Core Tool Command Line Programs” on page 1249](#)
▶ [“Invoking Core Tool Command Line Tool Help” on page 1249](#)

Invoking Core Tool Command Line Programs

This topic provides general guidance for running the Radiant software FPGA flow core tools. Refer to [“Command Line Program Overview” on page 1243](#) to see what these tools include and for further information.

For any the Radiant software FPGA command line programs, you begin by entering the name of the command line program followed by valid options for the program separated by spaces. Options include switches (**-f**, **-p**, **-o**, etc.), values for those switches, and file names, which are either input or output files. You start command line programs by entering a command in the Linux™ or DOS™ command line. You can also run command line scripts or command files.

See Table 2 on page 1247 for details and links to specific information on usage and syntax. You will find all of the usage information on the command line in the **Running FPGA Tools from the Command Line > Command Line Tool Usage** book topics.

See Also ▶ [“Command Line Reference Guide” on page 1243](#)
▶ [“Command Line Syntax Conventions” on page 1247](#)
▶ [“Invoking Core Tool Command Line Tool Help” on page 1249](#)
▶ [“Setting Up the Environment to Run Command Line” on page 1248](#)
▶ [“Using Command Files” on page 1290](#)

Invoking Core Tool Command Line Tool Help

To get a brief usage message plus a verbose message that explains each of the options and arguments, enter the FPGA family Radiant software application name on the command line followed by **-help** or **-h**. For example, enter **bitgen -h** for option descriptions for the **bitgen** program.

To redirect this message to a file (to read later or to print out), enter this command:

```
command_name -help | -h > filename
```

The usage message is redirected to the filename that you specify.

For those FPGA family Radiant software applications that have architecture-specific command lines (e.g., iCE UltraPlus), you must enter the application name, **-help** (or **-h**), and the architecture to get the verbose usage message specific to that architecture. If you fail to specify the architecture, you get a message similar to the following:

Use '[<apname> -help <architecture>](#)' to get detailed usage for a particular architecture.

See Also ▶ [“Command Line Reference Guide” on page 1243](#)

- ▶ [“Command Line Data Flow” on page 1245](#)
- ▶ [“Command Line General Guidelines” on page 1246](#)
- ▶ [“Command Line Syntax Conventions” on page 1247](#)
- ▶ [“Setting Up the Environment to Run Command Line” on page 1248](#)
- ▶ [“Using Command Files” on page 1290](#)

Command Line Tool Usage

This section contains usage information of all of the command line tools and valid syntax descriptions for each.

Topics include:

- ▶ [“Running cmpl_libs.tcl from the Command Line” on page 1251](#)
- ▶ [“Running HDL Encryption from the Command Line” on page 1253](#)
- ▶ [“Running SYNTHESIS from the Command Line” on page 1260](#)
- ▶ [“Running Postsyn from the Command Line” on page 1266](#)
- ▶ [“Running MAP from the Command Line” on page 1267](#)
- ▶ [“Running PAR from the Command Line” on page 1269](#)
- ▶ [“Running Timing from the Command Line” on page 1275](#)
- ▶ [“Running Backannotation from the Command Line” on page 1278](#)
- ▶ [“Running Bit Generation from the Command Line” on page 1280](#)
- ▶ [“Running Various Utilities from the Command Line” on page 1286](#)
- ▶ [“Using Command Files” on page 1290](#)
- ▶ [“Using Command Line Shell Scripts” on page 1292](#)

Running cmpl_libs.tcl from the Command Line

The cmpl_libs.tcl command allows you to perform simulation library compilation from the command line.

The following information is for running cmpl_libs.tcl from the command line using the tclsh application. The supported TCL version is 8.5 or higher.

If you don't have TCL installed, or you have an older version, perform the following:

- ▶ Add *<Radiant_install_path>/tcltk/windows/BIN* to the front of your PATH, and
- ▶ For Linux users only, add *<Radiant_install_path>/tcltk/linux/bin* to the front of your LD_LIBRARY_PATH

Note

The default version of TCL on Linux could be older and may cause the script to fail. Ensure that you have TCL version 8.5 or higher.

To check TCL version, type:

```
tclsh
% info tclversion
% exit
```

For script usage, type:

```
tclsh cmpl_libs.tcl [-h|-help|]
```

Notes

- ▶ If Modelsim/Quarta is already in your PATH and preceding any Aldec tools, you can use:
'-sim_path .' for simplification; '.' will be added to the front of your PATH.
- ▶ Ensure the FOUNDRY environment variable is set. If the FOUNDRY environment variable is missing, then you need to set it before running the script. For details, refer to "Setting Up the Environment to Run Command Line" on page 96.
- ▶ To execute this script error free, Questasim 10.4e or a later 10.4 version, or Questasim 10.5b or a later version should be used for compilation.

Check log files under *<target_path>* (default = .) for any errors, as follows:

- ▶ For Linux, type:

```
grep -i error *.log
```
- ▶ For Windows, type:

```
find /i "error" *.log
```

Subjects included in this topic:

- ▶ Running `cmpl_lib.tcl`
- ▶ Command Line Syntax
- ▶ `cmpl_libs.tcl` Options
- ▶ Examples

Running `cmpl_lib.tcl` `cmpl_libs.tcl` allows you to compile simulation libraries from the command line.

Command Line Syntax `tclsh <Radiant_install_path>/cae_library/simulation/scripts/cmpl_libs.tcl -sim_path <sim_path> [-sim_vendor {mentor<default>}] [-device {ice40up|all<default>}] [-target_path <target_path>]`

`cmpl_libs.tcl` Options The table below contains all valid options for `cmpl_libs.tcl`

Table 3: `cmpl_libs.tcl` Command Line Options

Option	Description
<code>-sim_path <sim_path></code>	The <code>-sim_path</code> argument specifies the path to the simulation tool executable (binary) folder. This option is mandatory. Currently only Modelsim and Questa simulators are supported. NOTE: If <code><sim_path></code> has spaces, then it must be surrounded by <code>" "</code> . Do not use <code>{ }</code> .
<code>[-sim_vendor {mentor<default>}]</code>	The <code>-sim_vendor</code> argument is optional, and intended for future use. It currently supports only Mentor Graphics simulators (Modelsim / Questa).
<code>[-device {ice40up all<default>}]</code>	The <code>-device</code> argument specifies the Lattice FPGA device to compile simulation libraries for. This argument is optional, and the default is to compile libraries for all the Lattice FPGA devices.
<code>[-target_path <target_path>]</code>	The <code>-target_path</code> argument specifies the target path, where you want the compiled libraries and modelsim.ini file to be located. This argument is optional, and the default target path is the current folder. NOTES: (1) This argument is recommended if the current folder is the Radiant software's startup (binary) folder, or if the current folder is write-protected. (2) If <code><target_path></code> has spaces, then it must be surrounded by <code>" "</code> . Do not use <code>{ }</code> .

Examples This section illustrates and describes a few examples of Simulation Libraries Compilation Tcl command.

Example 1 The following command will compile all the Lattice FPGA libraries for Verilog simulation, and place them under the folder specified by `-target_path`. The path to Modelsim is specified by `-sim_path`.

```
tclsh <c:/lsc/radiant/1.0/>/cae_library/simulation/script/  
cmpl_libs.tcl -sim_path C:/modeltech64_10.0c/win64 -target_path  
c:/mti_libs
```

See Also ▶ [“Command Line Program Overview” on page 1243](#)

Running HDL Encryption from the Command Line

Radiant software allows you to encrypt the individual HDL source files.

The tool supports encryption of Verilog HDL and VHDL files. Per command's execution, single source file is encrypted.

The HDL file can be partially or fully encrypted depending on pragmas' placements within the HDL file. To learn more about pragmas' placements, see [“Defining Pragmas” on page 1255](#).

Running HDL Encryption Before running the utility, you need to annotate the HDL file with the appropriate pragmas. Additionally, you may need to create a key file containing an encryption key. To view the key file's proper formatting, see [“Key File” on page 1259](#).

Command Line Syntax `encrypt_hdl [-k <keyfile>] [-l language]
[-o <output_file>] <input_HDL_file>`

Encryption Option The table below contains descriptions of all valid options for HDL encryption.

Table 4: Encryption Command Line Options

Option	Description
-h(elp)	Print command help message.
-k <keyfile>	A key repository file. Depending on the location of the key, this option is required or optional. ▶ If the HDL source file contains no pragma, the key file is required. The tool encrypts the entire HDL file using all key sets declared in the key file. ▶ If the HDL source file contains only <code>begin</code> and <code>end</code> pragmas and no key pragmas, the key file is required. The tool encrypts the section between <code>begin</code> and <code>end</code> using all key sets declared in the key file. ▶ If the HDL source file contains the proper key pragma, <code>key_keyowner</code>, <code>key_keyname</code>, but the key file is missing the provided <code>key_public_key</code>, the tool fetches the first public key string matching the <code>key_keyowner</code> and <code>key_keyname</code> requirement in the key file If the HDL source file contains the proper definition of key, this option is not required. NOTE: If the same key name is defined in both, HDL source file and key.txt file, the key defined in HDL source file has a precedence.
-l <language>	Directive language, vhdl or verilog (default).
-o <output_file>	An encrypted HDL file. This is an optional field. If not defined during the encryption, the tool generates a new output file <code><input_file_name>_enc.v</code> .

Examples This section illustrates and describes a few examples of HDL encryption using Tcl command.

Example1: This example shows a successful encryption of HDL file with default options. It is assumed that key is properly defined in HDL file. Since output file name was not specified, the tool generates an output file `<file_name>_enc.v` in the same directory as the location of the input file.

```
> encrypt_hdl -k source/impl_1/keys.txt -o top.v top.v
Options:
  Key repository file:    source/impl_1/keys.txt
  Directive language:    <not specified>, use verilog as default
  Output file:           top.v
Processed 2 envelopes.
```

Example2: This example shows a successful encryption of HDL file by generating a new output file.

```
> encrypt_hdl -k source/impl_1/keys.txt -o remote_files/top_v1_en.v remote_files/sources/top_v1_part.
Options:
  Key repository file:    source/impl_1/keys.txt
  Directive language:    <not specified>, use verilog as default
  Output file:           remote_files/top_v1_en.v
Processed 2 envelopes.
```


Example3: This example shows unsuccessful HDL encryption due to a missing key file. To correct this issue, the user must either define the appropriate key file key.txt or annotated the HDL file with appropriate pragmas. To correct the issue, define the key either in key.txt file or directly in HDL source file.

```
> encrypt_hdl -o remote_files/sources/top_vl_part_en.v remote_files/sources/top_vl_part.v
Options:
  Key repository file:    <not specified>
  Directive language:    <not specified>, use verilog as default
  Output file:  remote_files/sources/top_vl_part_en.v
ERROR - remote_files/sources/top_vl_part.v at line 88: missing key.
```

NOTE

A key is always required in the encryption tool while key file is optional. If the complete key: key_keyowner, key_keyname, key_method, and key_public_key, is defined within HDL source file, key file is not required.

For specific steps and information on how to encrypt HDL files in the Radiant software, refer to the following section in the Radiant Software Help: **User Guides > Securing the Design.**

Defining Pragmas

Pragmas are used to specify the portion of the HDL source file that must be encrypted. Pragma' definition is compliant with IEEE 1735-2014 V1 standard.

Pragma syntax in Verilog HDL file:

```
`pragma protect <pragma's option>
```

Pragma syntax in VHDL file:

```
`protect <pragma's option>
```

Table 5: List of available Pragma Options

Name	Available Values	Description
version	1 (default)	Specifies the current Radiant software encryption version.
author	string	Specifies the file creator.
author_info	string	Additional information you would like to include in file.
encoding	base64	The output format of processed data.
begin		The start point for data obfuscation.
end		The end point for data obfuscation.
key_keyowner	string	The key creator.
key_keyname	string	The RSA key name to specify the private key.
key_method	rsa	The cryptographic algorithm used for key obfuscation.

Table 5: List of available Pragma Options

Name	Available Values	Description
key_public_key		The RSA public key file name.
data_method	aes128-cbc aes256-cbc (default)	The AES encryption data method.

To encrypt HDL source file, encryption version, encoding type, and key specific pragmas must be defined in the HDL source file by HDL designer; only the content within the pragmas is encrypted.

NOTE

Multiple key sets can be declared in a single key file.

Example of Verilog source file marked with Pragmas:

```
// 3 bit counter with asynchronous reset
module count(c,clk,rst);

input clk,rst;
output [2:0]c;
reg [2:0]c;

`pragma protect version=1
`pragma protect author="<Your Name>"
`pragma protect author_info="<Your info>"
`pragma protect key_keyowner="Lattice Semiconductor"
`pragma protect key_keyname="LSCC_RADIANT_2"
`pragma protect key_method="rsa"
`pragma protect key_public_key
MIIBIjANBgkqhkiG9w0BAQEFAAOCAQ8AMIIBCgKCAQEAE0EZKUUhbuB6vSsc70hQJ
iNAWJR5SunW/OwP/LFI71eAl3s9bOYE201Kdxbai+ndIeo8xFt2btsetUzuR6Srvh
xR2Sj9BbW1QToo2u8JfzD3X7AmRv1wKRX8708DPo4LDHZMA3qh0kfDDWkp2Eausf
LzE2cVxgq7fy/bDhUeN8xKQCSKJ7aguG6kOI6ROoZz211jzDLUQzhm2qYf8SpU1o
tD8/uw53wLfSuhR3MBOB++xcn2imvSLqdgHWuhX6CtZIx5CD4y8inCbcLy/0Qrf6
sdTN5Sag2OZhjeNdzmqSWqhL2JTDw+Ou2fWzhEd0i/HN0y4NMr6h9fNn8nqxRyE7
IwIDAQAB

//put a blank line above
`pragma protect data_method="aes256-cbc"

`pragma protect begin
always @(posedge clk or posedge rst)
begin
if (rst)
c = 3'b000;
else
c = c + 1;
end
`pragma protect end

endmodule
```

The encrypted file may contain multiple encrypted key sets.

Example of encrypted Verilog file:

```
// 3 bit counter with asynchronous reset
module count(c,clk,rst);

input clk,rst;
output [2:0]c;
reg [2:0]c;

//put a blank line above

`pragma protect begin_protected
`pragma protect version=1
`pragma protect author="<Your Name>"
`pragma protect author_info="<Your info>"
`pragma protect encrypt_agent="Radiant encrypt_hdl"
`pragma protect encrypt_agent_info="Radiant encrypt_hdl Version 1.0"

`pragma protect encoding=(enctype="base64", line_length=64, bytes=256)
`pragma protect key_keyowner="Lattice Semiconductor"
`pragma protect key_keyname="LSCC_RADIANT_2"
`pragma protect key_method="rsa"
`pragma protect key_block
gOatWm+1rVPboQIqaGf2gvMduH/vTXzyRA4C+tdNxpqNWeeTXrTF+uIa21e9Io7S
K6ce3BVAXydtADq5Wy50EjcHzUi3Ym1eyEdfVn2pxCp+3csuiZSeNgbtYutonZjh
8ReQYzPqKt6fZvhZ1AqHiuuZhFUsZQFXqnT8IW4dQ7HWudREzx6jB7h+7vI+wJvH
N5kZMiHBFGRhiTePz+yDOQwFVvwITezEoS099I8MoRGW1lU9kb4/Kenk96MIqE3W
lKAiQiViIjXeWvLRqmOb0hNGRoOEYwjlY1pj9Gye1HoDC6czdyqOOWrunt//XNu
v/QtpeEe/co7c9arRtHbv==

`pragma protect data_method="aes256-cbc"
`pragma protect encoding=(enctype="base64", line_length=64, bytes=176)
`pragma protect data_block
z+c6t504d6NkyrL5x6j6/raeaQmg0v9xmQQVaj3oq45SAsUIhGaihFVt+sS2kbNJ
/KBaqdcixAJHW8uRjtE/4nB+gZbVQsVnhRUlH/beksDnhdhWnNw/fcX6v6xptGwh
wQxsY+IjzT+pGC9rOEmAo2tndK63cWjHlg8hIZYnnw7yZAzv2OqylztSWFkdR9T4
mHclplFr96xb59oCRqbpQQgeESGgX9L1Yfd8j0ZXkSM=

`pragma protect end_protected

endmodule
```

Example of VHDL source file marked by Pragmas:

```
library ieee;
use ieee.std_logic_1164.all;
use ieee.std_logic_unsigned.all;
use ieee.std_logic_arith.all;

entity top_test is

    port(
        cout : out std_logic_vector(7 downto 0);
        reset : in  std_logic;
        clk  : in  std_logic
    );
end top_test;

architecture top_test_arch of top_test is

    signal count : std_logic_vector(7 downto 0);

begin

    `protect version=1

    `protect begin

        P100 : PROCESS(clk, reset)
        BEGIN
            IF (reset = '1') THEN
                count <= (others => '0');
            ELSE
                IF (clk'event and clk = '1') THEN
                    count <= count + 1;
                    cout <= count;
                END IF;
            END IF;
        END PROCESS;

    `protect end

end top_test_arch;
```

Example of encrypted VHDL file:

```

library ieee;
use ieee.std_logic_1164.all;
use ieee.std_logic_unsigned.all;
use ieee.std_logic_arith.all;

entity top_test is

    port(
        cout : out std_logic_vector(7 downto 0);
        reset : in  std_logic;
        clk   : in  std_logic
    );
end top_test;

architecture top_test_arch of top_test is

    signal count : std_logic_vector(7 downto 0);

begin

    `protect begin_protected
    `protect version=1
    `protect author="Lattice Semiconductor Corporation"
    `protect author_info="Lattice Semiconductor Corporation"
    `protect encrypt_agent="Radiant encrypt_hdl"
    `protect encrypt_agent_info="Radiant encrypt_hdl Version 1.0"

    `protect encoding=(enctype="base64", line_length=64, bytes=256)
    `protect key_keyowner="Lattice Semiconductor"
    `protect key_keyname="LSCC_RADIANT_2"
    `protect key_method="rsa"
    `protect key_block
    Vi2YLO+x6rdARfQ9Dy5nkhsQDsmm1w6mdX+BGfDMCLHH9OVHbd1SmeHawCz0jXY8
    ZfRK8VF/h1zOBmoosgY1pKd/Dpc5/4xkcEtnLzRbiXr1PhvF+tAFMMYr1FsqzJF/
    0yoej8yT6mayYbd5mcEr5rzBmX29HuPBZbYr1zia5IBpHZUa0mcwwhFnj5kz4OB
    bVsqIay7v8ECP41vNzoRKrsTYKBOhiTa6UoG08ut2E4d8wJIXNBgZ0uShYYzuOuv
    1goVgwaRtFWrpINXEmrZJPr/ikXRHTFzORpkDM7yNwGVtNJPMJ2aQde2w0i6EZWe
    1NnRF4K2HK0Oz1NRbffIjQ==

    `protect data_method="aes128-cbc"
    `protect encoding=(enctype="base64", line_length=64, bytes=272)
    `protect data_block
    B7yNcwT9w4purXSxU1ln4f3rs1psxSP1V3pnWYipDj6rQSA7wniBxcC/1aFebwKE
    fvbKngIIn7N+W/1Den3kjpuznIvLy5oV/GTANFP0cWt9rnRrDCM5CYtNWgaMEZu7
    o2QLlIfpCvwEgygI0R06NQ85frKo/jQLgOhf68+VpqFPozfrGAYI/YkEofh0foDH
    9Scy06grmJQCqtkqX2N3p6738N3iCFdKWJ6Udo5t+++AT3YYeZZ7bprDN941k/BkU
    YZij8fJx+govJUWQmSUJYNnt2Vyoac4hsevXsFRxm439ssQtP4vHHEEnraBSrHdVG
    DJn0GqFID+tpC67tE61U2Y/y118psFhZGs8jmuvmuv9yGk=

    `protect end_protected

end top_test_arch;

```

See Also ▶ [“Running HDL Encryption from the Command Line” on page 1253](#)

▶ [“Key File” on page 1259](#)

Key File

The key repository file defines the cryptographic public key used for RSA encryption. In Radiant software, the key file contains Lattice public key. Additionally, it may contain some of the common EDA vendors public keys.

The Lattice public key file key.txt is located at *<Radiant_installed_directory>/ispfpga/data/* folder. Aside of Lattice public key, the current version contains the public key for Synopsys, Aldec, and Cadence.

NOTE

If using Synplify Pro synthesis tool, both, the Lattice Public Key and the Synplify Pro Public Key must be defined in the key file. The Synplify Pro Public Key is used during the synthesis step to decrypt an encrypted design. The Lattice Public Key is used during the post-synthesis flow to decrypt an encrypted design.

A key file must contain properly declared pragmas such as `key_keyowner`, `key_keyname`, `key_method`, and `key_public_key` for each of the specified keys. The key value follows the `key_public_key` pragma.

The key file typically also contains the `data_method` pragma. It defines the algorithm used in data block encryption of HDL source file.

Example of a Key File:

```
// Use Verilog pragma syntax in this file
`pragma protect version=1
`pragma protect author="<Your Name>"
`pragma protect author_info="<Your info>"
`pragma protect key_keyowner="Lattice Semiconductor"
`pragma protect key_keyname="LSCC_RADIANT_2"
`pragma protect key_method="rsa"
`pragma protect key_public_key
MIIBIjANBgkqhkiG9w0BAQEFAAOCAQ8AMIIBCgKCAQEAEZKUUHbuB6vSsc7OhQJ
iNAWJR5unW/OWp/LFI71eAl3s9bOYE201Kdxba1+ndIeo8xFt2btzetUzuR6Srvh
xR2Sj9BbW1QToo2u8JfzD3X7AmRv1wKRX8708DPo4LDHZMA3qh0kfDDWkp2Eausf
LzE2cVxgq7fy/bDhUeN8xKQCSKJ7aguG6k0I6ROoZz211jzDLUQzhm2qYf8SpU1o
tD8/uw53wLfSuhR3MBOB++xcn2imvSLqdgHWuhX6CtZIx5CD4y8inCbclY/0Qrf6
sdTNS5Ag20ZhjeNdzmqSWqhL2JTDw+Ou2FWzhEd0i/HN0y4NM6h9fNn8nqxRyE7
IwIDAQAB

// Put a blank line above
// Add additional public keys below this line

`pragma protect data_method="aes256-cbc"

// End of File
```

See Also ▶ [“Running HDL Encryption from the Command Line” on page 1253](#)

▶ [“Defining Pragmas” on page 1255](#)

Running SYNTHESIS from the Command Line

The Lattice synthesis tool SYNTHESIS allows you to synthesize Verilog and VHDL HDL source files into netlists for design entry into the Radiant software environment. Based on your strategy settings you specify in the Radiant software, a synthesis project (.synproj) file is created and then used by SYNTHESIS using the -f option. The Radiant software translates strategy options into command line options described in this topic.

Verilog source files are passed to the program using the **-ver** option and VHDL source files are passed using the **-vhd** option. For mixed language designs the language type is automatically determined by SYNTHESIS based on the top module of the design. For IP design, you must also specify IP location (**-ip_dir**), IP core name (**-corename**), and encrypted RTL file name (**-ertl_file**).

Subjects included in this topic:

- ▶ Running SYNTHESIS
- ▶ Command Line Syntax
- ▶ SYNTHESIS Options
- ▶ Examples

Running SYNTHESIS SYNTHESIS will convert your input netlist (.v) file into a structural Verilog file that is used for the remaining mapping process.

- ▶ To run SYNTHESIS, type **synthesis** on the command line with valid options. A sample of a typical SYNTHESIS command would be as follows:

There are many command line options that give you control over the way SYNTHESIS processes the output file. Please refer to the rest of the subjects in this topic for more details. See examples.

Command Line Syntax **synthesis** [-a <arch>] [-p <device>] [-sp <performance_grade>] [-t <package_name>] [{-path <searchpath>}] [-top <top_module_name>] [-ver {<verilog_file.v>}] [-lib <libname>] [-vhd {<vhd_file.vhd/vhdl>}] [-udb <udb_file.udb>] [-hdl_param < param_name param_value >] [-vh2008] [-optimization_goal <area | timing | balanced (default)>] [-force_gsr <auto(default) | yes | no>] [-ramstyle <auto(default) | distributed | block_ram(EBR) | registers>] [-romstyle <auto(default) | logic | EBR>] [-output_edif <filename.edf>] [-output_hdl <filename.v>] [-sdc <sdc_file.ldc>] [-logfile <synthesis_logfile>] [-frequency <target_frequency (default 200.0MHz (ICE40))>] [-max_fanout <max_fanout (default 1000)>] [-bram_utilization <bram_utilization (default 100%)>] [-use_dsp <0|1(default)>] [-dsp_utilization <dsp_utilization (default 100%)>] [-fsm_encoding_style <auto(default) | one-hot | gray | binary>] [-resolve_mixed_drivers <0(default)|1>] [-fix_gated_clocks <0|1(default)>] [-use_carry_chain <0|1(default)>] [-carry_chain_length <chain_length>] [-use_io_insertion <0|1(default)>] [-use_io_reg <0|1|auto(default)>] [-resource_sharing <0|1(default)>] [-propagate_constants <0|1(default)>] [-remove_duplicate_regs <0|1(default)>] [-loop_limit <max_loop_iter_cnt (default 1950)>] [-twr_paths <timing_path_cnt>] [-dt] [-comp] [-syn] [-ifd] [-f <project_file_name>]

SYNTHESIS Options The table below contains descriptions of all valid options for SYNTHESIS.

Table 6: SYNTHESIS Command Line Options

Option	Description
-a <arch>	Sets the FPGA architecture. This synthesis option must be specified and if the value is set to any unsupported FPGA device architecture the command will fail.
-p <device>	Specifies the device type for the architecture (optional).
-f <proj_file_name>	Specifies the synthesis project file name (.synproj). The project file can be edited by the user to contain all desired command line options.
-t <package_name>	Specifies the package type of the device.
-path <searchpath>	Add searchpath for Verilog "include" files (optional).
-top <top_module_name>	Name of top module (optional, but better to have to avoid ambiguity).
-lib <lib_name>	Name of VHDL library (optional).
-vhd <vhd_file.vhd/vhdl>	Names of VHDL design files (must have, if language is VHDL or mixed language).
-ver <verilog_file.v>	Names of Verilog design files (must have, if language is Verilog, or mixed language).
-hdl_param <name, value>	Allows you to override HDL parameter pairs in the design file.
-optimization_goal <balanced (default) area timing>	The synthesis tool allows you to choose among the following optimization options: ▶ balanced balances the levels of logic. ▶ area optimizes the design for area by reducing the total amount of logic used for design implementation. ▶ timing optimizes the design for timing. The default setting depends on the device type. Smaller devices, such as ice40tp default to balanced.
-force_gsr <auto yes no>	Enables (yes) or disables (no) forced use of the global set/reset routing resources. When the value is auto, the synthesis tool decides whether to use the global set/reset resources.

Table 6: SYNTHESIS Command Line Options

Option	Description
-ramstyle <auto (default) distributed block_ram(EBR) registers>	Sets the type of random access memory globally to <i>distributed</i>, <i>embedded block RAM</i>, or <i>registers</i>. The default is auto which attempts to determine the best implementation, that is, synthesis tool will map to technology RAM resources (EBR/Distributed) based on the resource availability. This option will apply a syn_ramstyle attribute globally in the source to a module or to a RAM instance. To turn off RAM inference, set its value to registers. ▶ registers causes an inferred RAM to be mapped to registers (flip-flops and logic) rather than the technology-specific RAM resources. ▶ distributed causes the RAM to be implemented using the distributed RAM or PFU resources. ▶ block_ram (EBR) causes the RAM to be implemented using the dedicated RAM resources. If your RAM resources are limited, for whatever reason, you can map additional RAMs to registers instead of the dedicated or distributed RAM resources using this attribute. ▶ no_rw_check (Certain technologies only). You cannot specify this value alone. Without no_rw_check, the synthesis tool inserts bypass logic around the RAM to prevent the mismatch. If you know your design does not read and write to the same address simultaneously, use no_rw_check to eliminate bypass logic. Use this value only when you cannot simultaneously read and write to the same RAM location and you want to minimize overhead logic.

Table 6: SYNTHESIS Command Line Options

Option	Description
-romstyle <auto (default) logic EBR>	Allows you to globally implement ROM architectures using <i>dedicated</i>, <i>distributed ROM</i>, or a <i>combination of the two</i> (auto). This applies the syn_romstyle attribute globally to the design by adding the attribute to the module or entity. You can also specify this attribute on a single module or ROM instance. Specifying a syn_romstyle attribute globally or on a module or ROM instance with a value of: ▶ auto allows the synthesis tool to choose the best implementation to meet the design requirements for performance, size, etc. ▶ EBR causes the ROM to be mapped to dedicated EBR block resources. ROM address or data should be registered to map it to an EBR block. If your ROM resources are limited, for whatever reason, you can map additional ROM to registers instead of the dedicated or distributed RAM resources using this attribute. Infer ROM architectures using a CASE statement in your code. For the synthesis tool to implement a ROM, at least half of the available addresses in the CASE statement must be assigned a value. For example, consider a ROM with six address bits (64 unique addresses). The case statement for this ROM must specify values for at least 32 of the available addresses.
-output_hdl <filename.v>	Specifies the name of the output Verilog netlist file.
-sdc <sdc_file.ldc>	Specifies a Lattice design constraint (.ldc) file input.
-loop_limit <max_loop_iter_cnt (default 1950)>	Specifies the iteration limits for “for” and “while” loops in the user RTL for loops that have the loop index as a variable and not a constant. The higher the loop_limit, the longer the run time. Also, for some designs, a higher loop limit may cause stack overflow during some of the optimizations during compile/synthesis. The default value is 1950. Setting a higher value may cause stack overflow during some of the optimizations during synthesis.
-logfile <synthesis_logfile>	Specifies the name of the synthesis log file in ASCII format. If you do not specify a name, SYNTHESIS will output a file named synthesis.log by default.
-frequency <target_frequency (default 200.0MHz (ICE40))>	Specifies the target frequency setting. Default frequency value is 200.0 MHz.
-max_fanout <value>	Specifies maximum global fanout limit to the entire design at the top level. Default value is 1000 fanouts.
-bram_utilization <value>	Specifies block RAM utilization target setting in percent of total vacant sites. Default is 100 percent.

Table 6: SYNTHESIS Command Line Options

Option	Description
-fsm_encoding_style <auto one-hot gray binary>	Specifies One-Hot, Gray, or Binary style. The -fsm_encoding_style. Allows the user to determine which style is faster based on specific design implementation. Valid options are <i>auto</i> , <i>one-hot</i> , <i>gray</i> , and <i>binary</i> . The default value is <i>auto</i> , meaning that the tool looks for the best implementation.
-use_carry_chain <0 1>	Turns on (1) or off (0) carry chain implementation for adders. The 1 or true setting is the default.
-carry_chain_length <chain_length>	Specifies the maximum length of the carry chain.
-use_io_insertion <0 1>	Specifies the use of I/O insertion. The 1 or true setting is the default.
-use_io_reg <0 1 auto(default)>	Packs registers into I/O pad cells based on timing requirements for the target Lattice families. The value 1 enables and 0 disables (default) register packing. This applies it globally forcing the synthesis tool to pack all input, output, and I/O registers into I/O pad cells. NOTE: You can place the syn_useioff attribute on an individual register or port. When applied to a register, the synthesis tool packs the register into the pad cell, and when applied to a port, packs all registers attached to the port into the pad cell. The syn_useioff attribute can be set on a: ▶ top-level port ▶ register driving the top-level port ▶ lower-level port, only if the register is specified as part of the port declaration
-resource_sharing <0 1>	Specifies the resource sharing option. The 1 or true setting is the default.
-propagate_constants <0 1>	Prevents sequential optimization such as constant propagation, inverter push-through, and FSM extraction. The 1 or true setting is the default.
-remove_duplicate_regs <0 1>	Specifies the removal of duplicate registers. The 1 or true setting is the default.
-twr_paths <timing_path_cnt>	Specifies the number of critical paths.
-dt	Disables the hardware evaluation capability.
-udb <udb_file.udb>	
-ifd	Sets option to dump intermediate files. If you run the tool with this option, it will dump about 20 intermediate encrypted Verilog files. If you supply Lattice with these files, they can be decrypted and analyzed for problems. This option is good to for analyzing simulation issues.

Table 6: SYNTHESIS Command Line Options

Option	Description
-fix_gated_clocks <0 1(default)>	Allows you to enable/disable gated clock optimization. By default, the option is enabled.
-vh2008	Enables VHDL 2008 support.

Examples Following are a few examples of SYNTHESIS command lines and a description of what each does.

```
synthesis -a "ice40tp" -p itpa08 -t SG48 -sp "6" -mux_style Auto
-use_io_insertion 1
-sdc "C:/my_radiant_tutorial/impl1/impl1 ldc"
-path "C:/lsc/radiant/1.0/ispfpga/ice40tp/data" "C:/my_radiant_tutorial/impl1"
"C:/my_radiant_tutorial"
-ver "C:/my_radiant_tutorial/impl1/source/LED_control.v"
"C:/my_radiant_tutorial/impl1/source/spi_gpio.v"
"C:/my_radiant_tutorial/impl1/source/spi_gui_led_top.v"
-path "C:/my_radiant_tutorial"
-top spi_gui_led_top
-output_hdl "LEDtest_impl1.vm"
```

See Also ▶ [“Command Line Program Overview” on page 1243](#)

Running Postsyn from the Command Line

The Postsyn process converts synthesized VM and integrates IPs into a completed design in UDB format for the remaining mapping process.

Command Line Syntax `postsyn [-w] [-a <architecture>] [-p <device>] [-t <package>] [-sp <performance>] [-ldc <ldc_file>] [-iplist <iplist_file>] [-o <output.ldb>] [-keeprtl] [-top ] <input.vm>`

Table 7:

Option	Description
-h(elp)	Print command help message.
-w	Overwrite output file.
-a	Target architecture name.
-p	Target device name.
-t	Target package name.
-sp	Target performance grade.
-oc	Target operating condition: commercial industrial automotive.
-ldc	Load LDC file.
-iplist	Load IP list file.
-o	Output UDB file.
-keeprtl	Keep RTL view if it exists in UDB file.
-top	Indicate that the input is for the top design.
<input.vm>	Input structural Verilog file.

See Also ▶ [“Command Line Program Overview” on page 1243](#)

Running MAP from the Command Line

The **Map Design** process in the Radiant software environment can also be run through the command line using the **map** program. The **map** program takes an input database (.udb) file and converts this design represented as a network of device-independent components (e.g., gates and flip-flops) into a network of device-specific components (e.g., PFUs, PFFs, and EBRs) or configurable logic blocks in the form of a Unified Database (.udb) file.

Subjects included in this topic:

- ▶ Running MAP
- ▶ Command Line Syntax
- ▶ MAP Options
- ▶ Examples

Running MAP MAP uses the database (.udb) file that was the output of the **Synthesis** process and outputs a mapped Unified Database (.udb) file with constraints embedded.

- ▶ To run MAP, type **map** on the command line with, at minimum, the required options to describe your target technology (i.e., architecture, device, package, and performance grade), the input .udb along with the input .ldc file. The output .udb file specified by the **-o** option. That additional physical constraint file (*.pdc) can be applied optionally. A sample of a typical MAP command would be as follows:

```
map counter_impl1_syn.udb impl1.pdc -o counter_impl1.udb
```

Note

The **-a** (architecture) option is not necessary when you supply the part number with the **-p** option. There is also no need to specify the constraint file here, but if you do, it must be specified after the input .udb file name. The constraint file automatically takes the name "**output**" in this case, which is the name given to the output .udb file. If the output file was not specified with the **-o** option as shown in the above case, **map** would place a file named input.udb into the current working directory, taking the name of the input file. If you specify the input.ldc file and it is not there, map will error out.

There are many command line options that give you control over the way MAP processes the output file. Please refer to the rest of the subjects in this topic for more details.

Command Line Syntax **map** [-h <arch>] <infile[.udb]> [<options>]

MAP Options The table below contains descriptions of all valid options for MAP.

Table 8: MAP Command Line Options

Option	Description
-h <arch>	Displays all of the available MAP command options for mapping to the specified architecture.
<infile[.udb]>	Specifies the output design file name in .udb format. The .udb extension is optional.
-inferGSR	GSR inferencing if applicable.
-o <name[.udb]>	Optional output design file .udb.
-mp <name[.mrp]>	Optional report file (.mrp).
-xref_sig	Report signal cross reference for renamed signals.
-xref_sym	Report symbol cross reference for renamed symbols.
-u	Unclip unused instances.

Examples Following are some examples of MAP command lines and a description of what each does.

Example 1 The following command maps an input database file named mapped.udb and outputs a mapped Unified Database file named mapped.udb.

```
map counter_impl1_syn.udb impl1.pdc -o counter_impl1.udb
```

See Also ▶ [“Command Line Data Flow” on page 1245](#)

▶ [“Command Line Program Overview” on page 1243](#)

Running PAR from the Command Line

The **Place & Route Design** process in the Radiant software environment can also be run through the command line using the **par** program. The **par** program takes an input mapped Unified Database (.udb) file and further places and routes the design, assigning locations of physical components on the device and adding the inter-connectivity, outputting a placed and routed .udb file.

The Implementation Engine multi-tasking option available in Linux is explained in detail here because the option is not available for PCs.

Subjects included in this topic:

- ▶ Running PAR
- ▶ Command Line Syntax
- ▶ General Options
- ▶ Placement Options
- ▶ Routing Options
- ▶ PAR Explorer (-exp) Options
- ▶ Examples
- ▶ PAR Multi-Tasking Options

Running PAR PAR uses your mapped Unified Database (.udb) file that were the outputs of the **Map Design** process or the **map** program. With these inputs, **par** outputs a new placed-and-routed .udb file, a PAR report (.par) file, and a PAD (specification (.pad) file that contains I/O placement information.

- ▶ To run PAR, type **par** on the command line with at minimum, the name of the input .udb file and the desired name of the output .udb file. Design constraints from previous stages are automatically embedded in the input .udb file, however the par program can accept additional constraints with either a .pdc or .sdc file. A sample of a basic PAR command would be as follows:

```
par input.udb output.udb
```

There are many command line options that give you control over PAR. Please refer to the rest of the subjects in this topic for more details.

Command Line Syntax **par** [-w] [-n <iterations:0,100>] [-t <iteration:0,100>] [-stopzero] [-s <savecount:0,100>] [-m <nodelistfile>] [-cores <number of cores>] [-r] [-k] [-p] [-x] [-pack <density:0,100>] [-sp

<setupspeedgrade>] [-hsp <holdspeedgrade>] [-dh] [-hos] [-sort <method>]
 <infile> <outfile> [<pdcfile>]

Note

All filenames without special switches must be in the order <infile> <outfile> <pdcfile>. Options may exist in any order.

General Options

Table 9: General PAR Command Line Options

Option	Description
-f	Read par command line arguments and switches from file.
-w	Overwrite. Allows overwrite of an existing file (including input file).
-n	Number of iterations (seeds). Use "-n 0" to run until fully routed and a timing score of zero is achieved. Default: 1.
-t	Start at this placer cost table entry. Default is 1.
-stopzero	Stop running iterations once a timing score of zero is achieved.
-s	Save "n" best results for this run. Default: Save All.
-m	Multi task par run. File "<node list file>", contains a list of node names to run the jobs on.
-cores	Run multiple threads on the local machine. You can specify "<number of cores>" to run the jobs. For cases when the user specifies both -cores and -m with a valid node list file, PAR should apply both settings (merge). If the user repeats the host machine in the node list file, the settings in the node list file take precedence over the setting in -cores (for backwards compatibility).
-p	Don't run placement.
-r	Don't run router.
-k	Keep existing routing in input UDB file. Note: only meaningful when used with -p.
-x	Ignore timing constraints.
-pack	Set the packing density parameter. Default: auto.

Table 9: General PAR Command Line Options

Option	Description
-sp	Change performance grade for setup optimization. Default: Keep current performance grade.
-hsp	Change performance grade for hold optimization. Default: M.
-dh	Disable hold timing correction.
-hos	Prioritize hold timing correction over setup performance.
-sort	Set the sorting method for ranking multiple iterations. <method> "c" sorts by cumulative slack, "w" sorts by worst slack. Default: c.
<infile>	Name of input UDB file.
<outfile>	Name of output UDB file.

Table 10: PAR Placement Command Line Options

Option	Description
<pdcfile>	Name of optional constraint file. Note: the contents of <pdcfile> will overwrite all constraints saved in the input UDB file <infile>.

Examples Following are a few examples of PAR command lines and a description of what each does.

Example 1 The following command places and routes the design in the file input.udb and writes the placed and routed design to output.udb.

```
par input.udb output.udb
```

Example 2 The following command runs 20 place and route iterations. The iterations begin at cost table entry 5. Only the best 3 output design files are saved.

```
par -n 20 -t 5 -s 3 input.udb output.udb
```

Example 3 (Lattice FPGAs only) This is an example of **par** using the **-io** switch to generate .udb files that contain only I/O for viewing in the PAD Specification file for adjustment of Idc_set_location constraints for optimal I/O placement. You can display I/O placement assignments in the Radiant Spreadsheet View and choosing **View > Display IO Placement**.

```
par -io -w lev1bist.udb lev1bist_io.udb
```

Using the PAR Multi-Tasking (-m) Option This section provides information about environment setup, node list file creation, and step-by-step instructions for running the PAR Multi-tasking (-m) option from the command line. The PAR -m option allows you to use multiple machines (nodes) that are networked together for a multi-run PAR job, significantly reducing the total amount of time for completion. Before the multi-tasking option was developed, PAR could only run multiple jobs in a linear or serial fashion. The total time required to complete PAR was equal to the amount of time it took for each of the PAR jobs to run.

For example, the PAR command:

```
par -n 10 mydesign.ldb output.ldb
```

tells PAR to run 10 place and route passes (-n 10). It runs each of the 10 jobs consecutively, generating an output .ldb file for each job, i.e., output_par.dir/5_1.ldb, output_par.dir/5_2.ldb, etc. If each job takes approximately one hour, then the run takes approximately 10 hours.

Suppose, however, that you have five nodes available. The PAR Multi-tasking option allows you to use all five nodes at the same time, dramatically reducing the time required for all ten jobs.

To run the PAR multi-tasking option from the command line:

1. First generate a file containing a list of the node names, one per line as in the following example:

```
# This file contains a profile node listing for a PAR multi
# tasking job.
[machine1]
SYSTEM = linux
CORENUM = 2
[machine2]
SYSTEM = linux
CORENUM = 2
Env = /home/user/setup_multipar.lin
Workdir = /home/user/myworkdir
```

You must use the format above for the node list file and fill in all required parameters. Parameters are case insensitive. The node or machine names are given in square brackets on a single line.

The **System** parameter can take linux or pc values depending upon your platform. However, the PC value cannot be used with Linux because it is not possible to create a multiple computer farm with PCs. **Corenum** refers to the number of CPU cores available. Setting it to zero will disable the node from being used. Setting it to a greater number than the actual number of CPUs will cause PAR to run jobs on the same CPU lengthening the runtime.

The **Env** parameter refers to a remote environment setup file to be executed before PAR is started on the remote machine. This is optional. If the remote machine is already configured with the proper environment, this line can be omitted. To test to see if the remote environment is responsive to PAR commands, run the following:

```
ssh <remote_machine> par <par_option>
```

See the [System Requirements](#) section below for details on this parameter.

Workdir is the absolute path to the physical working directory location on the remote machine where PAR should be run. This item is also optional. If an account automatically changes to the proper directory after login, this line can be omitted. To test the remote directory, run the following,

```
ssh <remote_machine> ls <udb_file>
```

If the design can be found then the current directory is already available.

2. Now run the job from the command line as follows:

```
par -m nodefile_name -n 10 mydesign.udb output.udb
```

This runs the following jobs on the nodes specified.

```
Starting job 5_1 on node NODE1 at ...
Starting job 5_2 on node NODE2 at ...
Starting job 5_3 on node NODE3 at ...
Starting job 5_4 on node NODE4 at ...
Starting job 5_5 on node NODE5 at ...
```

As the jobs finish, the remaining jobs start on the nodes until all 10 jobs are complete. Since each job takes approximately one hour, all 10 jobs will complete in approximately two hours.

Note

If you attempt to use the multi-tasking option and you have specified only one placement iteration, PAR will disregard the **-m** option from the command and run the job in normal PAR mode. In this case you will see the following message:

```
WARNING - par: Multi task par not needed for this job. -m switch will be ignored.
```

System Requirements **ssh** must be located through the PATH variable. On Linux, the utility program's secure shell (**ssh**) and secure shell daemon (**sshd**) are used to spawn and listen for the job requests.

The executables required on the machines defined in the node list file are as follows:

- ▶ /bin/sh
- ▶ par (must be located through the PATH variable)

Required environment variable on local and remote machines are as follows:

- ▶ **FOUNDRY** (points at FOUNDRY directory structure must be a path accessible to both the machine from which the Implementation Engine is run and the node)
- ▶ **LM_LICENSE_FILE** (points to the security license server nodes)
- ▶ **LD_LIBRARY_PATH** (supports par path for shared libraries must be a path accessible to both the machine from which the Implementation Engine is run and the node)

To determine if everything is set up correctly, you can run the **ssh** command to the nodes to be used.

Type the following:

```
ssh <machine_name> /bin/sh -c par
```

If you get the usage message back on your screen, everything is set correctly. Note that depending upon your setup, this check may not work even though your status is fine.

If you have to set up your remote environment with the proper environment variables, you must create a remote shell environment setup file. An example of an ASCII file used to setup the remote shell environment would be as follows for ksh users:

```
export FOUNDRY=<install_directory>/ispfpga/bin/lin64
export PATH=$FOUNDRY/bin/lin64:$PATH
export LD_LIBRARY_PATH=$FOUNDRY/bin/lin:$LD_LIBRARY_PATH
64
```

For csh users, you would use the **setenv** command.

Screen Output When PAR is running multiple jobs and is not in multi-tasking mode, output from PAR is displayed on the screen as the jobs run. When PAR is running multiple jobs in multi-tasking mode, you only see information regarding the current status of the feature.

For example, when the job above is executed, the following screen output would be generated:

```
Starting job 5_1 on node NODE1
Starting job 5_2 on node NODE2
Starting job 5_3 on node NODE3
Starting job 5_4 on node NODE4
Starting job 5_5 on node NODE5
```

When one of the jobs finishes, this message will appear:

```
Finished job 5_3 on node NODE3
```

These messages continue until there are no jobs left to run.

See Also ▶ “Implementing the Design” in the Radiant Software Help

▶ [“Command Line Data Flow” on page 1245](#)

▶ [“Command Line Program Overview” on page 1243](#)

Running Timing from the Command Line

The **MAP Timing** and **Place & Route Timing** processes in the Radiant software environment can also be run through the command line using the **timing** program. Timing can be run on designs using the placed and routed Unified Design Database (.udb) and associated timing constraints specified in the design's (.ldc, .fdc, .sdc or .pdc) file or device constraints extracted from the design. Using these input files, **timing** provides static timing analysis and outputs a timing report file (.tw1/.twr).

Timing checks the delays in the Unified Design Database (.udb) file against your timing constraints. If delays are exceeded, Timing issues the appropriate timing error. See “Implementing the Design” in the Radiant Software Help and associated topics for more information.

Subjects included in this topic:

- ▶ Running Timing
- ▶ Command Line Syntax
- ▶ Timing Options
- ▶ Examples

Running Timing Timing uses your input mapped or placed-and-routed Unified Design Database (.udb) file and associated constraint file to create a Timing Report.

- ▶ To run Timing, type **timing** on the command line with, at minimum, the names of your input .udb and sdc files to output a timing report (.twr) file. A sample of a typical Timing command would be as follows:

```
timing design.udb (constraint is embedded in udb)
```

Note

The above command automatically generates the report file named design.twr which is based on the name of the .udb file.

There are several command line options that give you control over the way Timing generates timing reports for analysis. Please refer to the rest of the subjects in this topic for more details. See Examples.

Command Line Syntax **timing** <udb file name> [-sdc <sdc file name>] [-hld | -sethld] [-o <output file name>] [-v <integer>] [-endpoints <integer>] [-help]

Timing Options The following tables contain descriptions of all valid options for Timing.

Table 11: Compulsory Timing Command Line Options

Compulsory Option	Description
-db-file arg	design database file name.

Table 12: Optional Timing Command Line Options

Optional Option	Description
-endpoints arg (=10)	number of end points.
-u arg (=10)	number of unconstrained end points printed in the table.
-ports arg (=10)	number of top ports printed in the table.
-help	print the usage and exit.
-hld	hold report only.
-sp arg (=None)	Setup speed grade.
-hsp arg (=M)	Hold speed grade.
-rpt-file arg	timing report file name.
-o arg	timing report file name.
-alt_report	Diamond like report.
-report_sdc	Parsed file appears in report file.
-sdc-file arg	sdc file name.
-sethld	both setup and hold report.
-v arg (=10)	number of paths per constraint.
-time_through_async	Timer will time through async resets.
-iotime	compute the input setup/hold and clock to output delays of the FPGA.
-io_allspeed	Get worst IO results for all speed grades.
-pwrprd	Output clock information for PowerCalculator.
-nperend arg (=1)	Number of paths per end point.
-html	HTML format report.
-gui	Call from GUI.
-msg arg	Message log file.
-msgset arg	Message setting.
-io_allspeed	Get worst IO results for all speed grades.

Table 12: Optional Timing Command Line Options

Optional Option	Description
-dump_cdc	Dump CDC registers.
-dump_uncovered	Dump load pins of connections that are not covered.

Examples Following are a few examples of Timing command lines and a description of what each does.

Example 1 The following command verifies the timing characteristics of the design named design1.udb, generating a summary timing report. Timing constraints contained in the file group1.prf are the timing constraints for the design. This generates the report file design1.twr.

```
timing design1.udb (constraint is embedded in udb)
```

Example 2 The following command produces a file listing all delay characteristics for the design named design1.udb. Timing constraints contained in the file group1.prf are the timing constraints for the design. The file output.twr is the name of the verbose report file.

```
timing -v design1.udb -o output.twr
```

Example 3 The following command analyzes the file design1.udb and reports on the three worst errors for each constraint embedded in design1.udb and in additional timing constraint file (e.g. test.ldc). The report is called design1.twr.

```
timing -e 3 -sdc-file test.ldc design1.udb
```

Example 4 The following command analyzes the file design1.udb and produces a verbose report to check on hold times on any timing constraints embedded in design1.udb. With the output report file name unspecified here, a file using the root name of the .udb file (i.e., design1.twr) will be output by default.

```
timing -v -hld design1.udb
```

Example 5 The following command analyzes the file design1.udb and produces a summary timing report to check on both setup and hold times on any timing constraints embedded in design1.udb. With the output report file name unspecified here, a file using the root name of the .udb file (i.e., design1.twr) will be output by default.

```
timing -sethld design1.udb
```

See Also ▶ [“Command Line Program Overview” on page 1243](#)

▶ [“Command Line Data Flow” on page 1245](#)

Running Backannotation from the Command Line

The **Generate Timing Simulation Files** process in the Radiant software environment can also be run through the command line using the **backanno** program. The **backanno** program back-annotates physical information (e.g., net delays) to the logical design and then writes out the back-annotated design in the desired netlist format. Input to **backanno** is a Unified Database file (.udb) a mapped and partially or fully placed and/or routed design.

Subjects included in this topic:

- ▶ Running Backanno
- ▶ Command Line Syntax
- ▶ Backanno Options
- ▶ Examples

Running Backanno backanno uses your input mapped and at least partially placed-and-routed Unified Database (.udb) file to produce a back-annotated netlist (.v) and standard delay (.sdf) file. This tool supports all FPGA design architecture flows. Only Verilog netlist is generated.

- ▶ To run backanno, type **backanno** on the command line with, at minimum, the name of your input .udb file. A sample of a typical backanno command would be as follows:

```
backanno backanno.udb
```

Note

The above command back annotates backanno.udb and generates a Verilog file backanno.v and an SDF file backanno.sdf. If the target files already exist, they will not be overwritten in this case. You would need to specify the **-w** option to overwrite them.

There are several command line options that give you control over the way backanno generates back-annotated netlists for simulation. Please refer to the rest of the subjects in this topic for more details.

Command Line Syntax (Verilog) **backanno** [-w] [-pre <prfx>] [-sp <grade>] [-neg] [-pos] [-sup] [-min] [-x] [-fc] [-slice] [-slice0] [-slice1] [-noslice] [-t] [-dis []] [-m [<limit>]] [-u] [-i] [-nopur] [-l <libtype>] [-s <separator>] [-o <verilog<<.v>>]] [-d <delays[.sdf]>] [-gui] [-msg <msglogfile>] [-msgset <msgtypefile>] [<udbfile>]

Backanno Options The table below contains descriptions of all valid options for backanno.

Table 13: Backanno Options

Option	Description
-w	Overwrite the output files.
-sp <grade>	Override performance grade for backannotation.
-pre <prfx>	Prefix to add to module name to make them unique for multi-chip simulation.
-min	Override performance grade to minimum timing for hold check.
-dis 	Distribute routing delays by splitting the signal and inserting buffers. is the maximum delay (in ps) between each buffer (1000ps by default).
-m <limit>	Shortens the block names to a given character limit in terms of some numerical integer value.
-u	Add pads for top-level dangling nets.
-neg	Negative setup/hold delay support. Without this option, all negative numbers are set to 0 in SDF.
-pos	Write out 0 for negative setup/hold time in SDF for SC.
-x	Generate x for setup/hold timing violation.
-i	Create a buffer for each block input that has interconnection delay.
-nopur	Do not write PUR instance in the backannotation netlist. Instead, user has to instantiate it in a test bench.
<type>	Netlist type to write out.
<libtype>	Library element type to use.
<netfile>	The name of the output netlist file. The extension on this file will change depending on which type of netlist is being written. Use -h <type>, where <type> is the output netlist type, for more specific information.
<udb file>	Input file '.udb '.

Examples Following are a few examples of backanno command lines and a description of what each does.

Example 1 The following command back annotates design.udb and generates a Verilog file design.vo and an SDF file design.sdf. If the target files exist, they will be overwritten.

```
backanno -w design.udb
```

Example 2 The following command back annotates design.ldb and generates a Verilog file backanno.vo and an SDF file backanno.sdf. Any signal in the design that has an interconnection delay greater than 2000 ps (2 ns) will be split and a series of buffers will be inserted. The maximum interconnection delay between each buffer would be 2000 ps.

```
backanno -dis 2000 -o backanno design.ldb
```

Example 3 The following command re-targets backannotation to performance grade -2, and puts a buffer at each block input to isolate the interconnection delay (ends at that input) and the pin to pin delay (starts from that input).

```
backanno -sp 2 -i design.ldb
```

Example 4 The following command generates Verilog netlist and SDF files without setting the negative setup/hold delays to 0:

```
backanno -neg -n verilog design.ldb
```

See Also ▶ [“Command Line Program Overview” on page 1243](#)

▶ [“Command Line Data Flow” on page 1245](#)

Running Bit Generation from the Command Line

The **Bitstream** process in the Radiant software environment can also be run through the command line using the bit generation (**bitgen**) program. This topic provides syntax and option descriptions for usage of the **bitgen** program from the command line. The **bitgen** program takes a fully routed Unified Database (.ldb) file as input and produces a configuration bitstream (bit images) needed for programming the target device.

Subjects included in this topic:

- ▶ Running BITGEN
- ▶ Command Line Syntax
- ▶ BITGEN Options
- ▶ Examples

Running BITGEN BITGEN uses your input, fully placed-and-routed Unified Database (.ldb) file to produce bitstream (.bit, .msk, or .rpt) for device configuration.

- ▶ To run BITGEN, type **bitgen** on the command line with, at minimum, the **bitgen** command. There is no need to specify the input .ldb file if you run **bitgen** from the directory where it resides and there is no other .ldb present.

There are several command line options that give you control over the way BITGEN outputs bitstream for device configuration. Please refer to the rest of the subjects in this topic for more details.

iCE40UP Command Line Syntax `bitgen [-d] [-b] [-a] [-w] [-noebrinitq1] [-noebrinitq2] [-noebrinitq3] [-noheader] [-simbitmap] [-nvcm] [-freq <frequency_bit_setting>] [-spilowpower] [-warmboot] [-nvcmsecurity] {-g <setting_value>} <infile> [<outfile>]`

All devices except iCE40UP Command Line Syntax `bitgen [-d] [-w] [-m <format>] {-site <seirule>} {-site <seitype>} <infile> [<outfile>]`

BITGEN Options The table below contains descriptions of all valid options for BITGEN.

Note

Many BITGEN options are only available for certain architectures. Please use the **bitgen -h <architecture>** help command to see a list of valid bitgen options for the particular device architecture you are targeting.

Table 14: iCE40UP BITGEN Command Line Options

Option	Description
-d	Disable DRC.
-b	Produce .rpt file (ASCII form of binary).
-a	Produce .hex file.
-w	Overwrite an existing output file.
-freq <frequency_bit_setting>	Can setup different frequency: 0 = slow, 1 = medium, 2 = fast. Depending on the speed of external PROM, this options adjusts the frequency of the internal oscillator used by the iCE40UP device during configuration. This is only applicable when the iCE40UP device is used in SPI Master Mode for configuration.
-nvcm	Produce NVCM file.
-nvcmsecurity	Set security. Ensures that the contents of the Non-Volatile Configuration Memory (NVCM) are secure and the configuration data cannot be read out of the device.
-spilowpower	SPI flash low power mode. Places the PROM in low-power mode after configuration. This option is applicable only when the iCE40UP device is used as SPI Master Mode for configuration.
-warmboot	Enable warm boot. Enables the Warm Boot functionality, provided the design contains an instance of the WARMBOOT primitive.
-noheader	Don't include the bitstream header.
-noebrinitq0	Don't include EBR initialization for quadrant 0.
-noebrinitq1	Don't include EBR initialization for quadrant 1.

Table 14: iCE40UP BITGEN Command Line Options

Option	Description
-noebrinitq2	Don't include EBR initialization for quadrant 2.
-noebrinitq3	Don't include EBR initialization for quadrant 3.
-g NOPULLUP:ENABLED	No IO pullup. Removes the pullup on the unused I/Os, except Bank 3 I/Os which do not have pullup.
-h <architecture> or -help <architecture>	Display available BITGEN command options for the specified architecture. The bitgen -h command with no architecture specified will display a list of valid architectures.
<infile>	The input post-PAR design database file (.udb).
<outfile>	The output file. If you do not specify an output file, BITGEN creates one in the input file's directory. If you specify -b, the extension is .rpt. If you specify -a, the extension is .hex. If you specify -nvcn, the extension is .nvcn. Otherwise the extension is .bin. A report (.bgn) file containing all of BITGEN's output is automatically created under the same directory as the output file.

Table 15: BITGEN Command Line Options (all devices except iCE40UP)

Option	Description														
-d	Disable DRC.														
-w	Overwrite an existing output file.														
-m <format>	Create "mask" and "readback" files. Valid formats are: 0: Output files in ASCII 1: Output files in binary.														
-g <opt:val>	Set option to value, options are (First is default): <table> <tr> <td>CfgMode</td><td>Disable, Flowthrough, Bypass</td></tr> <tr> <td>RamCfg</td><td>Reset, NoReset</td></tr> <tr> <td>DONEPHASE</td><td>T3, T2, T1, T0</td></tr> <tr> <td>GOEPHASE</td><td>T1, T3, T2</td></tr> <tr> <td>GSRPHASE</td><td>T2, T3, T1</td></tr> <tr> <td>GWEPHASE</td><td>T2, T3, T1</td></tr> <tr> <td>ES</td><td>Yes, No.</td></tr> </table>	CfgMode	Disable, Flowthrough, Bypass	RamCfg	Reset, NoReset	DONEPHASE	T3, T2, T1, T0	GOEPHASE	T1, T3, T2	GSRPHASE	T2, T3, T1	GWEPHASE	T2, T3, T1	ES	Yes, No.
CfgMode	Disable, Flowthrough, Bypass														
RamCfg	Reset, NoReset														
DONEPHASE	T3, T2, T1, T0														
GOEPHASE	T1, T3, T2														
GSRPHASE	T2, T3, T1														
GWEPHASE	T2, T3, T1														
ES	Yes, No.														
-h <architecture> or -help <architecture>	Display available BITGEN command options for the specified architecture. The bitgen -h command with no architecture specified will display a list of valid architectures.														

Table 15: BITGEN Command Line Options (all devices except iCE40UP)

Option	Description
<infile>	The input post-PAR design database file (.udb).
<outfile>	The output file. If you do not specify an output file, BITGEN creates one in the input file's directory. If you specify -b, the extension is .rpt. If you specify -a, the extension is .hex. If you specify -nvc, the extension is .nvc. Otherwise the extension is .bin. A report (.bgn) file containing all of BITGEN's output is automatically created under the same directory as the output file.

Example The following command tells **bitgen** to overwrite any existing bitstream files with the **-w** option, prevents a physical design rule check (DRC) from running with **-d**, specifies a raw bits (.rpt) file output with **-b**. Notice how these three options can be combined with the **-wdb** syntax.

```
bitgen -wdb <design.udb>
```

See Also ▶ [“Command Line Program Overview” on page 1243](#)

▶ [“Command Line Data Flow” on page 1245](#)

Running Programmer from the Command Line

You can run Programmer from the command line. The **PGRCMD** command uses a keyword preceded by a hyphen for each command line option.

Running PGRCMD PGRCMD allows you to download data files to an FPGA device.

- ▶ To run PGRCMD, type **pgrcmd** on the command line with, at minimum, the **pgrcmd** command.

There are several command line options that give you control over the way PGRCMD programs devices. Please refer to the rest of the subjects in this topic for more details.

Command Line Syntax The following describes the PGRCMD command line syntax:

```
pgrcmd [-help] [-infile <input_file_path>] [-logfile <log_file_path>] [-cabletype <cable>]
```

-cabletype

```
lattice [ -portaddress < 0x0378 | 0x0278 | 0x03bc | 0x<custom address> > ]
```

```
usb [ -portaddress < EZUSB-0 | EZUSB-1 | ... | EZUSB-15 > ]
```

```
usb2 [ -portaddress < FTUSB-0 | FTUSB-1 | ... | FTUSB-15 > ]
```

TCK [0, 1, 2, 3, 4, 5, 6, 7, 8, 9, 10]

PGRCMD Options The following are PGRCMD options.

Help (Optional)

Option	Description
-help or -h	Displays the Programmer command line options.

Input File (required)

Option	Description
-infile <i>filename.xcf</i>	Specifies the chain configuration file (.xcf). If the file path includes spaces, enclose the path in quotes.

Log File (optional)

Option	Description
-logfile <i>logfilename.log</i>	Specifies the location of the Programmer log file.

Cable Type (optional)

Option	Description
-cabletype <i>lattice</i>	Lattice HW-DLN-3C parallel port programming cable (default).
-cabletype <i>usb</i>	Lattice HW-USBN-2A USB port programming cable.
-cabletype <i>usb2</i>	Lattice FHW-USBN-2B (FTDI) USB programming cable and any FTDI based demo boards.

Parallel Port Address (optional)

Option	Description
-portaddress <i>0x0378</i>	LPT1 parallel port (default)
-portaddress <i>0x0278</i>	LPT2 parallel port
-portaddress <i>0x03BC</i>	LPT3 parallel port
-portaddress <i>0x<custom address></i>	Custom parallel port address

This option is only valid with parallel port cables.

USB Port Address (optional)

Option	Description
-portaddress EZUSB-0 ... EZUSB-15	HW-USBN-2A USB cable number 0 through 15
-portaddress FTUSB-0 ... FTUSB-15	FTDI based demo board or FTDI USB2 cable number 0 through 15

Default is EZUSB-0 and FTUSB-0. Only valid with the USB port cables.

FTDI Based Demo Board or Cable Frequency Control (optional)

Option	Description
-TCK 0, 1, 2, 3, 4, 5, 6, 7, 8, 9, 10	0 = 30 Mhz 1 = 15 Mhz (default) 2 = 10 Mhz 3 = 7.5 Mhz 4 = 6 Mhz 5 = 5 Mhz 6 = 4 Mhz 7 = 3 Mhz 8 = 2 Mhz 9 = 1 Mhz 10 = 900 Khz

Calculation formula for USB-2B (2232H FTDI USB host chip): Frequency = 60 MHz / (1 + ClockDivider) *2

Calculation formula for USB-2B (2232D FTDI USB host chip): Frequency = 12 MHz / (1 + ClockDivider) *2

Only applicable for FTDI based demo boards or programming cable.

Return Codes

Code	Definition
0	Success
-1	Log file error
-2	Check configuration setup error
-3	Out of memory error
-4	NT driver error

Code	Definition
-5	Cable not detected error
-6	Power detection error
-7	Device not valid error
-8	File not found error
-9	File not valid error
-10	Output file error
-11	Verification error
-12	Unsupported operation error
-13	File name error
-14	File read error
-17	Build SVF file error
-18	Build VME file error
-19	Command line syntax error

Examples The following is a PGRCMD example.

```
pgrcmd -infile c:\test.xcf
```

See Also ▶ [“Command Line Data Flow” on page 1245](#)
[“Command Line Program Overview” on page 1243](#)

Running Various Utilities from the Command Line

The command line utilities described in this section are not commonly used by command line users, but you often see them in the auto-make log when you run design processes in the Radiant software environment. Click each link below for its function, syntax, and options.

Note

For information on commonly-used FPGA command line tools, see [“Command Line Basics” on page 1245](#).

Synpwrap

The **synpwrap** command line utility (wrapper) is used to manage Synplicity Synplify and Synplify Pro synthesis programs from the Radiant software environment processes: **Synplify Synthesize Verilog File** or **Synplify Synthesize VHDL File**.

The **synpwrap** utility can also be run from the command line to support a batch interface. For details on Synplify see the Radiant Software Help. The **synpwrap** program drives **synplify_pro** programs with a Tcl script file containing the synthesis options and file list.

Note

This section supersedes the “Process Optimization and Automation” section of the *Synplicity Synplify and Synplify Pro for Lattice User Guide*.

This section illustrates the use of the **synpwrap** program to run Synplify Pro for Lattice synthesis scripts from the command line. For more information on synthesis automation of Synplify Pro, see the “User Batch Mode” section of the *Synplicity Synplify and Synplify Pro for Lattice User Guide*.

If you use Synplify Pro, the Lattice OEM license requires that the command line executables **synplify_pro** be run by the Lattice “wrapper” program, **synpwrap**.

Command Line Syntax **synpwrap** [-log <log_file>] [-nolog] [-int <command_file>] [-gui] [-int <project_file> | -prj <project_file>] [-dyn] [-notoem] [-oem] [-notpro] [-pro] [-rem] [-scriptonly <script_file>] -e <command_file> -target <device_family> -part <device_name> [-options <arguments>]

Table 16: SYNPPWRAP Command Line Options

Option	Description
-log <log_file>	Specifies the log file name.
-nolog	Does not print out the log file after the process is finished.
-options <arguments>	Passes all arguments to Synplify/Pro. Ignores all other options except -notoem/-oem and -notpro/-pro. The -options switch must follow all other synpwrap options.
-prj <project_file>	Runs Synplify or Synplify Pro using an external prj Tcl file instead of the Radiant software command file.
-rem	Does not automatically include Lattice library files.
-e <command_file>	Runs the batch interface based on a Radiant software generated command file. The synpwrap utility reads <project>.cmd with its command line to obtain user options and creates a Tcl script file.
-gui	Invokes the Synplify or Synplify Pro graphic user interface.
-int <command_file>	Enables the interactive mode. Runs Synplify/Pro UI with project per command file.
-dyn	Brings the Synplify installation settings in the Radiant software environment.
-notoem	Does not use the Lattice OEM version of Synplify or Synplify Pro.

Table 16: SYNPPWRAP Command Line Options

Option	Description
-oem	Uses the Lattice OEM version of Synplify or Synplify Pro.
-notpro	Does not use the Synplify Pro version.
-pro	Uses the Synplify Pro version.
-target <device_family>	Specifies the device family name.
-part <device_name>	Specifies the device. For details on legal <device_name> values.
-scriptonly <script_file>	Generates the Tcl file for Synplify or Synplify Pro. Does not run synthesis.

Example Below shows a synppwrap command line example.

```
synppwrap -rem -e prep1 -target iCE40UP
```

See Also ▶ [“Command Line Program Overview” on page 1243](#)

▶ [“Command Line Data Flow” on page 1245](#)

IP Packager

The IP Packager (ippkg) tool can be run from the command line, allowing IP developers to select files from disks and pack them into one IPK file.

The process of IP packager is as following:

- ▶ IP author prepares metadata files, RTL files, HTML files, etc (all files of a Soft IP).
- ▶ IP Packager GUI provides UI for IP author to select files from the disk, and call IP Packaging engine to pack them into an IPK file.
- ▶ IP Packaging engine encrypts RTL files if IEEE P1735-2014 V1 pragmas are specified in RTL source

Command Line Syntax **ippkg [-h] (-metadata METADATA_FILE | -metadata_files METADATA_LIST_NAME) (-rtl RTL_FILE | -rtl_files RTL_LIST_NAME) [-encrypt FORCE_ENCRYPT_FILE | -encrypt_files FORCE_ENCRYPT_LIST_NAME] [-plugin PLUGIN_FILE] [-ldc LDC_FILE] [-fdc FDC_FILE] [-testbench TESTBENCH_FILE | -testbench_files TESTBENCH_LIST_NAME] [-driver_file DRIVER_FILE | -driver_files DRIVER_LIST_NAME] [-eval_file EVAL_FILE | -eval_files EVAL_LIST_NAME] (-help_file HELP_FILE | -help_files**

HELP_LIST_NAME) [-license_file LICENSE_FILE] [-o OUTPUT_ZIP_FILE]
[-key_file KEY_FILE]

Table 17: IPPKG Command Line Options

Option	Description
-metadata	The file name will be fixed to 'metadata.xml'.
-metadata_files	Location of the file which stores the metadata files. One line is a file path in specified file. Must have a file named metadata.xml.
-rtl	Specify the IP RTL file.
-rtl_files	One line is a file path in specified file.
-encrypt	Encrypt the whole RTL files.
-encrypt_files	One line is a file path in specified file.
-plugin	The file name will be fixed to 'plugin.py'.
-ldc	Specify the LDC file.
-fdc	Specify the FDC file.
-testbench	Specify the testbench file.
-testbench_files	One line is a file path in specified file.
-driver	Specify the driver file.
-driver_files	One line is a file path in specified file.
-eval	Specify the IP evaluation file.
-eval_files	One line is a file path in specified file.
-help_file	Specify the help file, must be <path>/introduction.html.
-help_files	One line is a file path in specified file.
-license_file	Specify the license file.
-o	Specify the output zip file.
-key_file	Specify the key file to encrypt the RTL files.

Example The following is an ippkg command line example:

```
ippkg -metadata c:/test/test.xml -rtl_files c:/test/rtl_list -  
help_file c:/test/introduction.html
```

See Also ▶ [“Command Line Program Overview” on page 1243](#)

▶ [“Command Line Data Flow” on page 1245](#)

ECO Editor

The ECO Editor tool can be run from the command line too.

ECO Editor is also able to dump the ECO TCL commands which user acted in GUI view without saving any UDB file.

In the meanwhile, we will have one non-GUI ECO engine tool, it accepts the dumped TCL script file with a UDB file and output a new UDB file.

User can set 'Place & Route design' milestone post-script by Tcl command `prj_set_postscript par <eco.tcl>`, then Radiant flow runs the ECO Tcl script automatically after running place & route.

Command Line Syntax `ecoc [-s <script_file>] [-o <output.udb>] <input.udb>]`

Table 18: ECO Editor Command Line Options

Option	Description
<code>-s</code>	ECO Tcl script file.
<code>-o</code>	Output UDB file.
<code><input.udb></code>	Input UDB file.

Example The following is an ecoc command line example:

```
ecoc -s mem.tcl ebr_test_impl_1.udb
```

See Also ▶ [“Command Line Program Overview” on page 1243](#)

▶ [“Command Line Data Flow” on page 1245](#)

Running Standalone Timing Analyzer from the Command Line

The Standalone Timing Analyzer can be run from the command line too.

Command Line Syntax `tavmain <udb file name> [<pdc file name>]`

Example The following is a tavmain command line example:

```
tavmain design.udb design.pdc
```

See Also ▶ [“Command Line Program Overview” on page 1243](#)

▶ [“Command Line Data Flow” on page 1245](#)

Using Command Files

This section describes how to use command files.

Creating Command Files The command file is an ASCII file containing command arguments, comments, and input/output file names. You can use

any text editing tool to create or edit a command file, for example, **vi**, **emacs**, **Notepad**, or **Wordpad**.

Here are some guidelines when you should observe when creating command files:

- ▶ Arguments (executables and options) are separated by space and can be spread across one or more lines within the file.
- ▶ Place new lines or tabs anywhere white space would otherwise be allowed on the Linux or DOS command line.
- ▶ Place all arguments on the same line, or one argument per line, or any combination of the two.
- ▶ There is no line length limitation within the file.
- ▶ All carriage returns and other non-printable characters are treated as space and ignored.
- ▶ Comments should be preceded with a # (pound sign) and go to the end of the line.

Command File Example This is an example of a command file:

```
#command line options for par for design mine.ldb
-a -n 10
-w
-l 5
-s 2 #will save the two best results
/home/users/jimbo/b/designs/mine.ldb
#output design name
/home/users/jimbo/b/designs/output.dir
#use timing constraint file
/home/users/jimbo/b/designs/mine.prf
```

Using the Command File The -f Option Use the **-f** option to execute a command file from any command line tool. The **-f** option allows you to specify the name of a command file that stores and then executes commonly used or extensive command arguments for a given FPGA command line executable tool. You can then execute these arguments at any time by entering the Linux or DOS command line followed by the name of the file containing the arguments. This can be useful if you frequently execute the same arguments each time you perform the command, or if the command line becomes too long. This is the recommended way to get around the DOS command line length limitation of 127 characters. (Equivalent to specifying a shell Options file.)

The **-f** indicates fast startup, which is performed by not reading or executing the commands in your `.cshrc` | `.kshrc` | `.shrc` (C-shell, Korn-shell, Bourne-shell) file. This file typically contains your path information, your environment variable settings, and your aliases. By default, the system executes the commands in this file every time you start a shell. The **-f** option overrides this process, discarding the 'set' variables and aliases you do not need, making the process much faster. In the event you do need a few of them, you can add them to the command file script itself.

Command File Usage Examples You can use the command file in two ways:

- ▶ To supply all of the command arguments as in this example:

```
par -f <command_file>
```

where:

<command_file> is the name of the file containing the command line arguments.

- ▶ To insert certain command line arguments within the command line as in the following example:

```
par -i 33 -f placeoptions -s 4 -f routeoptions design_i.ldb design_o.ldb
```

where:

placeoptions is the name of a file containing placement command arguments.

routeoptions is the name of a file containing routing command arguments.

Using Command Line Shell Scripts

This topic discusses the use of shell scripts to automate either parts of your design flow or entire design flows. It also provides some examples of what you can do with scripts. These scripts are Linux-based; however, it is also possible to create similar scripts called batch files for PC but syntax will vary in the DOS environment.

Creating Shell Scripts A Linux shell script is an ASCII file containing commands targeted to a particular shell that interprets and executes the commands in the file. For example, you could target Bourne Shell (**sh**), C-Shell (**csh**), or Korn Shell (**ksh**). These files also can contain comment lines that describe part of the script which then are ignored by the shell. You can use any text editing tool to create or edit a shell script, for example, **vi** or **emacs**.

Here are some guidelines when you should observe when creating shell scripts:

- ▶ It is recommended that all shell scripts with “#!” followed by the path and name of the target shell on the first line, for example, #!/bin/ksh. This indicates the shell to be used to interpret the script.
- ▶ It is recommended to specify a search path because oftentimes a script will fail to execute for users that have a different or incomplete search path. For example:

```
PATH=/home/usr/lsmith:/usr/bin:/bin; export PATH
```

- ▶ Arguments (executables and options) are separated by space and can be spread across one or more lines within the file.

- ▶ Place new lines or tabs anywhere white space would otherwise be allowed on the Linux command line.
- ▶ Place all arguments on the same line, or one argument per line, or any combination of the two.
- ▶ There is no line length limitation within the file.
- ▶ All carriage returns and other non-printable characters are treated as space and ignored.
- ▶ Comments are preceded by a # (pound sign) and can start anywhere on a line and continue until the end of the line.
- ▶ It is recommended to add exit status to your script, but this is not required.

```
# Does global timing meet acceptable requirement range?
if [ $timing -lt 5 -o $timing -gt 10 ]; then
    echo 1>&2 Timing \"$timing\" out of range
    exit 127
fi
etc...
# Completed, Exit OK
exit 0
```

Advantages of Using Shell Scripts Using shell scripts can be advantageous in terms of saving time for tasks that are often used, in reducing memory usage, giving you more control over how the FPGA design flow is run, and in some cases, improving performance.

Scripting with DOS Scripts for the PC are referred to as batch files in the DOS environment and the common practice is to ascribe a .bat file extension to these files. Just like Linux shell scripts, batch files are interpreted as a sequence of commands and executed. The COMMAND.COM or CMD.EXE (depending on OS) program executes these commands on a PC. Batch file commands and operators vary from their Linux counterparts. So, if you wish to convert a shell script to a DOS batch file or vice-versa, we suggest you find a good general reference that shows command syntax equivalents of both operating systems.

Examples The following example shows running design “counter” on below device package

Architecture: iCE40UP

Device: iCE40UP3K

Package: UWG30

Performance: Worst Case

Command 1: logic synthesis

```
synthesis -f counter_impl1_lattice.synproj
          which the *.synproj contains
-a "iCE40UP"
-p iCE40UP3K
-t UWG30
-sp "Worst Case"
-optimization_goal Area
```

```

-bram_utilization 100
-ramstyle Auto
-romstyle auto
-dsp_utilization 100
-use_dsp 1
-use_carry_chain 1
-carry_chain_length 0
-force_gsr Auto
-resource_sharing 1
-propagate_constants 1
-remove_duplicate_regs 1
-mux_style Auto
-max_fanout 1000
-fsm_encoding_style Auto
-twr_paths 3
-fix_gated_clocks 1
-loop_limit 1950
-use_io_reg auto
-use_io_insertion 1
-resolve_mixed_drivers 0
-sdc "impl1.ldc"
-path "C:/lsc/radiant/1.0/ispfpga/ice40tp/data" "impl1"
-ver "C:/lsc/radiant/1.0/ip/pmi/pmi.v"
-ver "count_attr.v"
-path "."
-top count
-udb "counter_impl1.udb"
-output_hdl "counter_impl1.vm"

```

Command 2: post synthesis process

```

postsyn -a iCE40UP -p iCE40UP3K -t UWG30 -sp Worst Case -top -
ldc counter_impl1.ldc -keeprtl -w -o counter_impl1.udb
counter_impl1.vm

```

Command 3: Mapper

```

map "counter_impl1_syn.udb" "impl1.pdc" -o "counter_impl1.udb"

```

Command 4: Placer and router

```

par -f "counter_impl1.p2t" "counter_impl1_map.udb"
"counter_impl1.udb"

```

Command 5: Timer

```

timing -sethld -v 10 -u 10 -endpoints 10 -nperend 1 -html -rpt
"counter_impl1_twr.html" "counter_impl1.udb"

```

Command 6: back annotation

```

backanno "counter_impl1.udb" -n Verilog -o
"counter_impl1_vo.vo" -w -neg

```

Command 7: bitstream generation

```

bitgen -w "counter_impl1.udb" -f "counter_impl1.t2b"

```


Tcl Command Reference Guide

The Radiant software supports Tcl (Tool Command Language) scripting and provides extended Radiant software Tcl commands that enable a batch capability for running tools in the Radiant software's graphical interface. The command set and the Tcl Console used to run it affords you the speed, flexibility and power to extend the range of useful tasks that the Radiant software tools are already designed to perform.

In addition to describing how to run the Radiant software's Tcl Console, this guide provides you with a reference for Tcl command line usage and syntax for all Radiant software point tools within the graphical user interface so that you can create command scripts, modify commands, or troubleshoot existing scripts.

About the Radiant software Tcl Scripting Environment The Radiant software development software features a powerful script language system. The user interface incorporates a complete Tcl command interpreter. The command interpreter is enhanced further with additional Radiant software-specific support commands. The combination of fundamental Tcl along with the commands specialized for use with the Radiant software allow the entire Radiant software development environment to be manipulated.

Using the command line tools permits you to do the following:

- ▶ Develop a repeatable design environment and design flow that eliminates setup errors that are common in GUI design flows
- ▶ Create test and verification scripts that allow designs to be checked for correct implementation
- ▶ Run jobs on demand automatically without user interaction

The Radiant software command interpreter provides an environment for managing your designs that are more abstract and easier to work with than using the core Radiant software engines. The Radiant software command interpreter does not prevent use of the underlying transformation tools. You

can use either the TCL commands described in this section or you can use the core engines described in the [“Command Line Reference Guide” on page 1243](#).

Additional References If you are unfamiliar with the Tcl language you can get help by visiting the Tcl/tk web site at <https://www.tcl.tk>. If you already know how to use Tcl, see the Tcl Manual supplied with this software. For information on command line syntax for running core tools that appear as Radiant software processes, such as synthesis, map, par, backanno, and timing, see the [“Command Line Reference Guide” on page 1243](#).

See Also ▶ [“Running the Tcl Console” on page 1296](#)

- ▶ [“Accessing Command Help in the Tcl Console” on page 1298](#)
- ▶ [“Radiant Software Tool Tcl Command Syntax” on page 1302](#)
- ▶ [“Creating and Running Custom Tcl Scripts” on page 1298](#)
- ▶ [“Accessing Command Help in the Tcl Console” on page 1298](#)
- ▶ [Tcl Manual](#)

Running the Tcl Console

The Radiant software TCL Console environment is made available for your use in multiple different ways. In order to take full advantage of the FPGA development process afforded by the Radiant software you must gain access to the Radiant Tcl Console user interface.

On Windows In Windows you can interact with the Tcl Console by any one of the following methods:

- ▶ To launch the Radiant software GUI from the Windows Start menu, choose **Start > Lattice Radiant Software (version_number) > Radiant Software**.

After the Radiant software loads, you can click on the **TCL Console** tab. With the **Tcl Console** tab active, you are able to start entering standard syntax TCL commands or the Radiant software-specific support commands.

- ▶ To launch the **TCL Console** independently from the Radiant software GUI from the Windows Start menu choose **Start > Lattice Radiant Software (version_number) > TCL Console**.

A Windows command interpreter will be launched that automatically runs the **TCL Console**.

- ▶ To run the interpreter from the command line, type the following:

```
c:/lsc/radiant/<version_number>/bin/nt64/pnmainc
```

The Radiant **TCL Console** is now available to run.

- ▶ To run the interpreter from a Windows PowerShell from the Windows Start menu choose **Start > Windows PowerShell > Windows PowerShell (x86)**.

A PowerShell interpreter window will open. At the command line prompt type the following:

```
c:/lsc/radiant/<version_number>/bin/nt64/pnmainc
```

The Radiant **TCL Console** is now available to run.

Note

The arrangement and location of each of the programs in the Windows Start menu will differ depending on the version of Windows you are running.

On Linux In Linux operating systems you can interact with the Tcl Console by one of the following methods:

- ▶ To launch the Radiant software GUI from the command line, type the following:

```
/usr/<user_name>/radiant/<version_number>/bin/lin64/radiant
```

The path provided assumes the default installation directory and that the Radiant software is installed. After the Radiant software loads you can click on the **TCL Console** tab. With the **TCL Console** tab active, you are able to start entering standard syntax TCL commands or the Radiant software specific support commands.

- ▶ To launch the **TCL Console** independently from the Radiant software GUI from the command line, type the following:

```
/usr/<user_name>/Radiant/<version_number>/bin/lin64/radiantc
```

The path provided assumes the default installation directory and that the Radiant software is installed, and that you have followed the Radiant software for Linux installation procedures. The Radiant **TCL Console** is now ready to accept your input.

The advantage of running the **TCL Console** from an independent command interpreter is the ability to directly pass the script you want to run to the Tcl interpreter. Another advantage is that you have full control over the Tk graphical environment, which allows you to create your own user interfaces.

See Also ▶ [“Running the Tcl Console” on page 1296](#)

- ▶ [“Radiant Software Tool Tcl Command Syntax” on page 1302](#)
- ▶ [“Creating and Running Custom Tcl Scripts” on page 1298](#)
- ▶ [“Accessing Command Help in the Tcl Console” on page 1298](#)
- ▶ [Tcl Manual](#)

Accessing Command Help in the Tcl Console

You can access command syntax help for all of the tools in the Tcl Console.

To access command syntax help in the Tcl Console:

1. In the prompt, type **help <tool_name>*** and press **Enter** as shown below:

```
help prj*
```

A list of valid command options appears in the Tcl Console.

2. In the Tcl Console, type the name of the command or function for more details on syntax and usage. For the prj tool, for example, type and enter the following:

```
prj_open
```

A list of valid arguments for that function appears.

Note

Although you can run the Radiant software's core tools such as synthesis, postsyn, map, par, and timing from the Tcl Console, the syntax for accessing help is different. For proper usage and syntax for accessing help for core tools, see the ["Command Line Reference Guide" on page 1243](#).

See Also ▶ ["Running the Tcl Console" on page 1296](#)

- ▶ ["Radiant Software Tool Tcl Command Syntax" on page 1302](#)
- ▶ ["Creating and Running Custom Tcl Scripts" on page 1298](#)
- ▶ ["Running Tcl Scripts When Launching the Radiant Software" on page 1301](#)
- ▶ [Tcl Manual](#)

Creating and Running Custom Tcl Scripts

This topic describes how to easily create Tcl scripts using the Radiant software's user interface and manual methods. FPGA design using Tcl scripts provides some distinct advantages over using the graphical user interface's lists, views and menu commands. For example, Tcl scripts allow you to do the following:

- ▶ Set the tool environment to exactly the same state for every design run. This eliminates human errors caused by forgetting to manually set a critical build parameter from a drop-down menu.
- ▶ Manipulate intermediate files automatically, and consistently on every run. For example, .vm file errors can be corrected prior to performing additional netlist transformation operations.
- ▶ Run your script automatically by using job control software. This gives you the flexibility to run jobs at any time of day or night, taking advantage of idle cycles on your corporate computer system.

Creating Tcl Scripts There are a couple of different methods you can use to create the Radiant software Tcl scripts. This section will discuss each one and provide step-by-step instructions for you to get started Tcl scripting repetitious Radiant software commands or entire workflows.

One method you have available is to use your favorite text editor to enter a sequence of the Radiant software Tcl commands. The syntax of each the Radiant software Tcl commands is available in later topics in this portion of the Help. This method should only be used by very experienced Radiant software Tcl command line users.

The preferred method is to let the Radiant software GUI assist you in getting the correct syntax for each Tcl command. When you interact with the Radiant software user interface each time you launch a *scriptable* process and the corresponding Radiant software Tcl command is echoed to the Tcl Console. This makes it much simpler to get the correct command line syntax for each Radiant software command. Once you have the fundamental commands executed in the correct order, you can then add additional Tcl code to perform error checking, or customization steps.

To create a Tcl command script in the Radiant software:

1. Start the Radiant software design software and close any project that may be open.
2. In the Tcl Console execute the custom **reset** command. This clears the Tcl Console command history.
3. Use the Radiant software graphical user interface to start capturing the basic command sequence. The Tcl Console echos the commands in its window. Start by opening the project for which you wish to create the TCL script. Then click on the processes in the Process bar to run them. For example, run these processes in their chronological order in the design flow:

- ▶ Synthesize Design
- ▶ Map Design
- ▶ Place & Route Design
- ▶ Export Files

4. In the Tcl Console window enter the command,

```
save_script <filename.ext>
```

The <filename.ext> is any file identifier that has no spaces and contains no special characters except underscores. For example, **myscript.tcl** or **design_flow_1.tcl** are acceptable save_script values, but **my\$script** or **my script** are invalid. The <filename.ext> entry can be preceded with a absolute or relative path. Use the "/" (i.e. forward slash) character to delimit the path elements. If the path is not specified explicitly the script is saved in the current working directory. The current working directory can be determined by using the TCL *pwd* command.

5. You can now use your favorite text editor to make any changes to the script you feel are necessary. Start your text editor, navigate to the

directory the *save_script* command saved the base script, and open the file.

Note

In most all cases, you will have to clean up the script you saved and remove any invalid arguments or any commands that cannot be performed in the Radiant software environment due to some conflict or exception. You will likely have to revisit this step later if after running your script you experience any run errors due to syntax errors or technology exceptions.

Sample Radiant software Tcl Script The following the Radiant software Tcl script shows a very simple script that opens a project, runs the entire design flow through the Place & Route process, then closes the project. A typical script will contain more tasks and will check for failure conditions. Use this simple example as a general guideline.

Figure 1: Simple Radiant software Script

```
prj_archive -dir "C:/my_radiant/counter" -extract "C:/lsccl/radiant/1.1/examples/counter.zip"
prj_run_par
prj_close
```

Running Tcl Scripts The Radiant software TCL scripts are run exclusively from the Radiant TCL Console. You can use either the TCL Console integrated into the Radiant software UI, or by launching the stand-alone TCL Console.

To run a Tcl script in the Radiant software:

1. Launch the Radiant software GUI, or the stand-alone TCL Console.
Open the Radiant software but do not open your project. If your project is open, choose **File > Close Project**.
2. If you are using the Radiant software main window, click the small arrow pane switch in the bottom of the Radiant software main window, and then click on the **Tcl Console tab** in the Output area at the bottom to open the console.
3. Use the TCL *source* command to load and run your TCL script. The *source* command requires, as it's only argument, the filename of the script you want to load and run. Prefix the script file name with any required relative or absolute path information. To run the example script shown in the previous section use:

```
source C:/lsccl/radiant/<version_number>/examples/counter/
myscript2.tcl
```

As long as there are no syntax errors or invalid arguments, the script will open the project, synthesize, map, and place-and-route the design. Once the design finishes it closes the project. If there are errors in the script, you will see the errors in red in the Tcl Console after you attempt to run it. Go back to your script and correct the errors that prevented the script from running.

- See Also** ▶ [“Running the Tcl Console” on page 1296](#)
- ▶ [“Radiant Software Tool Tcl Command Syntax” on page 1302](#)
 - ▶ [“Running Tcl Scripts When Launching the Radiant Software” on page 1301](#)
 - ▶ [Tcl Manual](#)

Running Tcl Scripts When Launching the Radiant Software

This topic describes how launch the Radiant software and automatically run Tcl scripts using a command line shell or the stand-alone Tcl console. Your Tcl script can be standard Tcl commands as well as the Radiant software-specific Tcl commands.

Refer to [“Creating and Running Custom Tcl Scripts” on page 1298](#) for more information on creating custom Tcl scripts.

To launch the Radiant software and run a Tcl script from a command line shell or the stand-alone Tcl console:

- ▶ Enter the following command:

On Windows:

```
pnmain.exe -t <tcl_path_file>
```

On Linux:

```
radiant -t <tcl_path_file>
```

Sample Radiant software Tcl Script The following Radiant software Tcl script shows a very simple script, running in Windows, that opens a project and runs the design flow through the MAP process. Use this simple example as a general guideline.

Figure 2: Simple Radiant Software Script

```
prj_open C:/test/iobasic_radiant/io1.rdf  
prj_run_map
```

The above example is saved in Windows as the file mytcl.tcl in the directory C:/test. By running the following command from either a DOS shell or the Tcl console in Windows, the Radiant software GUI starts, the project io1.rdf opens, and the MAP process automatically runs.

```
pnmain.exe -t c:/test/mytcl.tcl
```

- See Also** ▶ [“Running the Tcl Console” on page 1296](#)
- ▶ [“Radiant Software Tool Tcl Command Syntax” on page 1302](#)

- ▶ [“Creating and Running Custom Tcl Scripts” on page 1298](#)
- ▶ [Tcl Manual](#)

Radiant Software Tool Tcl Command Syntax

This part of the Tcl Command Reference Guide introduces the syntax of each of the Radiant software tools and provides you with examples to help you construct your own commands and scripts.

The Radiant software tries to make it easy to develop TCL scripts by mirroring the correct command syntax in the Tcl Console based on the actions performed by you in the GUI. This process works well for most designs, but there are times when a greater degree of control is required. More control over the build process is made available through additional command line switches. The additional switches may not be invoked by actions taken by you when using the GUI. This section provides additional information about all of the Tcl commands implemented in the Radiant software.

The Tcl Commands are broken into major categories. The major categories are:

- ▶ [Radiant Software Tcl Console Commands](#)
- ▶ [Radiant Software Timing Constraints Tcl Commands](#)
- ▶ [Radiant Software Physical Constraints Tcl Commands](#)
- ▶ [Radiant Software Project Tcl Commands](#)
- ▶ [Reveal Inserter Tcl Commands](#)
- ▶ [Reveal Analyzer Tcl Commands](#)
- ▶ [Power Calculator Tcl Commands](#)
- ▶ [Programmer Tcl Commands](#)
- ▶ [Engineering Change Order Tcl Commands](#)

Radiant Software Tcl Console Commands

The Radiant software Tcl Console provides a small number of commands that allow you to perform some basic actions upon the Tcl Console Pane. The Radiant software Tcl Console commands differ from the other Tcl commands provided in the Radiant software. This dtc program's general Tcl Console commands do not use the *dtc_* prefix in the command syntax as is the convention with other tools in the Radiant software.

Note

TCL Command Log is always listed after the project is closed. You can find it in the Reports section under Misc Report > TCL Command Log.

The following table provides a listing of all valid Radiant software Tcl Console-related commands.

Table 19: Radiant Software Tcl Console Commands

Command	Arguments	Description
clear	N/A	The <i>clear</i> command erases anything present in the Tcl Console pane, and prints the current <i>prompt</i> character in the upper left corner of the Tcl Console pane without erasing the command history.
history	N/A	The <i>history</i> command lists the command history in the Tcl Console that you executed in the current session. Every command entered into the Tcl Console, either by the GUI, or by direct entry in the Tcl Console, is recorded so that it can be recalled at any time. The command history list is cleared when a project is <i>opened</i> or when the Tcl Console <i>reset</i> command is executed.
reset	N/A	The <i>reset</i> command clears anything present in the Tcl Console pane, and erases all entries in the command line history. **It's only used in GUI Tcl console and not supported in stand-alone Tcl console.
save_script	<filename.ext>	Saves the contents of the command line history memory buffer into the script file specified. The script is, by default, stored into the current working directory. File paths using forward slashes used with an identifier are valid if using an absolute file path to an existing script folder. **It's only used in GUI Tcl console and not supported in stand-alone Tcl console.
set_prompt	<new_character>	The default prompt character in the Tcl Console is the "greater than" symbol or angle bracket (i.e., >). You can change this prompt character to some other special character such as a dollar sign (\$) or number symbol (#) if you prefer. **It's only used in GUI Tcl console and not supported in stand-alone Tcl console.

Radiant Software Tcl Console Command Examples This section illustrates and describes a few samples of Radiant Tcl Console commands.

Example 1 To save a script, you simply use the **save_script** command in the Tcl Console window with a name or file path/name argument. In the first example command line, the file path is absolute, that is, it includes the entire path. Here you are saving “myscript.tcl” to the existing current working directory. The second example creates the same “myscript.tcl” file in the current working directory.

```
save_script C:/lsc/radiant/myproject/scripts/myscript.tcl

save_script myscript.tcl
```

See [“Creating and Running Custom Tcl Scripts” on page 1298](#) for details on how to save and run scripts in the Radiant software.

Example 2 The following **set_prompt** command reassigns the prompt symbol on the command line as a dollar sign (\$). The default is an angle bracket or “greater than” sign (>).

```
set_prompt $
```

Example 3 The following **history** command will print all of the command history that was recorded in the current Tcl Console session.

```
history
```

Radiant Software Timing Constraints Tcl Commands

The following table provides a listing of all valid Radiant software Timing Constraints Tcl commands.

Table 20: Radiant Software Timing Constraints Tcl Commands

Command	Arguments	Description
create_clock	create_clock -period <period_value> [-name <clock_name>] [-waveform <edge_list>] [<port_list pin_list net_list>]	Create a named or virtual clock.
create_generated_clock	create_generated_clock [-name <clock_name>] -source <master_pin>[-edges <edge_list>] [- divide_by <factor>] [-multiply_by <factor>] [-duty_cycle <percent>] [-invert] [-add [<pin_list net_list port_list>]	Create a generated clock object.

Table 20: Radiant Software Timing Constraints Tcl Commands

Command	Arguments	Description
ldc_define_attribute	ldc_define_attribute -attr <attr_type> -value <attr_value> -object_type <object type> -object <object> [-disable] [-comment <comment>]	Set LSE synthesis attributes for given objects
set_clock_groups	set_clock_groups -group <clock_list> <-logically_exclusive -physically_exclusive -asynchronous>	Set clock groups.
set_clock_latency	set_clock_latency [-rise] [-fall] [-early -late] <source> <latency> <object_list>	Defines a clock's source or network latency
set_clock_uncertainty	set_clock_uncertainty [-setup] [-hold] [-from <clock>] [-to <clock>] <uncertainty> [<clock_list>]	Set clock uncertainty.
set_false_path	set_false_path [-from <port_list pin_list instance_list net_list clock_list>] [-to <port_list pin_list instance_list net_list clock_list>] [-through <port_list pin_list instance_list net_list>] [-rise_from <clock_list>] [-rise_to <clock_list>] [-fall_from <clock_list>] [-fall_to <clock_list>] [-comment string]	Define false path
set_input_delay	set_input_delay -clock <clock_name> [-clock_fall] [-max] [-min] [-add_delay] <delay_value> <port_list>	Set input delay on ports
set_load	set_load <capacitance> <objects>	Commands to set capacitance on ports

Table 20: Radiant Software Timing Constraints Tcl Commands

Command	Arguments	Description
set_max_delay	set_max_delay [-from <port_list pin_list instance_list net_list clock_list> [-to <port_list pin_list instance_list net_list clock_list> [-through <port_list pin_list instance_list net_list> [-rise_from <clock_list>] [-rise_to <clock_list> [-fall_from <clock_list>] [-fall_to <clock_list>] <delay_value> [-comment string]	Specify maximum delay for timing paths
set_min_delay	set_min_delay [-from <port_list pin_list instance_list net_list clock_list> [-to <port_list pin_list instance_list net_list clock_list> [-through <port_list pin_list instance_list net_list> [-rise_from <clock_list>] [-rise_to <clock_list> [-fall_from <clock_list>] [-fall_to <clock_list>] <delay_value>	Specify minimum delay for timing paths
set_multicycle_path	set_multicycle_path [-from <port_list pin_list instance_list net_list clock_list> [-to <port_list pin_list instance_list net_list clock_list> [-through <port_list pin_list instance_list net_list> [-rise_from <clock_list>] [-rise_to <clock_list> [-fall_from <clock_list>] [-fall_to <clock_list> [-setup -hold] [-start -end] <path_multiplier>	Define multicycle path
set_output_delay	set_output_delay -clock <clock_name> [-clock_fall] [-max] [-min] [-add_delay] <delay_value> <port_list>	Set output delay on ports

Radiant Software Physical Constraints Tcl Commands

The following table provides a listing of all valid Radiant software Physical Constraints Tcl commands

Table 21: Radiant Software Physical Constraints Tcl Commands

Command	Arguments	Description
ldc_create_group	ldc_create_group -name <group_name> [-bbox {height width}] <objects>	Defines a single identifier that refers to a group of objects
ldc_create_region	ldc_create_region -name <region_name> -site <site> -width <width> -height <height>	Define a rectangular area
ldc_create_vref	ldc_create_vref -name <vref_name> -site <site_name>	Define a voltage reference
ldc_prohibit	ldc_prohibit -site <site> -region <region>	Prohibits the use of a site or all sites inside a region
ldc_set_attribute	ldc_set_attribute <key-value list> [objects]	Set object attributes
ldc_define_global_attribute	ldc_define_global_attribute -attr <attr_type> -value <attr_value> [-disable] [-comment <comment>]	Set LSE synthesis global attributes
ldc_define_attribute	ldc_define_attribute -attr <attr_type> -value <attr_value> -object_type <object type> -object <object> [-disable] [-comment <comment>]	Set LSE synthesis attributes for given objects
ldc_set_location	ldc_set_location [-site <site_name>] [-bank <bank_num>] [-region <region_name>] <object>	Set object location
ldc_set_port	ldc_set_port [-iobuf [-vref <vref_name>]] [-sso] <key-value list> <ports>	Set port constraint attributes
ldc_set_sysconfig	ldc_set_sysconfig <key-value list>	Set sysconfig attributes
ldc_set_vcc	ldc_set_vcc [-bank bank -core] [-derate derate] [voltage]	Sets the voltage and/or derate for the bank or core

Radiant Software Physical Constraints Tcl Commands Examples This section illustrates and describes a few samples of Radiant Physical Constraints Tcl commands.

Example 1 The following **ldc_create_group** command creates a sample group with 3 instances, and places all instances within the group to a 2x2 bbox.

```
ldc_create_group -name sample_group -bbox {2 2} [get_cells  
{i16_1_lut i18_2_lut i25_3_lut }]
```

Example 2 The following **ldc_set_location** command places the port clk to pin E7.

```
ldc_set_location -site {E7} [get_ports clk]
```

Example 3 The following **ldc_set_port** command sets IO_TYPE, DRIVE, SLEWRATE attributes of the port rst.

```
ldc_set_port -iobuf {IO_TYPE=LVCOS33 DRIVE=8 SLEWRATE=FAST}  
[get_ports rst]
```

Radiant Software Project Tcl Commands

The Radiant software Project Tcl Commands allow you to control the contents and settings applied to the tools, and source associated with your design. Projects can be opened, closed, and configured to a consistent state using the commands described in this section.

Radiant Software Project Tcl Command Descriptions The following table provides a listing of all valid Radiant software project-related Tcl command options and describes option functionality.

Table 22: Radiant Software Project Tcl Commands

Command	Function (Argument)	Description
prj_create	prj_create -name <project name> [-dev <device name>] [-performance <performance grade>] [-impl <initial implementation name>] [impl_dir <initial implementation directory>] [-synthesis <synthesis tool name>]	Creates a new project inside the current working directory. The <i>new</i> command can only be used when no other project is currently open. The -name <project name> argument specifies the name of the project. This creates a <project name>.rdf file in the current working directory. The -impl <initial implementation name> argument specifies the active implementation when the project is created. If this left unspecified a default implementation called "Implementation0" is created. The -dev <device name> argument specifies the FPGA family, density, footprint, performance grade, and temperature grade to generate designs for. Use the Lattice OPN (Ordering Part Number) for the <device name> argument. The -performance <performance grade> argument specifies the device performance grade explicitly. For iCE40UP device, performance grade can't be inferred from the device part name such as iCE40UP3K-UWG30ITR. If no performance grade specified, default performance value is used. The -impl_dir <initial implementation directory> argument defines the directory where temporary files are stored. If this is not specified the current working directory is used.
prj_close	prj_close	Exits the current project. Any unsaved changes are discarded.
prj_open	prj_open <projectfile.rdf>	Opens the specified project in the software environment.
prj_save	prj_save [projectfile.rdf]	Updates the project with all changes made during the current session and the project file is saved.
prj_saveas	prj_saveas -name <new project name> -dir <new project directory> [-copy_gen]	Save the current project as a new project with specified name and directory.

Table 22: Radiant Software Project Tcl Commands

Command	Function (Argument)	Description
prj_set_opt	prj_set_opt : List all the options in the current project prj_set_opt <option name> [option value list] : List or set the option value prj_set_opt -append <option name> <option value> : Append a value to the specified option value prj_set_opt -rem <option name>... : Remove the options of the current project	List, set or remove a project option.
prj_archive	prj_archive [-includeAll] <archive_file> : Archive the current project into the archive_file prj_archive -extract -dir <destination directory> <archive_file> : Extract the archive file and load the project	Archive the current project.
prj_set_device	prj_set_device [-family <family name>] [-device <device name>] [-package <package name>] [-performance <performance grade>] [-operation <operation>] [-part <part name>] : Change the device to the specified family, device, package, performance, operation, part	Set the device.

Table 22: Radiant Software Project Tcl Commands

Command	Function (Argument)	Description
prj_add_source	prj_add_source [-impl <implement name>] [-simulate_only -synthesis_only] [-include <path list for Verilog include search path>] [-work <VHDL lib name>] [-format <format name>] [[-opt <name=value>] ...] [-exclude <src file>...	Adds a VHDL source file to the specified or active implementation. The syntax used for the Add function depends upon the source file's implementation language. [-work <VHDL lib name>]: Assigns the source code to the specified library name space. [-impl <implementation name>]: This switch is used to add a source file to a Radiant software implementation. If this switch is not specified the source file is added to the active implementation. [-format <format name>]: This switch is used to add a source file format is Verilog or VHDL. [-opt name=value]: The -opt argument allows you to set a custom, user-defined option. See Example 7 for guidelines and usage. <sre file>...: One or more VHDL source files to add to the specified implementation.
prj_enable_source	prj_enable_source [-impl <implement name>] <src file> ...	Enables the excluded design sources from the current project, that is, it will activate a source file for synthesis, to be used as a constraint, or for Reveal debugging.
prj_disable_source	prj_disable_source [-impl <implement name>] <src file> ...	Disables the excluded design sources from the current project, that is, it will activate a source file for synthesis, to be used as a constraint or for Reveal debugging.
prj_remove_source	prj_remove_source [-impl <implement name>] -all :Remove all the design sources in project prj_remove_source [-impl <implement name>] <src file>	Deletes the specified source files from the specified implementation. If an implementation is not listed explicitly the source files are removed from the active implementation. The source files are not removed from the file system, they are only removed from consideration in the specified implementation.

Table 22: Radiant Software Project Tcl Commands

Command	Function (Argument)	Description
prj_set_source_opt	prj_set_source_opt -src <source name> [-impl <implement name>] : List all the options in the specified source prj_set_source_opt -src <source name> [-impl <implement name>] <option name> [option value list] : List or set the source's option value prj_set_source_opt -src <source name> [-impl <implement name>] -append <option name> <option value> : Append a value to the specified option value prj_set_source_opt -src <source name> [-impl <implement name>] -rem <option name>... : Remove the options of the source	List, set or remove a source option.
prj_set_source_format	prj_set_source_format -src <source name> [-impl <implement name>] <format name> : Currently, format name only Verilog or VHDL is supported	Set a source format Verilog or VHDL.
prj_create_impl	prj_create_impl <new impl name> [-dir <implementation directory>] [-strategy <default strategy name>] [-synthesis <synthesis tool name>]	Create a new implementation in the current project with '<new impl name>'. The new implementation will use the current active implementation's strategy as the default strategy if no valid strategy is set.
prj_remove_impl	prj_remove_impl <implement name>	Delete the specified implementation in the current project with '<impl name>'.

Table 22: Radiant Software Project Tcl Commands

Command	Function (Argument)	Description
prj_set_impl_opt	prj_set_impl_opt [-impl <implement name>] : List all the options in the specified implementation prj_set_impl_opt [-impl <implement name>] <option name> [option value list] : List or set the implementation's option value prj_set_impl_opt [-impl <implement name>] -append <option name> <option value> : Append a value to the specified option value prj_set_impl_opt [-impl <implement name>] -rem <option name>... : Remove the the options in the implementation	Allows you to add, list, or remove implementation options with the name <implement name> in the specified or active implementation of the current project. If the -rem option is used, the following option names appearing after it will be removed. If no argument is used (i.e., "prj_set_impl_opt"), the default is to list all implementation options. If only the <option name> argument is used (i.e., "prj_impl option <option name>"), then the value of that option in the project will be returned. If the <option name> is VerilogStandard, then the valid option value must be Verilog 95, Verilog 2001 or System Verilog. The command will set the option value to the option specified by <option name>. If the <option value> is empty then the option will be removed and ignored (e.g., prj_impl option -rem). The -run_flow argument allows you to switch from the normal mode to an "initial" incremental flow mode and "incremental" which is the mode you should be in after an intial design run during the incremental design flow. With no value parameters specified, -run_flow will return the current mode setting.
prj_set_prescript	prj_set_prescript [-impl <implement name>] <milestone name> <script_file> : milestone name can be 'syn', 'map', 'par', 'export'	List or set user Tcl script before running milestone.
prj_set_postscript	prj_set_postscript [-impl <implement name>] <milestone name> <script_file> : milestone name can be 'syn', 'map', 'par', 'export'	List or set user Tcl script after running milestone.
prj_activate_impl	prj_activate_impl <implement name>	Activates the implementation with the name <implement name>.
prj_clean_impl	prj_clean_impl [-impl <implement name>]	Clean up the implementation result files in the current project.

Table 22: Radiant Software Project Tcl Commands

Command	Function (Argument)	Description
prj_clone_impl	prj_clone_impl <new impl name> [-dir <new impl directory>] [-copyRef] [-impl <original impl name>]	Clone an existing implementation.
prj_run_synthesis	prj_run_synthesis	Run synthesis process.
prj_run_map	prj_run_map	Run map process.
prj_run_par	prj_run_par	Run par process.
prj_run_bitstream	prj_run_bitstream	Run bitstream process.
prj_create_strategy	prj_create_strategy -name <new strategy name> -file <strategy file name>	Create a new strategy with default setting.
prj_remove_strategy	prj_remove_strategy <strategy name>	Deletes an existing strategy.
prj_copy_strategy	prj_copy_strategy -from <source strategy name> -name <new strategy name> -file <strategy file name>	Copies an existing strategy and saves it to a newly created strategy file.
prj_import_strategy	prj_import_strategy -name <new strategy name> -file <strategy file name>	Import an existing strategy file.
prj_set_strategy	prj_set_strategy [-impl <implementation name>] <strategy name>	Associate the strategy with the specified implementation.
prj_list_strategy	prj_list_strategy [-strategy <strategy name>] <pattern>	List value to a strategy item.
prj_set_strategy_value	prj_set_strategy_value [-strategy <strategy name>] <option name=option value> ...	Set value to a strategy item.

Radiant Software Project Tcl Command Examples This section illustrates and describes a few samples of Radiant software Project Tcl commands.

Example 1 To create a new project, your command may appear something like the following which shows the creation of a ThunderPlus device.

```
prj_create -name "m" -impl "m" -dev iCE40UP3K-UWG30ITR
```

Example 2 To save a project and give it a certain name (save as), use the project save command as shown below:

```
prj_save "my_project"
```

To simply save the current project just use the save function with no values:

```
prj_save
```

Example 3 To open an existing project, the command syntax would appear with the absolute file path on your system as shown in the following example:

```
prj_open "C:/projects/radiant/adder/my_project.rdf"
```

Example 4 To add a source file, in this case a source LDC file, use the `prj_src add` command as shown below and specify the complete file path:

```
prj_add_source "C:/my_project/radiant/counter/counter.ldc"
```

Example 5 The following examples below shows the `prj_run` command being used:

```
prj_run_par
```

In this final example, synthesis is run.

```
prj_run_synthesis
```

Example 6 To copy another project strategy that is already established in another Radiant software project from your console, use the `prj_copy_strategy copy` command as shown below and specify the new strategy name and the strategy file name.

```
prj_copy_strategy -from source_strategy -name new_strategy -  
file strategy.stg
```

Example 7 The `prj_add_source` command allows you to set a custom, user-defined option. This `-opt` argument value, however, cannot conflict with existing options already in the system, that is, its identifier must differ from system commands such as "include" and "lib" for example. In addition, a user-defined option may not affect the internal flow but can be queried for any usage in a user's script to arrange their design and sources. All user-defined options can be written to the Radiant software project RDF file.

In the example below, the `-opt` argument is used as a qualifier to make a distinction between to .rvl file test cases.

```
prj_add_source test1.rvl -opt "debug_case=golden_case"  
prj_add_source test2.rvl -opt "debug_case=bad_case"
```

Example 8 After you modify your strategy settings in the Radiant software interface the values are saved to the current setting via a Tcl command. For example, a command similar to the following will be called if Synplify frequency and area options are changed.

```
prj_set_strategy_value -strategy strategy1 SYN_Frequency=300  
SYN_Area=False
```

Example 9 To set the top-level module of a project, use the `prj_set_impl_opt` command as shown below and specify the implementation name and module name that will be set as top-level.

```
prj_set_impl_opt -impl impl1 top count
```

Simulation Libraries Compilation Tcl Commands

This section provides Simulation Libraries Compilation extended Tcl command syntax and usage examples.

Simulation Libraries Compilation Tcl Command Descriptions The following table provides a listing of all valid Simulation Libraries Compilation Tcl Command arguments and describes their usage.

Note

Running `cmpl_libs` may take a long time and may cause the Radiant software to hang.

- ▶ It is recommended to run `cmpl_libs` using the Radiant TCL Console (**Start Menu > Lattice Radiant Software > Accessories > TCL Console**).

or,

- ▶ Run `cmpl_libs.tcl` using the command line console. Refer to [“Running `cmpl\_libs.tcl` from the Command Line” on page 1251](#).

Table 23: Simulation Libraries Compilation Tcl Command

Command	Function (Argument)	Description
<code>cmpl_libs</code>	-sim_path <sim_path> [-sim_vendor {mentor<default>}] [-device {ice40up LIFCL all<default>}] [-target_path <target_path>]	The <code>-sim_path</code> argument specifies the path to the simulation tool executable (binary) folder. This option is mandatory. Currently only Modelsim and Questa simulators are supported. NOTE: If you are a Windows user and prefer the <code>\</code> notation in the path, you must surround it with <code>{}</code>. And <code>""</code> or <code>{}</code> will be needed if the path has spaces. NOTE: To execute this command error free, Questasim 10.4e or a later 10.4 version, or Questasim 10.5b or a later version should be used for compilation. The <code>-sim_vendor</code> argument is optional, and intended for future use. It currently supports only Mentor Graphics simulators (Modelsim / Questa). The <code>-device</code> argument specifies the Lattice FPGA device to compile simulation libraries for. This argument is optional, and the default is to compile libraries for all the Lattice FPGA devices. The <code>-target_path</code> argument specifies the target path, where you want the compiled libraries and <code>modelsim.ini</code> file to be located. This argument is optional, and the default target path is the current folder. NOTES: (1) This argument is recommended if you did not change the current folder from the Radiant software startup (binary) folder, or if the current folder is write-protected. (2) If you are a Windows user and prefer the <code>\</code> notation in the path, you must surround it with <code>{}</code>. And <code>""</code> or <code>{}</code> will be needed if the path has spaces.

Simulation Libraries Compilation Tcl Command Examples This section illustrates and describes a few examples of Simulation Libraries Compilation Tcl command.

Example 1 The following command will compile all the Lattice FPGA libraries for both Verilog and VHDL simulation, and place them under the folder specified by `-target_path`. The path to Modelsim is specified by `-sim_path`.

```
cmpl_libs -sim_path C:/questasim64_10.4e/win64 -target_path c:/
mti_libs
```

Reveal Inserter Tcl Commands

This section provides Reveal Inserter extended Tcl command syntax, command options, and usage examples.

Reveal Inserter Tcl Command Descriptions The following table provides a listing of all valid Reveal Inserter Tcl command options and describes option functionality.

Table 24: Reveal Inserter Tcl Commands

Command	Function (Argument)	Description
<code>rvl_new_project</code>	<code>rvl_new_project <rvl file></code>	Create a new reveal inserter project.
<code>rvl_open_project</code>	<code>rvl_open_project <rvl file></code>	Open a reveal inserter project file.
<code>rvl_save_project</code>	<code>rvl_save_project <rvl file></code>	Save the current reveal inserter project.
<code>rvl_close_project</code>	<code>rvl_close_project</code>	Close the current reveal inserter project.
<code>rvl_run_project</code>	<code>rvl_run_project [-save] [-saveAs <file>] [-overwrite] [-drc] [-insert_core <core_name>]</code>	▶ "Run inserting debug core task or DRC checking on the current reveal inserter project" ▶ <code>-save</code>: Save the project before run command ▶ <code>-saveAs</code>: Save as a different file before run command ▶ <code>-overwrite</code>: Overwrite the existing file if the saved as to file exists already ▶ <code>-drc</code>: Run DRC checking only ▶ <code>-insert_core</code>: Specify the core to be inserted. All cores will be inserted if none is specified"
<code>rvl_add_core</code>	<code>rvl_add_core <core name></code>	Add a new core in current project.
<code>rvl_del_core</code>	<code>rvl_del_core <core name></code>	Remove an existing core from current project.

Table 24: Reveal Inserter Tcl Commands

Command	Function (Argument)	Description
rvl_rename_core	rvl_rename_core <core name> <new core name>	Rename an existing core from current project.
rvl_set_core	rvl_set_core [core name]	List the default core or select a core as the default core in current project.
rvl_list_core	rvl_list_core	List all cores in current project.
rvl_add_serdes	rvl_add_serdes	Add the IO EOM core into current project.
rvl_del_serdes	rvl_del_serdes	Remove the IO EOM core from current project.
rvl_set_serdes	rvl_set_serdes [clk=<clock name>] [rst=<reset signal, default value is VLO>]	List or set options of IO EOM core.
rvl_add_controller	rvl_add_controller	Add the Controller Core into current project.
rvl_del_controller	rvl_del_controller	Remove the Controller Core from current project.
rvl_set_controller	rvl_set_controller [-item LED Switch Register PLL1 PLL2 ...] [-set_opt {opt_list}] [-set_sig {sig_list}]	List or set options of Controller items in current project You can set opt_list with the following: ▶ Insert=[on off] for all item ▶ Width=[1..32] for LED and Switch ▶ AddrWidth=[4..16] for Register ▶ DataWidth=[4..32] for Register. sig_list with the following: ▶ SWn=signal where n=1 to Width for Switch. ▶ LEDn=signal where n=1 to Width for LED. ▶ Clock=clk_signal for Register. ▶ Enable=en_signal for Register. ▶ Wr_Rdn=wr_rdn_signal for Register. ▶ Address=addr_bus for Register. ▶ WData=wdata_bus for Register. ▶ RData=rdata_bus for Register.
rvl_add_trace	rvl_add_trace [-core <core name>] [-insert_at <position>] <signals list>	Add trace signals in a debug core in current project. You can specify an existing trace signal/bus name or a position number in a trace bus as the inserting position.
rvl_del_trace	rvl_del_trace [-core <core name>] <signals list>	Delete trace signals in a debug core in current project.

Table 24: Reveal Inserter Tcl Commands

Command	Function (Argument)	Description
rvl_rename_trace	rvl_rename_trace [-core <core name>] -bus <bus name> <new bus name>	Change the name of a trace bus in a debug core in current project.
rvl_list_trace	rvl_list_trace [-core <core name>]	List all trace signals in a debug core in current project.
rvl_move_trace	rvl_move_trace [-core <core name>] [-move_to <position>] <signals list>	Move and rearrange the order of trace signals in a debug core in current project. You can specify an existing trace signal/bus name or a position number in a trace bus as the new position.
rvl_group_trace	rvl_group_trace [-core <core name>] -bus <bus name> <signals list>	Group specified trace signals in a debug core in current project into a bus.
rvl_ungroup_trace	rvl_ungroup_trace [-core <core name>] <bus name>	Ungroup trace signals in a trace bus in a debug core in current project.
rvl_set_traceoption	rvl_set_traceoption [-core <core name>] [option=value]	List or set trace options of a debug core in current project. You can set the following option: SampleClk = [signal name].
rvl_set_trigoption	rvl_set_trigoption [-core <core name>] [option=value]	List or set trigger options of a debug core in current project. You can set the following option: DefaultRadix = [bin oct dec hex] EventCounter = [on off] CounterValue = [2,4,8,16,...,65536] (depend on FinalCounter is on) TriggerOut = [on off] OutNetType = [IO NET BOTH] (depend on TriggerOut is on) OutNetName = [net name] (depend on TriggerOut is on) OutNetPri = [Active_Low Active_High] (depend on TriggerOut is on) OutNetMPW = [pulse number] (depend on TriggerOut is on).
rvl_list_tu	rvl_list_tu [-core <core name>]	List all trigger units in a debug core in current project.

Table 24: Reveal Inserter Tcl Commands

Command	Function (Argument)	Description
rvl_add_tu	rvl_add_tu [-core <core name>] [-radix <bin oct dec hex>] [-name <new TU name>] <TU definition>	Add a new trigger unit to a debug core in current project. TU definition format: "{signal list} Operator Value" Operator must be "==", "!=", ">", ">=", "<", "<=", ".RE."(rising edge), ".FE."(falling edge) and ".SC."(serial compare). A default trigger unit name will be created if it's omitted in command..
rvl_del_tu	rvl_del_tu [-core <core name>] <TU name>	Remove an existing core from current project.
rvl_rename_tu	rvl_rename_tu [-core <core name>] <old name> <new name>	Rename an existing core in current project.
rvl_set_tu	rvl_set_tu [-core <core name>] [-radix <bin oct dec hex>] -name <TU name> [-add_sig <signal list>] [-del_sig <signal list>] [-set_sig <signal list>] [-expr <TU definition>] [-op operator] [-val value]	Set a trigger unit in a debug core in current project. TU definition format: "{signal list} Operator Value" Operator must be "==", "!=", ">", ">=", "<", "<=", ".RE."(rising edge), ".FE."(falling edge) and ".SC."(serial compare)..
rvl_list_te	rvl_list_te [-core <core name>]	List all trigger expressions in a debug core in current project.
rvl_add_te	rvl_add_te [-core <core name>] [-ram <EBR Slice>] [-name <new TE name>] [-expression <expression string>] [-max_seq_depth <max depth>] [-max_event_count <max event count>]	Add a new trigger expression to a debug core in current project. A default trigger expression name will be created if it's omitted in command.
rvl_del_te	rvl_del_te [-core <core name>] <TE name>	Delete an existing trigger expression in a debug core in current project.
rvl_rename_te	rvl_rename_te [-core <core name>] <old name> <new name>	Rename an existing trigger expression in a debug core in current project.
rvl_set_te	rvl_set_te [-core <core name>] [-ram <EBR Slice>] [-expression <expression string>] [-max_seq_depth <max depth>] [-max_event_count <max event count>] <TE name>	Change an existing trigger expression in a debug core in current project.

Table 24: Reveal Inserter Tcl Commands

Command	Function (Argument)	Description
rvl_add_controller	rvl_add_controller	Add the Controller Core into current project.
rvl_del_controller	rvl_del_controller	Remove the Controller Core from current project.
rvl_set_controller	rvl_set_controller [-item LED Switch Register PLL1 PLL2 ...] [-set_opt {opt_list}] [-set_sig {sig_list}] You can set opt_list with the following: Insert=[on off] for all item Width=[1..32] for LED and Switch AddrWidth=[4..16] for Register DataWidth=[4..32] for Register You can set sig_list with the following: SWn=signal where n=1 to Width for Switch LEDn=signal where n=1 to Width for LED Clock=clk_signal for Register Enable=en_signal for Register Wr_Rdn=wr_rdn_signal for Register Address=addr_bus for Register WData=wdata_bus for Register RData=rdata_bus for Register	List or set options of Controller items in current project.

Reveal Inserter Tcl Command Examples This section illustrates and describes a few samples of Reveal Inserter Tcl commands.

Example 1 To create a new Reveal Inserter project with the .rvl file extension in your project directory, use the rvl_project command as shown below using the new option.

```
rvl_new_project my_project.rvl
```

Example 2 The following example shows how to set up TU parameters for Reveal Inserter:

```
rvl_set_tu -name TU -add_sig {count[7:0]} -op == -val C3 -radix Hex
```

Reveal Analyzer Tcl Commands

This section provides Reveal Analyzer extended Tcl command syntax, command options, and usage examples.

Reveal Analyzer Tcl Command Descriptions The following table provides a listing of all valid Reveal Analyzer Tcl command options and describes option functionality.

Table 25: Reveal Analyzer Tcl Commands

Command	Function (Argument)	Description
rva_new_project	rva_new_project <file>	Create a new Reveal Analyzer project.
rva_open_project	rva_open_project <file>	Open a Reveal Analyzer project file.
rva_save_project	rva_save_project <file>	Save the current Reveal Analyzer project.
rva_close_project	rva_close_project <file>	Close the current Reveal Analyzer project.
rva_export_project	rva_export_project -vcd <file name> [-module <title>]	Export VCD file. Optional to include a title in the VCD file. By default the title will be "<unknown>".
	rva_export_project -txt <file name> [-siglist <signal list>]	Export TEXT file. Optional to export selected signal list only. By default all signals are exported.
rva_set_project	rva_set_project [-frequency <val>] [-period <val>] [-tckdelay <val>] [-cabletype <val>] [-cableport <val>]	No arguments specified will return options. -frequency: sets the frequency value for sample clock in MHz -period: sets a period value for sample clock in ns or ps -tckdelay: sets a TCK clock pin pulse width delay value -cabletype: sets the type of cable. Values are LATTICE USB USB2 -cableport: sets the port number as integer >= 0.
rva_run	rva_run	Runs until trigger condition to capture data.
rva_stop	rva_stop	Stops without capturing data.
rva_manualtrig	rva_manualtrig	Manual Trigger to capture data.
rva_get_trace	rva_get_trace	Lists all trace signals in a core.
rva_set_core	rva_set_core [-name <name>] [-run <on off>]	No arguments return list of core. -name: Select core. Needed for other actions -run: Turns run option on/off for core.

Table 25: Reveal Analyzer Tcl Commands

Command	Function (Argument)	Description
rva_set_tu	rva_set_tu [-name <name>] [-operator {== != > >= < <= "rising edge" "falling edge"}] [-value <value>] [/?radix {bin oct dec hex <token>}]	No arguments, return list of TU. -name: Select TU. If no options, return options and value for the selected TU. -operator: Sets the comparison operator. Operators are equal to (==), not equal to (!=), greater than (>), greater than or equal to (>=), less than (<), less than or equal to (<=), "rising edge", "falling edge", and serial compare (serial). -value: Sets TU value -radix: Sets TU radix. Options are binary (bin), octal (oct), decimal (dec), hexadecimal (hex), or the name of a token set.
rva_rename_tu	rva_rename_tu <name> <new name>	This function renames TU.
rva_set_te	rva_set_te [-name <name>] [-expression <expression list>] [-enable <on off>]	No arguments, return list of TE. -name: Select TE. If no options, return options and value for the selected TE. -expression: Sets TE expression -enable: Enables/disables TE.
rva_rename_te	rva_rename_te <name> <new name>	This function renames TE.
rva_set_trigoptn	rva_set_trigoptn [-teall <AND OR>] [-finalcounter <on off>] [-finalcountervalue <val>] [-samples <val>] [-numtriggers <val>] [-position <pre center post val>]	No arguments specified will return list of options. -teall: Sets AND ALL or OR ALL for all TEs -finalcounter: Turns final trigger counter on/off -finalcountervalue: Sets final trigger counter value -samples: Sets number of samples to capture -numtriggers: Sets number of triggers to capture -position: Sets trigger position to pre-selected or user value.
rva_add_token	rva_add_token <tokenset name> <name=value>	Add a token with new name and value in a specific token set.
rva_del_token	rva_del_token <tokenset name> <token name>	Delete a specific token in a specific token set.

Table 25: Reveal Analyzer Tcl Commands

Command	Function (Argument)	Description
rva_set_token	rva_set_token <tokenset name> <token name> -name <new token name> -value <new token value>	Select specific token in specific token set. -name: Set token name -value: Set token value.
rva_add_tokens et	rva_add_tokenset [-tokenset <tokenset name>] [-bits <token bits>] [-token <name=value>]	No arguments, add a token set with default name and bits. -tokenset: Set token set name -bits: Set token set bits -token: Add extra tokens.
rva_del_tokens et	rva_del_tokenset <tokenset name>	Delete the specific token set.
	rva_del_tokenset -all	Delete all token set.
rva_set_tokens et	rva_set_tokenset <tokenset name> -name <new token set name> -bits <new token bits>	Select specific token set -name: Rename a token set -bits: Set number of bits in tokens.
rva_export_token set	rva_export_tokenset <file name>	Export all token set to a specific file.
rva_import_token set	rva_import_tokenset <file name>	Import and merge all token set from a specific file.
rva_open_controller	rva_open_controller	Open Controller connection to Lattice device before read/write begins.
rva_target_controller	rva_target_controller	Set Controller core as target before read/ write begins.
rva_close_controller	rva_close_controller	Close Controller connection to Lattice device after read/write finished.
rva_read_controller	rva_read_controller -addr <addr32>	Read data from 32-bit address in hex.
rva_write_controller	rva_write_controller -addr <addr32> -data <data>	Write data to 32-bit address in hex.

Table 25: Reveal Analyzer Tcl Commands

Command	Function (Argument)	Description
rva_set_controller	rva_set_controller [-option <value>] rva_set_controller -hard_ip <ipname> [-option <value>]	Set the options for Controller core. -cable_type: Set type of cable as USB or USB2. -cable_port: Set logical port of cable as integer. If no arguments specified, then return list of options and values. Set options of DPHY1, DPHY2, I2CFIFO1, SGMIIICDR1, SGMIIICDR2, PCIELL1, PCSCH1 to PCSCH8. If no option/value specified, then return list of options and values. DPHY1, DPHY2 options and values: -high_speed_select: Set value as &#34;1.5Gbps and below&#34;,&#34;2.5Gbps&#34;. -bypass_pll_clock: Set value as Enabled,Disabled. -pll_cn_counter: Set value as integer from 0 to 31. -pll_cm_counter: Set value as integer from 0 to 255. -pll_co_counter: Set value as integer from 0 to 7. I2CFIFO1 options and values: -generic_call_response: Set value as Enabled,Disabled. -sda_input_delay: Set value as Enabled,Disabled. -sda_output_delay: Set value as Enabled,Disabled. -fifo_mode_select: Set Value as &#34;Register Mode&#34;,&#34;FIFO Mode&#34;. -fifo_clock_stretching: Set value as Enabled,Disabled. -fifo_almost_full_value: Set value as integer from 0 to 31. -fifo_sleep_clock: Set value as Enabled,Disabled. -slave_address_match_wakeup: Set value as Enabled,Disabled. -fifo_almost_full_wakeup: Set value as Enabled,Disabled. -master_read_complete_wakeup: Set value as Enabled,Disabled.

Table 25: Reveal Analyzer Tcl Commands

Command	Function (Argument)	Description
		SGMIICDR1, SGMIICDR2 options and values: -cdr_loss_of_lock: Set value as &#34;Lock=+/-1000ppm x2 Unlock=+/-1500ppm x2&#34;; &#34;Lock=+/-2000ppm x2 Unlock=+/-2500ppm x2&#34;; &#34;Lock=+/-4000ppm Unlock=+/-7000ppm&#34;; &#34;Lock=+/-300ppm Unlock=+/-450ppm&#34;. -loopback_control: Set value as Disabled,&#34;Bit Clock to IO&#34;;&#34;Serial Data to IO&#34;. PCIELL1 options and values: -swing_control: Set value as full,reduced. PCSCH1 to PCSCH8 options and values: -tx_differential_amplitude: Set value as 1000mV,900mV,800mV,700mV,600mV,500mV,400mV,300mV,200mV,100mV,0mV. -tx_output_termination: Set value as &#34;100 ohm&#34;;&#34;150 ohm&#34;. -tx_postcursor_ratio: Set value as integer from 0 to 63 of 128th. -tx_precursor_ratio: Set value as integer from 0 to 63 of 128th. -tx_invert_polarity: Set value as on,off. -rx_input_termination: Set value as &#34;100 ohm&#34;; &#34;150 ohm&#34;. -rx_equalization: Set value as &#34;RL2plus 8GT/s&#34;; &#34;RL2plus 5GT/s&#34;; &#34;RL2plus 2.5GT/s&#34;; &#34;SS_LMS 8GT/s&#34;;&#34;SS_LMS 5GT/s&#34;;&#34; SS_LMS 2.5GT/s&#34;. -rx_invert_polarity: Set value as on,off. -loopback_mode: Set value as Disabled,&#34;PMA Near-End Serial Loopback&#34;; &#34;PMA Far-End Parallel Loopback&#34;; &#34;8B/10B PCS Near-End Parallel Loopback&#34;;&#34;8B/10B PCS Far-End Parallel Loopback&#34;; &#34;64B/66B PCS Loopback Path A&#34;; &#34;64B/66B PCS Loopback Path B&#34;; &#34;64B/66B PCS Loopback Path C&#34;.

Table 25: Reveal Analyzer Tcl Commands

Command	Function (Argument)	Description
		-reset_tx_mpcs: Set value as on,off.
		-reset_rx_mpcs: Set value as on,off.
		-reset_tx_pll: Set value as on,off.
		-reset_rx_pll: Set value as on,off.
		-force_tx_driver: Set value as on,off.
		-force_rx_driver: Set value as on,off.
		-eye_quality: Set quality of eye diagram as high, normal, low.

Reveal Analyzer Tcl Command Examples This section illustrates and describes a few samples of Reveal Analyzer Tcl commands.

Example 1 The following command line example shows how to specify a new project that uses a parallel cable port.

```
rva_new_project -rva untitled -rvl "count.rvl" -dev "LFXP2-5E:0x01299043" -port 888 -cable LATTICE
```

Example 2 The following example shows how to set up TU parameters for Reveal Analyzer:

```
rva_set_tu -name TU1 -operator == -value 10110100 -radix bin
```

Power Calculator Tcl Commands

This section provides Power Calculator extended Tcl command syntax, command options, and usage examples.

Power Calculator Tcl Command Descriptions The following table provides a listing of all valid Power Calculator Tcl command options and describes option functionality.

Table 26: Power Calculator Tcl Commands

Command	Function (Argument)	Description
pwc_new_project	pwc_new_project <file>	Create a new project.
pwc_open_project	pwc_open_project <file>	Open a project file.
pwc_save_project	pwc_save_project <file>	Save the current project.
pwc_close_project	pwc_close_project	Close the current project.
pwc_set_afpervcd	pwc_set_afpervcd <file>	Open vcd file and set frequency and activity factor.
pwc_set_device	pwc_set_device -family <family name>	Set family.
	pwc_set_device -device <device name>	Set device.

Table 26: Power Calculator Tcl Commands

Command	Function (Argument)	Description
	pwc_set_device -package <package name>	Set package.
	pwc_set_device -speed <speed name>	Set speed.
	pwc_set_device -operating <operating name>	Set operating.
	pwc_set_device -part <part name>	Set part.
pwc_set_processtype	pwc_set_processtype <value>	Set device power process type.
pwc_set_ambienttemp	pwc_set_ambienttemp <value>	Set ambient temperature value.
pwc_set_thetaja	pwc_set_thetaja <value>	Set user defined theta JA.
pwc_set_freq	pwc_set_freq <frequency>	Set default frequency.
	pwc_set_freq -clock <frequency>	Set Clock frequency.
	pwc_set_freq -freq <frequency> -usefreqtwr <0 1> -freqtwropt <min pref trace>	Set frequency by importing timing report.
pwc_set_af	pwc_set_af <value>	Set default activity factor.
pwc_set_estimation	pwc_set_estimation <value>	Sets estimated routing option.
pwc_set_supply	pwc_set_supply -type <value> -voltage <value> -dpm <value>	Set multiplication factor and voltage of named power supply.
pwc_add_ipblock	pwc_add_ipblock -iptype <iptype name>	Add IP Block row.
pwc_set_ipblock	pwc_set_ipblock -iptype <iptype name> -matchkeys {<key1> <value1>}+ -setkey <key> <value> : iptypename mapping to PGT section, key mapping to _KEY in pgd session, value is its value	Set IP Block row.
pwc_remove_ipblock	pwc_remove_ipblock -iptype <iptype name> -matchkeys {<key1> <value1>}+	Remove IP Block row.
pwc_gen_report	pwc_gen_report <file>	Generate text report and write to file.
pwc_gen_htmlreport	pwc_gen_htmlreport <file>	Generate HTML report and write to file.

Power Calculator Tcl Command Examples This section illustrates and describes a few samples of Power Calculator Tcl commands.

Example 1 The follow command below creates a PWC project (.pcf) file named “abc.pcf” from an input UDB file named “abc.UDB”:

```
pwc_new_project abc.pcf -udb abc.udb
```

Example 2 To set the default frequency to, for example, 100 Mhz:

```
pwc_set_freq 100
```

Example 3 The command below saves the current project to a new name:

```
pwc_save_project newname.pcf
```

Example 4 To create an HTML report, you would run a command like the one shown below:

```
pwc_gen_htmlreport c:/abc.html
```

Programmer Tcl Commands

This section provides the Programmer extended Tcl command syntax, command options, and usage examples. The below commands are only supported in standalone Programmer currently.

Programmer Tcl Command Descriptions The following table provides a listing of all valid Programmer Tcl command options and describes option functionality.

Table 27: Programmer Tcl Commands

Command	Function (Argument)	Description
pgr_project	pgr_project open <project_file>	The open command will open the specified project file in-memory.
	pgr_project save [<file_path>]	Writes the current project to the specified path. If there is no file path specified then it will overwrite the original file.
	pgr_project close	Closes the current project. If a Programmer GUI is open with the associated project, then the corresponding Programmer GUI will be closed as well.
	pgr_project help	Displays help for the pgr_project command.

Table 27: Programmer Tcl Commands

Command	Function (Argument)	Description
pgr_program	<no_argument>	When pgr_program is run without arguments it will display the current status of the available settings. Note that specifying a key without a value will display the current value. The following keys can be used to modify those settings. Generally, the pgr_program command and its sub-commands allow you to run the equivalent process commands from the TCL Console window in the Radiant software interface. These commands can override connection options that are set in user defaults.
	pgr_program set -cable <LATTICE USB USB2>	Sets the cable for downloading.
	pgr_program set -portaddress <0x0378 0x0278 0x03bc 0x0378 0x0278 0x03bc 0x<custom address>> <EzUSB-0 EzUSB-1 EzUSB-2 ... EzUSB-15> <FTUSB-0 FTUSB-1 FTUSB-2 ... FTUSB-15>	Sets the port address for the downloading.
	pgr_program run	Executes the current xcf with the current settings. Note that there may be warnings that are displayed in the TCL Console window. These warnings will be ignored and processing will continue.
	pgr_program help	Displays help for pgr_program command.
pgr_genfile	<no_argument>	Programmer generate files command (not supported for customer use)
	pgr_genfile set -process <svf vme12>	Sets file type for file generation.
	pgr_genfile set -outfile <file path>	Sets the output file.
	pgr_genfile run	Generates file based on the current xcf and current settings.
	pgr_genfile help	Displays help for pgr_genfile command.

Programmer Tcl Command Examples This section illustrates and describes a few samples of Programmer Tcl commands.

Example 1 The first command below opens a Programmer XCF project file that exists in the system. There can be many programming files associated with one project. In the GUI interface, the boldfaced file in the Radiant software is the active project file.>

```
pgr_project open /home/mdm/config_file/myfile.xcf
```

Example 2 The following command sets programming option using a USB2 cable at port address “FTUSB-1, then using pgr_program run to program”.

```
pgr_program set -cable USB2 -portaddress FTUSB-1
```

Example 3 The following command sets the file generation type for JTAG SVF file, then using pgr_genfile run to generates an output file “mygenfile.svf” in a relative path.

```
pgr_genfile set -process svf -outfile ../genfiles/mygenfile.svf
```

Engineering Change Order Tcl Commands

This section provides Engineering Change Order (ECO) extended Tcl command syntax, command options, and usage examples.

ECO Tcl Command Descriptions The following table provides a listing of all valid ECO Tcl command options and describes option functionality.

Table 28: ECO Tcl Commands

Command	Function (Argument)	Description
eco_save_design	eco_save_design [-udb <udb_file>]	Saves an existing design or macro.
eco_config_sysio	eco_config_sysio -comp <comp name> {<key=value>}...	Config sysio setting.
eco_config_memory	eco_config_memory -mem_id <memory_id> {-init_file <mem_file> -format HEX BIN ADDR} -all_0 -all_1	Update memory initial value.

ECO Tcl Command Examples This section illustrates and describes a few samples of ECO Tcl commands.

Example 1 The following demonstrates the `sysio` command.

```
eco_config_sysio -comp {data}
{clamp=OFF;diffdrive=NA;diffresistor=OFF;drive=2;glitchfilter=OFF;hysteresis=NA;opendrain=OFF;pullmode=NONE;slewrates=LOW;termination=OFF;vref=OFF}
```

Example 2 The following demonstrates the `memory` command.

```
eco_config_mem -mem_id {mem} -init_file {D:/mem/init_hex.mem} -format HEX
```

Control Message Tcl Commands

This section provides Control Message extended Tcl command syntax, command options, and usage examples.

Control Message Tcl Command Descriptions The following table provides a listing of all valid Control Message Tcl command options and describes option functionality.

Table 29: Control Message Tcl Commands

Command	Function (Argument)	Description
msg_promote	msg_promote <message Id> [-type <0 1 2>]	Set the message promotion type, 0 is error, 1 is warning, and 2 is info.
msg_demote	msg_demote <message Id>	Demote the message from the current project.
msg_disable	msg_disable <message Id>	Disable the message from the current project.
msg_list_status	msg_list_status [-promoted -disabled]	List all promoted or disabled messages from the current project.
msg_clean	msg_clean <message Id> -all	Reset the setting of promoted and disabled message(s) from the current project.
msg_save	msg_save [<message xml file>]	Save the setting of all promoted and disabled messages to the file, default is promote.xml.
msg_load	msg_load [<message xml file>]	Load the setting of all promoted and disabled messages from the file, default is promote.xml.

Control Message Tcl Command Examples This section illustrates and describes a few samples of Control Message Tcl commands.

Example 1 The following demonstrates how to promote the message to error from the current project.

```
msg_promote 70001926 -type 0
```

Example 2 The following demonstrates how to show all promoted messages from the current project.

```
msg_list_status -promoted
```

Example 3 The following demonstrates how to save the settings of all promoted and disabled messages to default promote.xml file.

```
msg_save
```


Chapter 10

Advanced Topics

This chapter explains advanced concepts, features and operational methods for the Radiant software.

Shared Memory Environment

The Radiant software design environment uses a shared memory architecture. Shared memory allows all internal tool views to access the same image of the design at any point in time. Understanding how shared memory is being used can give you insight into managing the environment for optimum performance, especially when your design is large.

There is one shared database that contains the device, design, and constraint information in system memory.

Generating the hierarchy of your design uses an additional database separate from the primary shared memory database.

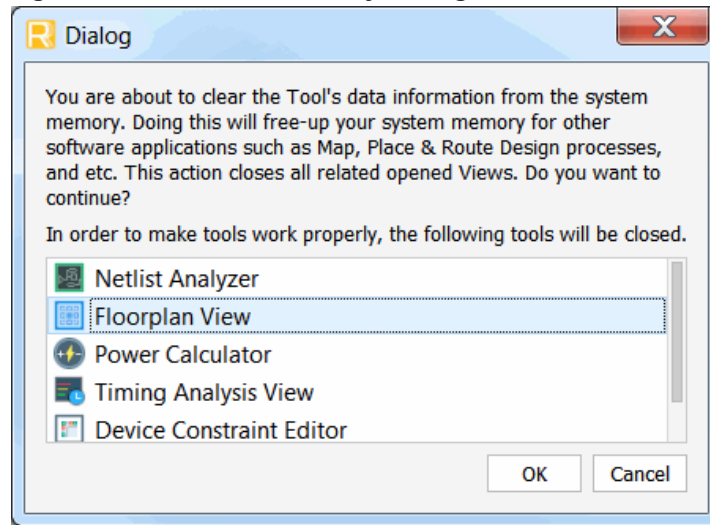
External tools referenced from within the Radiant software, such as those for synthesis and simulation, use their own memory in addition to what is used by the Radiant software.

Because it is accessing shared memory, the initial tool view launch will take longer than the launch of subsequent views.

Clear Tool Memory

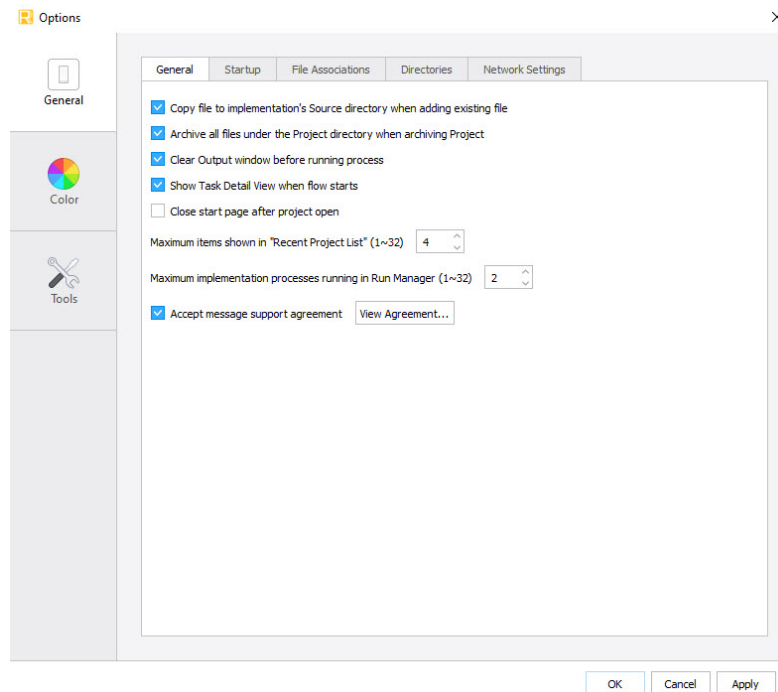
The “Clear Tool Memory” command, available from the Tools menu, clears the device, design, and constraint information from system memory. Clearing the tool memory can speed up memory-intensive processes such as place and route. When your design is very large, it is good practice to clear memory prior to running place and route.

If you have open tool views that will be affected by clearing the tool memory, a confirmation dialog box will open to give you the opportunity to cancel the memory clear.

Figure 83: Clear Tool Memory Dialog

Environment and Tool Options

The Radiant software provides many environment control and tool view display options that enable you to customize your settings. Choose **Tools > Options** to access these options.

Figure 84: Tools Options

The Options dialog box is organized into functional folders.

Commonly configured items include:

- ▶ General settings -- allows you to set some common options, including.
 - ▶ Startup – enables you to configure the default action at startup and also to control the frequency of checking for software updates.
 - ▶ File Associations – allows you to set the programs to be associated with different file types based on the file extensions.
 - ▶ Directories – Set directory location for Synthesis and Simulation tools.
 - ▶ Network Settings – Apply a proxy server and specify a host and port.
- ▶ Color settings -- allows you to set colors for such GUI features as fonts and backgrounds for various Radiant software tools. You can also change the Theme color of the Radiant software (Dark or Light) using the Themes drop-down menu.
- ▶ Tools settings -- allows you to set options for various Radiant software tools, including the Device Constraint Editor, Netlist Analyzer, Timing Constraint Editor, and Source Editor. You can also set the constraint design rule check (DRC) time to real time, prior to saving or when launching a tool.

Batch Tool Operation

The core tools in the FPGA implementation design flow can all be run in batch mode using command-line tool invocation or scripts. For detailed information, see the *Command Line Reference Guide*, available from the Radiant software Start Page.

Tcl Scripts

The Radiant Extended Tcl language enables you to create customized scripts for tasks that you perform often in the Radiant software. Automating these operations through Tcl scripts not only saves time, but also provides a uniform approach to design. This is especially useful when you try to find an optimal solution using numerous design implementations.

Creating Tcl Scripts from Command History

A good first step in Tcl programming is to create a Tcl script by saving some command history and then modifying it as needed. This allows you to get started by using existing command information.

To create a Tcl command script using command history:

1. In the Tcl Console window, perform a *reset* command so that your script won't contain any of the actions that may have already been executed.

```
reset
```

2. Open the project and perform the commands that you want to save as a script.
3. Optionally, enter the history command in the Tcl Console window to ensure that the commands you wish to save are in the console's memory. In the Tcl Console window type

```
save_script <filename.tcl>
```

The <filename.tcl> can be any file identifier that has no spaces and contains no special characters except underscores. For example, **myscript.tcl** or **design_flow_1.tcl** are acceptable save_script values, but **my\$script** or **my script** are invalid. A file name with an extension of .ext will not work.

The <filename.ext> entry can be preceded with an absolute or relative path. Use the "/" (i.e. forward slash) symbol to delimit the path elements. If the path is not specified, the script is saved in the current working directory. The current working directory can be determined by using the TCL *pwd* command.

4. Navigate to your script file and use the text editing tool of your choice to make any necessary changes, such as deleting extraneous lines or invalid arguments.

In most cases, you will need to edit the script you saved and take out any invalid arguments or any commands that cannot be performed in the Radiant software environment due to a conflict or exception. You will likely have to revisit this step later if after running your script, you experience any run errors due to syntax errors or technology exceptions.

Creating Tcl Scripts from Scratch

Tcl commands can be written directly into a script file. You can use any text editor, such as Notepad or vi, to create a file and type in the Tcl commands.

Sample Tcl Script

The following Tcl example shows a simple script that opens a project, runs the entire design flow through the Place & Route process, and then closes the project. A typical script would probably contain more steps, but you can use this example as a general guideline.

```
prj_project open "C:/lsc/Radiant/counter/counter.rdf"
prj_run PAR -impl counter -forceAll
prj_project close
save_script c:/lsc/radiant/examples/counter/myscript2.tcl
```

Running Tcl Scripts

You can run scripts from the Radiant software integrated Tcl Console whether your project is opened or not. You can also run scripts from the external Tcl Console prompt window. The following example procedure uses the integrated Tcl Console and the sample Tcl script from the previous section:

To run a Tcl script that opens a project, runs processes and closes the project:

1. Open the Radiant software but do not open your project. If your project is open, choose **File > Close Project**.
2. If you are using the Radiant software main window, click the small arrow pane switch in the bottom of the Radiant software main window, and then click on the **Tcl Console tab** in the Output area at the bottom to open the console.
3. If there are previously issued commands in the console, type `reset` in the console command line to refresh your session and clear out all previous commands.

```
reset
```

4. Use the TCL `source` command to load and run your TCL script. Since it's the only argument, the `source` command requires the filename of the script you want to load and run. Prefix the script file name with any required relative or absolute path information. To run the example script shown in the previous section use the following:

```
source C:/lsc/radiant/<version_number>/examples/counter/  
myscript2.tcl
```

5. As long as there are no syntax errors or invalid arguments, the script will open the project, synthesize, map, and place-and-route the design. Once the design finishes it closes the project. If there are errors in the script, you will see the errors in red in the Tcl Console after you attempt to run it. Go back to your script and correct the errors that prevented the script from running.

Project Archiving

A Radiant software project archive is a single compressed file (.zip) of your entire project. The project archive can contain all of the files in your project directory, or it can be limited to source-related files. When you use the **File > Archive Project** command, the dialog box provides the option to "Archive all files under the Project directory." When you select this option, the entire project is archived. When you clear this option, only the project's source-related files, including strategies, are archived. Many of these source-related files must be archived in order to achieve the same bitstream results for a fully implemented design.

Whichever archiving method you select, if your project contains source files stored outside the project folder, the remote files will be compressed under the `remote_files` subdirectory in the archive; for example:

```
<project_name>/remote_files/sources
<project_name>remote_files/strategies
```

When unarchiving, you must manually move the archived remote files to the original locations or the project will not work.

File Descriptions

This section provides a list of the file types used in the Radiant software, including those generated during design implementation. The Archive column indicates the files that must be archived in order to achieve the same bitstream results.

Table 34: Source Files

File Type	Definition	Function	Archive?
.fdc	FPGA Design Constraint file	Used for specifying design constraints for Synplify-Pro synthesis tool.	✓
.ipk	Radiant Software IP Package file.	Package file for Radiant software Soft IP.	
.ipx	Manifest file generated by IP Catalog.	Enables changes to be made to a module or IP Catalog.	✓
.ldc	Lattice Design Constraint file	Used for specifying timing constraints for LSE synthesis flow. The .ldc contents will be combined into design database file .udb.	✓
.pcf	Power Calculator project file	Stores power analysis results from information extracted from the design project.	✓
.pdc	Post-Synthesis constraint file.	Used for specifying post-synthesis constraints (timing/physical) for Lattice engines such as MAP and PAR.	✓
.rdf	Radiant Software Project file	Used for managing and implementing all project files in the Radiant software.	✓
.rva	Reveal Analyzer file	Defines the Reveal Analyzer project and contains data about the display of signals in Waveform View.	✓
.rvi	Reveal Inserter debug file	Defines the Reveal project and its modules, identifies the trace and trigger signals, and stores the trace and trigger options.	✓
.sdc	SDC constraints file	Used for specifying design-specific constraints for Synplify-Pro. SDC is replaced by the FDC format in but is still supported in the Radiant software.	✓
.spf	Simulation project file, a script file produced by the Simulation Wizard	Used for running the simulator for your project from the Radiant software.	✓

Table 34: Source Files (Continued)

File Type	Definition	Function	Archive?
.sty	Strategy file	Defines the optimization control settings of implementation tools such as logic synthesis, mapping, and place and route.	✓
.sv	SystemVerilog		
.v	Verilog source file	Contains Verilog description of the circuit structure and function.	✓
.vhd	VHDL source file	Contains VHDL description of the circuit structure and function.	✓
.xcf	Configuration chain file	Used for programming devices in a JTAG daisy chain.	✓

Table 35: IP Files

File Type	Definition	Function	Archive?
<instName>_bb.v	Verilog IP black box file	Provides the Verilog synthesis black box for the IP core and defines the port list.	✓
<instName>.cfg	IP parameter configuration file	Used for re-creating or modifying the core in the IP Catalog tool.	✓
<instName>.svg	Scalable Vector Graphics file	A graphic file used to display module/IP schematic and ports.	✓
<instName>_tmpl.v	Verilog template file	A template for instantiating the generated module. This file can be copied into a user Verilog file.	✓
<instName>_tmpl.vhd	VHDL module template file	A template for instantiating the generated module. This file can be copied into a user VHDL file.	✓
<instName>.v	Verilog module file	Verilog netlist generated by IP Catalog to match the user configuration of the module.	✓

Table 36: Implementation Files

File Type	Definition	Function	Archive?
.bgn	Bitstream generation report file	Reports results of a bit generation (bitgen) run and displays information on options that were set.	
.bin	Bitstream file	Used for SRAM FPGA programming.	
.ibs	Post-Route I/O buffer information specification file (IBIS)	Used for analyzing signal integrity and electromagnetic compatibility (EMC) on printed circuit boards.	
.mcs	PROM file	Used for SRAM FPGA programming.	
.mrp	Map Report file	Provides statistics about component usage in the mapped design.	

Table 36: Implementation Files (Continued)

File Type	Definition	Function	Archive?
.pad	Post-Route PAD report file	Lists all programmable I/O cells used in the design and their associated primary pins.	
.par	Post-Route Place & Route report file	Summarizes information from all iterations and shows the steps taken to achieve a placement and routing solution.	
.sso	Post-PAR SSO analysis file	Reports the noise caused by simultaneously switching outputs.	
.tw1	Post-Map Timing analysis file	Estimates pre-route timing.	
.twr	Post-PAR Timing analysis file	Reports post-route timing.	
.vo	Post-Route Verilog simulation file	Used for post-route simulation.	
<Design name>_vo.sdf	Post-Route SDF simulation file for Verilog	Used for timing simulation.	
.vm	Synthesized netlist file	Netlist file generated by the Radiant software Synthesis tools.	
.udb	Unified Design Database file	Compiled from HDL design source. It may contain both design netlist and constraints.	

Revision History

The following table gives the revision history for this document.

Date	Version	Description
11/21/2022	2022.1	Updated to reflect changes in Radiant 2022.1 software.
6/8/2022	3.2	Updated to reflect changes in Radiant 3.2 software.
3/29/2022	3.1.1 Update	Updated to reflect changes in Radiant 3.1.1 MachXO5-NX Device Update.
3/22/2022	3.1.1	Updated to reflect changes in Radiant 3.1.1 software.
12/7/2021	3.1	Updated to reflect changes in Radiant 3.1 software.
6/14/2021	3.0	Updated screen captures. Updated Chapter 5, Pre-Synthesis Constraint File.
4/5/2021	3.0	Updated to reflect changes in Radiant 3.0 software.
10/20/2020	2.2	Updated to reflect changes in Radiant 2.2 software, including replacing the Active-HDL simulator with Mentor ModelSim.
7/10/2020	2.1.1	Updated to reflect changes in Radiant 2.1.1 software.
5/7/2020	2.1	Updated to reflect changes in Radiant 2.1 software.
11/27/2019	2.0	Updated to reflect changes in Radiant 2.0 software. Removed Appendix A, Reveal User Guide. Starting with Radiant software v2.0, this is now a stand-alone user guide. Removed Appendix B, Programming Tools User Guide. Starting with Radiant software v2.0, this is now a stand-alone user guide.
03/25/2019	1.1	Updated to reflect changes in Radiant 1.1 software.
02/08/2018	1.0	Initial release.