

Design Compiler®

User Guide

Version P-2019.03, March 2019

SYNOPSYS®

Copyright Notice and Proprietary Information

©2019 Synopsys, Inc. All rights reserved. This Synopsys software and all associated documentation are proprietary to Synopsys, Inc. and may only be used pursuant to the terms and conditions of a written license agreement with Synopsys, Inc. All other use, reproduction, modification, or distribution of the Synopsys software or the associated documentation is strictly prohibited.

Destination Control Statement

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

Disclaimer

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

Trademarks

Synopsys and certain Synopsys product names are trademarks of Synopsys, as set forth at <http://www.synopsys.com/company/legal/trademarks-brands.html>.

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

Free and Open-Source Software Licensing Notices

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

Third-Party Links

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

www.synopsys.com

Contents

About This Manual	xxx
Customer Support	xxxiii
1. Design Compiler Introduction	
About Design Compiler	1-2
Design Compiler NXT	1-2
Design Compiler Graphical	1-3
DC Ultra	1-4
DC Expert	1-5
The Design Compiler Family	1-5
About DC Explorer	1-6
About HDL Compiler	1-7
About Library Compiler	1-7
About Power Compiler	1-8
About DFT Compiler and DFTMAX	1-8
About Design Vision	1-8
Design Compiler in the Design Flow	1-9
High-Level Design Flow Tasks	1-10
Design Terminology	1-13
Designs	1-13
Design Objects	1-14
Design	1-14
Reference	1-14
Instance or Cell	1-15

Ports	1-15
Pins	1-15
Nets	1-15
Relationship Between Designs, Instances, and References	1-16
Selecting and Using a Compile Strategy	1-16
Optimization Basics	1-18
2. Working With Design Compiler	
Running Design Compiler	2-2
Design Compiler Modes	2-2
Wire Load Mode (Default)	2-3
Topographical Mode	2-3
Multimode	2-5
UPF Mode.	2-5
Working With Licenses	2-5
License Requirements	2-6
Enabling License Queuing	2-6
Listing the Licenses in Use	2-7
Checking Out Licenses	2-7
Checking DesignWare Licenses	2-8
Releasing Licenses.	2-8
The Setup Files	2-9
Naming Rules Section of the .synopsys_dc.setup File	2-10
Starting the Tool in Wire Load Mode	2-11
Starting the Tool in Topographical Mode	2-12
Design Compiler Startup Tasks	2-12
Entering dcnxt_shell or dc_shell Commands.	2-13
Redirecting the Output of Commands	2-14
Interrupting or Terminating Command Processing	2-14
Finding Session Information in the Log Files.	2-15
Command Log Files	2-15
Compile Log Files	2-15
File Name Log Files	2-17
Using Script Files.	2-17
Getting Help on the Command Line	2-18
Saving Designs and Exiting Design Compiler	2-18
The Synthesis Flow	2-19

A Design Compiler Session Example	2-23
Using Multicore Technology	2-25
Enabling Multicore Functionality	2-25
Measuring Runtime	2-26
Runtime Improvement Techniques Outside of Synthesis	2-27
Running Commands in the Background	2-27
Verification Guidance Support	2-28
Reporting Jobs Submitted To Run in the Background	2-28
Enabling Parallel Command Execution	2-29
Running Commands in Parallel in the Foreground	2-29
Supported Commands for Parallel Execution	2-30
Parallel Command Execution Design Flow	2-31
3. Preparing for Synthesis	
Managing the Design Data	3-2
Controlling the Design Data	3-2
Organizing the Design Data	3-3
Partitioning for Synthesis	3-4
Partitioning for Design Reuse	3-5
Keeping Related Combinational Logic Together	3-5
Registering Block Outputs	3-7
Partitioning by Design Goal	3-7
Partitioning by Compile Technique	3-8
Keeping Sharable Resources Together	3-8
Keeping User-Defined Resources With the Logic They Drive	3-9
Isolating Special Functions	3-10
HDL Coding for Synthesis	3-11
Writing Technology-Independent HDL	3-11
Inferring Components	3-12
Using HDL Constructs	3-14
General HDL Constructs	3-14
Using Verilog Macro Definitions	3-18
Using VHDL Port Definitions	3-18
Writing Effective Code	3-19
Guidelines for Identifiers	3-19
Guidelines for Expressions	3-20

Guidelines for Functions	3-21
Guidelines for Modules	3-22
Instantiating RTL PG Pins	3-22
Performing Design Exploration	3-23
Creating Constraints	3-24
4. Setting Up and Working With Libraries	
Selecting a Semiconductor Vendor	4-3
Library Requirements	4-3
Logic Libraries	4-4
Target Libraries	4-5
Link Libraries	4-6
Symbol Libraries	4-6
DesignWare Libraries	4-7
Physical Libraries	4-7
Specifying Logic Libraries	4-9
Specifying DesignWare Libraries	4-10
Specifying a Library Search Path	4-10
Setting Minimum Timing Libraries	4-11
Specifying Physical Libraries	4-11
Using TLUPlus Files for RC Estimation	4-12
Working With Libraries	4-13
Loading Libraries	4-14
Listing Libraries	4-14
Reporting Library Contents	4-14
Specifying Library Objects	4-15
Excluding Cells From the Target Libraries	4-15
Verifying Library Consistency	4-16
Removing Libraries From Memory	4-16
Saving Libraries	4-17
Target Library Subsets	4-17
Specifying Target Library Subsets	4-17
Setting Target Library Subset Examples	4-19
Checking Target Library Subsets	4-20

Reporting Target Library Subsets	4-20
Removing Target Library Subsets	4-20
Library Subsets for Sequential Cells and Instantiated Combinational Cells	4-20
Specifying the Library Cell Subsets	4-21
Reporting the Library Cell Subsets	4-22
Removing the Library Cell Subsets	4-23
Link Library Subsets	4-23
Specifying Link Library Subsets	4-23
Setting Link Library Subset Examples	4-25
Reporting Link Library Subsets	4-25
Removing Link Library Subsets	4-25
Library-Aware Mapping and Synthesis	4-26
Generating the ALIB File	4-26
Using the ALIB Library	4-26
Analyzing Multithreshold Voltage Library Cells	4-27
Handling Black Boxes	4-29
Supported Black Boxes	4-29
Black Box Flow	4-30
Defining Timing in Quick Timing Model Format	4-31
Defining Physical Dimensions	4-32
Estimating the Size of Black Boxes	4-32
Determining the Gate Equivalent Area	4-33
Identifying Black Box Cells	4-33
Automatic Creation of Physical Library Cells	4-34

5. Using a Milkyway Database

About the Milkyway Database	5-2
Required License and Files	5-2
Invoking the Milkyway Environment Tool	5-3
Guidelines for Using the Milkyway Database	5-3
Preparing to Use the Milkyway Database	5-4
Writing the Milkyway Database	5-5
Important Points About the write_milkyway Command	5-5
Limitations When Writing Milkyway Format	5-6

6. Working With Designs in Memory

Reading Designs	6-3
Supported Design Input Formats.....	6-3
Reading HDL Files.....	6-3
Reading Designs With Dependencies Automatically	6-4
Running the <code>read_file</code> Command	6-5
Running the <code>analyze</code> and <code>elaborate</code> Commands.....	6-5
Differences Between the <code>read_file</code> Command and the <code>analyze</code> and <code>elaborate</code> Commands	6-6
Running the <code>read_verilog</code> or <code>read_vhdl</code> Command.....	6-7
Reading .ddc Files	6-7
Reading .db Files	6-8
Listing Designs in Memory	6-8
Setting the Current Design	6-9
Linking Designs	6-9
How the Tool Resolves References.....	6-10
Locating Designs by Using a Search Path	6-11
Changing Design References	6-12
Querying Design References.....	6-13
Mapping Physical Variant Cells in Netlists or DEF Files	6-13
Listing Design Objects	6-14
Specifying Design Objects	6-15
Using a Relative Path	6-15
Using an Absolute Path	6-16
Using Attributes	6-17
Setting Attribute Values.....	6-18
Viewing Attribute Values	6-19
Saving Attribute Values.....	6-20
Defining Attributes.....	6-20
Propagating Attributes	6-21
Removing Attributes	6-22
The Object Search Order	6-22
Creating Designs	6-23
Copying Designs.....	6-23
Renaming Designs	6-24

Changing the Design Hierarchy	6-25
Adding Levels of Hierarchy	6-26
Removing Levels of Hierarchy	6-27
Ungrouping Hierarchies Before Optimization	6-27
Ungrouping Hierarchies Explicitly During Optimization	6-28
Ungrouping Hierarchies Automatically During Optimization	6-29
Preserving Hierarchical Pin Timing Constraints During Ungrouping	6-30
Merging Cells From Different Subdesigns	6-31
Editing Designs	6-31
Translating Designs From One Technology to Another	6-33
Translating Designs in Design Compiler Wire Load Mode	6-34
Translating Designs in Design Compiler Topographical Mode	6-34
Restrictions on Translating Between Technologies	6-35
Removing Designs From Memory	6-35
Saving Designs	6-36
Supported Design File Output Formats	6-36
Writing a Design Netlist or Schematic	6-37
Writing To a Milkyway Database	6-37
Saving Designs Using GUI Commands	6-37
Ensuring Name Consistency Between the Design Database and the Netlist	6-38
Specifying the Name Mapping and Replacement Rules	6-38
Resolving Naming Problems in the Flow	6-38
Avoiding Bit-Blasted Ports in SystemVerilog and VHDL Structures	6-39
Summary of Commands for Changing Names	6-40
7. Defining the Design Environment	
Operating Conditions	7-3
Defining Operating Conditions	7-3
Reporting Operating Conditions	7-4
Modeling the System Interface	7-4
Defining Drive Characteristics for Input Ports	7-5
Defining Loads on Input and Output Ports	7-7
Defining Fanout Loads on Output Ports	7-7
Setting Logic Constraints on Ports	7-8

Allowing Assignment of Any Signal to an Input	7-9
Specifying Input Ports as Always One or Zero	7-10
Wire Load Models.....	7-10
Hierarchical Wire Load Models	7-11
Determining Available Wire Load Models	7-13
Specifying Wire Load Models and Modes	7-14
Defining the Environment Using Topographical Mode	7-16
General Gate-Level Power Optimization.....	7-16
Power Correlation	7-17
Multivoltage Designs	7-17
Low Power Intent.....	7-18
Multicorner-Multimode Designs	7-19
Leakage Power and Dynamic Power Optimization.....	7-19
Leakage Power Optimization Based on Threshold Voltage.....	7-20
Multithreshold Voltage Library Attributes	7-20
Setting Multithreshold Voltage Constraints.....	7-21
Leakage Optimization for Multicorner-Multimode Designs	7-22
Comparing Design Compiler Topographical and IC Compiler Environments	7-22
Comparing Design Settings Between Design Compiler and IC Compiler II	7-23
8. Defining Design Constraints	
Constraint Types.....	8-2
Design Rule Constraints.....	8-3
Maximum Transition Time	8-4
Specifying Clock-Based Maximum Transition.....	8-5
Maximum Fanout	8-5
Maximum Fanout Calculation Example	8-6
Defining Maximum Fanout	8-7
Defining Expected Fanout for Output Ports	8-7
Maximum Capacitance	8-8
Specifying Frequency-Based Maximum Capacitance	8-8
Generating Maximum Capacitance Reports.....	8-9
Minimum Capacitance	8-10
Defining Minimum Capacitance	8-10

Cell Degradation	8-11
Connection Class	8-11
Summary of Design Rule Commands and Objects	8-12
Fixing Design Rule Violations	8-12
Disabling DRC Violation Fixing on Special Nets	8-13
Design Rule Constraint Precedence	8-14
Design Rule Scenarios	8-16
Optimization Constraints	8-16
Defining Timing Constraints	8-17
Maximum Delay	8-17
Minimum Delay	8-21
Defining Area Constraints (DC Expert Only)	8-22
Defining Power Constraints	8-23
Calculating Maximum Power Cost	8-23
Managing Constraint Priorities	8-24
Disabling the Cost Function	8-25
Reporting Constraints	8-26
Reporting Runtime Issues Related to SDC Constraints	8-27
Propagating Constraints in Hierarchical Designs	8-29
Characterizing Subdesigns	8-30
Using the characterize Command	8-30
Removing Previous Annotations	8-32
Optimizing Bottom Up Versus Optimizing Top Down	8-32
Deriving the Boundary Conditions	8-32
Saving Attributes and Constraints	8-33
The characterize Command Calculations	8-33
Characterizing Subdesign Port Signal Interfaces	8-37
Combinational Design Example	8-38
Sequential Design Example	8-39
Characterizing Subdesign Constraints	8-40
Characterizing Subdesign Logical Port Connections	8-40
Characterizing Multiple Instances	8-41
Characterizing Designs With Timing Exceptions	8-41
Limitations of the characterize Command	8-43
Propagating Constraints up the Hierarchy	8-43
Handling Conflicts Between Designs	8-44

9. Using Floorplan Physical Constraints

Importing Floorplan Information	9-3
Using the write_def Command in IC Compiler or IC Compiler II	9-4
Reading DEF Information in Design Compiler	9-4
Physical Constraints Imported in the DEF File	9-6
Extracting Physical-Only Cells From a DEF File	9-14
Macro and Port Name Matching With the extract_physical_constraints Command	9-14
Site Name Matching	9-15
Using the write_floorplan Command in IC Compiler	9-15
Reading the Floorplan Script in Design Compiler	9-16
Physical Constraints Imported in the Floorplan File	9-16
Macro and Port Name Matching With the read_floorplan Command	9-18
Reading and Writing Preroute Information for Power and Ground Nets and Physical-Only Cells	9-19
Manually Defined Physical Constraints	9-20
Defining Physical Constraints Overview	9-21
Defining the Die Area	9-23
Defining the Core Placement Area With the create_site_row Command	9-24
Defining Placement Area With the set_aspect_ratio and set_utilization Commands	9-24
Defining Port Locations	9-25
Defining Relative Port Locations	9-25
Defining Exact Port Locations	9-26
Defining Macro Location and Orientation	9-27
Defining Placement Blockages	9-28
Defining Voltage Areas	9-30
Defining Placement Bounds	9-31
Placement Bounds Overview	9-32
Creating Placement Bounds	9-32
Order for Creating Placement Bounds	9-35
Guidelines for Defining Placement Bounds Effectively	9-35
Returning a Collection of Bounds	9-36
Creating Wiring Keepouts	9-36
Creating Preroutes	9-37
Creating Preroutes for Power and Ground Nets	9-38
Creating User Shapes	9-39

Defining Physical Constraints for Pins	9-40
Creating Design Via Masters	9-41
Creating Vias	9-42
Creating Routing Tracks	9-42
Creating Keepout Margins	9-44
Computing Polygons	9-45
Including Physical-Only Cells	9-46
Specifying Physical-Only Cells Manually	9-47
Extracting Physical-Only Cells From a DEF File	9-48
Creating Collections With Physical-Only Cells	9-49
Reporting Physical-Only Cells	9-50
Saving Physical-Only Cells	9-51
Specifying Relative Placement	9-51
Relative Placement Overview	9-52
Benefits of Relative Placement	9-54
Methodology for the Relative Placement Flow	9-54
Relative Placement Flow Overview	9-55
Creating Relative Placement Using HDL Compiler Directives	9-57
Summary of Relative Placement Tcl Commands	9-57
Creating Relative Placement Groups	9-58
Anchoring Relative Placement Groups	9-59
Applying Compression to Relative Placement Groups	9-60
Specifying Alignment	9-61
Adding Objects to a Group	9-63
Querying Relative Placement Groups	9-70
Checking Relative Placement Constraints	9-71
Saving Relative Placement Information	9-72
Removing Relative Placement Group Attributes	9-73
Sample Script for a Relative Placement Flow	9-74
Magnet Placement	9-74
Resetting Physical Constraints	9-75
Saving Physical Constraints Using the write_floorplan Command	9-75
Saving Physical Constraints in IC Compiler Format	9-76
Saving Physical Constraints in IC Compiler II Format	9-76
Saving Physical Constraints Using the write_def Command	9-77
Reporting Physical Constraints	9-78

Reporting Routing Tracks	9-79
Reporting Preroutes	9-80
Reporting Design Via Masters	9-80
Reporting Keepout Margins	9-81

10. Compiling the Design

Compile Commands	10-2
The compile Command	10-2
The compile_ultra Command	10-3
Full and Incremental Compilation	10-5
Compile Strategies	10-6
Top-Down Compilation	10-8
Using the Top-Down Hierarchical Compile Strategy	10-8
Top-Down Hierarchical Compile Strategy Example	10-9
Bottom-Up Compilation	10-11
Using the Bottom-Up Hierarchical Compile Strategy	10-13
Bottom-Up Compile Script Example	10-14
Mixed Compile	10-15
Performing a Top-Level Compile	10-16
Using the -top Option With Other Compile Options	10-17
Limiting Optimization to Paths Within a Specific Range	10-17
Fixing Timing Violations For All Paths	10-17
Compile Log	10-18
Resolving Multiple Instances of a Design Reference	10-19
The Uniquify Method	10-20
Compile-Once-Don't-Touch Method	10-22
Ungroup Method	10-23
Test-Ready Compile	10-25

11. Optimizing the Design

Overview of the Optimization Process	11-2
Architectural Optimization	11-2
Logic-Level Optimization	11-3
Gate-Level Optimization	11-3

Optimization Phases	11-4
Combinational Optimization	11-5
Technology-Independent Optimization	11-6
Mapping	11-6
Technology-Specific Optimization	11-7
Sequential Optimization	11-7
Initial Sequential Optimization	11-7
Final Sequential Optimization	11-8
Local Optimizations	11-9
Compile Cost Function	11-10
Optimization Flow	11-11
Automatic Ungrouping	11-13
High-Level Optimization and Datapath Optimization	11-13
Multiplexer Mapping and Optimization	11-13
Sequential Mapping	11-13
Structuring and Mapping	11-14
Automatic Uniquification	11-14
Implementing Synthetic Parts	11-14
Timing-Driven Combinational Optimization	11-15
Register Retiming	11-15
Delay and Leakage Optimization	11-15
Design Rule Fixing	11-16
Area Optimization	11-16
Optimization Techniques	11-16
Optimizing Once for Best- and Worst-Case Conditions	11-17
Constraint-Related Commands	11-18
Reporting Commands	11-18
Optimizing With Multiple Libraries	11-19
Preserving Subdesigns	11-20
Preserving Cells, References, and Designs	11-20
Preserving Nets	11-21
Removing a dont_touch Setting	11-22
Preserving the Clock Network After Clock Tree Synthesis	11-22
Optimizing Datapaths	11-23
Creating Path Groups	11-24
Controlling the Optimization of Your Design	11-24
Optimizing Near-Critical Paths	11-25

Optimizing All Paths	11-26
Controlling Automatic Path Group Creation	11-27
Controlling Path Group Weights During High-Effort Area Optimization	11-28
Isolating Input and Output Ports	11-28
Examples	11-30
Removing and Reporting Port Isolation Cells	11-30
Fixing Heavily Loaded Nets	11-31
Fixing Nets Connected to Multiple Ports	11-32
Optimizing Buffer Trees	11-34
Building Balanced Buffer Trees	11-34
Reporting Buffer Trees	11-35
Removing Buffer Trees	11-36
Optimizing Multibit Registers	11-36
Optimizing for Multiple Clocks Per Register	11-39
Example	11-40
Defining a Signal for Unattached Master Clocks	11-41
Example 1	11-41
Example 2	11-42
Optimizing Total Negative Slack During Placement	11-43

12. Optimizing Across Hierarchical Boundaries

Boundary Optimization	12-2
Disabling Boundary Optimization Throughout the Design	12-3
Disabling Boundary Optimization for a Specific Design	12-4
Controlling Constant Propagation	12-5
Controlling Constant Propagation When Boundary Optimization is Disabled	12-5
Disabling Constant Propagation Through Specific Hierarchical Pins	12-6
Controlling Phase Inversion	12-6
Propagating Unconnected Registers and Unconnected Bits of Multibit Registers Across Hierarchies With Boundary Optimization Disabled	12-7
Port Punching in Design Compiler Graphical	12-7
Port Punching and Phase Inversion With Automatic High-Fanout Synthesis	12-8
Other Optimizations That Affect Hierarchical Boundaries	12-8
Automatic Ungrouping	12-8
Automatic Ungrouping of Hierarchies	12-9

Automatic Ungrouping of Designs With Timing Exceptions.....	12-10
Exceptions to Automatic Ungrouping.....	12-11
13. High-Level Optimization and Datapath Optimization	
Design Compiler Arithmetic Optimization	13-2
Synthetic Operators	13-3
High-Level Optimizations	13-4
Tree Delay Minimization and Arithmetic Simplifications.....	13-4
Resource Sharing	13-5
Common Subexpression Elimination	13-5
Sharing Mutually Exclusive Operations	13-6
Enhanced Resource Sharing	13-7
Datapath Optimization	13-8
Datapath Extraction.....	13-8
Datapath Implementation.....	13-10
Advanced Datapath Transformations	13-11
Analyzing Datapath Extraction	13-11
Reporting Resources and Datapath Blocks	13-13
14. Multiplexer Mapping and Optimization	
Inferring SELECT_OPs.....	14-3
Inferring MUX_OPs.....	14-4
Library Cell Requirements for Multiplexer Optimization	14-7
Mapping Multiplexers on Asynchronous Signal Lines.....	14-7
Mapping to One-Hot Multiplexers	14-8
Inferring One-Hot Multiplexers.....	14-8
Library Requirements for One-Hot Multiplexers	14-9
Optimization of One-Hot Multiplexers	14-11
15. Sequential Mapping	
Register Inference	15-3
Directing Register Mapping	15-5
Specifying the Default Flip-Flop or Latch	15-6

Specifying a Constant Value for Disabled Registers.....	15-6
Reporting Register Types	15-7
Reporting the Register Type Specifications for the Design	15-7
Reporting the Register Type Specifications for Cells.....	15-7
Unmapped Registers in a Compiled Design.....	15-8
Automatically Removing Unnecessary Registers.....	15-9
Removing Unconnected Registers	15-9
Eliminating Constant Registers	15-10
Controlling Constant Propagation Optimization	15-10
Constant Propagation Optimization for Complex Conditions	15-11
Merging Equal and Opposite Registers	15-12
Inverting the Output Phase of Sequential Elements.....	15-13
Mapping to Falling-Edge Flip-Flops	15-14
Resizing Black Box Registers.....	15-16
Preventing the Exchange of the Clock and Clock Enable Pin Connections	15-17
Mapping to Registers With Synchronous Reset or Preset Pins	15-17
Performing a Test-Ready Compile	15-21
Overview of Test-Ready Compile	15-22
Scan Replacement	15-23
Selecting a Scan Style.....	15-25
Mapping to Libraries Containing Only Scan Registers	15-26
Mapping to the Dedicated Scan-Out Pin	15-26
Automatic Identification of Shift Registers	15-27
Using Register Replication	15-32
Additional Register Replication Features in Topographical Mode	15-34
16. Adaptive Retiming	
Comparing Adaptive Retiming With Pipelined-Logic Retiming	16-2
Adaptive Retiming Examples	16-2
Performing Adaptive Retiming	16-6
Controlling Adaptive Retiming.....	16-6
Reporting the <code>dont_retime</code> Attribute	16-7

Removing the dont_retime Attribute	16-7
Verifying Retimed Designs	16-7
17. Pipelined-Logic Retiming	
Pipelined-Logic Retiming Overview	17-2
Pipelined-Logic Retiming Commands	17-3
Register Retiming Example	17-3
Register Retiming Concepts	17-5
Basic Definitions and Concepts	17-5
Flip-Flops and Registers	17-5
SEQGENs	17-6
Control Nets	17-7
Register Classes	17-8
Forward Retiming	17-9
Backward Retiming	17-10
Register Transformation Methods	17-11
Transforming Synchronous Input Pins Through Combinational Decomposition	17-11
Multiclass Retiming	17-14
Reset State Justification	17-15
Retiming the Design	17-16
Register Retiming Steps	17-16
Preventing Retiming	17-17
Selecting Transformation Options	17-18
Recommended Transformation Options for Pipelines	17-18
Recommended Transformation Options for Nonpipelines	17-19
Retiming Designs With Multiple Clocks	17-20
Retiming Registers With Path Group Constraints	17-21
Netlist Changes Performed by Register Retiming	17-24
Delay Threshold Optimization	17-24
Analyzing Retiming Results	17-24
Standard Output	17-25
Checking for Design Features That Limit the Quality of Results	17-25
Output Before Registers Are Moved	17-25
Output After Registers Are Moved	17-27
Displaying the Sequence of Cells That Limits Delay Optimization	17-28

Verifying Retimed Designs	17-30
18. Gate-Level Optimization	
Delay Optimization	18-2
Power Optimization.	18-4
Design Rule Fixing	18-5
Area Recovery	18-5
19. Using Topographical Technology	
Overview of Topographical Technology	19-3
Inputs and Outputs in Design Compiler Topographical Mode.	19-4
Defining the Design Environment	19-5
Performing Automatic High-Fanout Synthesis	19-6
Performing Manual High-Fanout Synthesis	19-7
Test Synthesis in Topographical Mode	19-7
Compile Flows in Topographical Mode	19-8
Performing an Incremental Compile	19-8
Performing a Bottom-up Hierarchical Compile	19-9
Overview of Bottom-Up Compile	19-10
Compiling the Subblock	19-12
Compiling the Design at the Top Level	19-15
Reducing Runtime	19-18
Handling Unsupported Commands, Options, and Variables.	19-21
Using the Design Compiler Graphical Tool	19-21
20. Using Design Compiler Graphical	
Using Synopsys Physical Guidance	20-2
Physical Guidance Overview	20-3
Reducing Routing Congestion	20-6
Routing Congestion Overview	20-6
Reducing Congestion in Highly Congested Designs	20-8
Enabling MUX Congestion Optimization.	20-8
Optimizing High-Pin-Density Cells	20-9

Reducing Congestion in Incremental Compile	20-9
Reducing Congestion by Optimizing RTL Structures	20-10
Specifying Block-Level Congestion Optimization	20-11
Controlling Congestion Optimization	20-12
Reporting Congestion	20-12
Viewing Congestion With the Design Vision Layout Window	20-14
Congestion Map Calculations	20-14
Reducing Congestion in the Floorplan	20-16
Controlling Placement Density	20-16
Separating High-Pin-Density Cells	20-16
Allowing Cell Clusters	20-17
Improving Placement of Wide Cells	20-17
Clustering Logic Modules to Minimize QoR Variations	20-18
Specifying Design Constraints and Power Settings	20-18
Design-Specific Settings	20-19
Physical Constraints	20-20
Power Optimization Settings	20-21
Using Layer Optimization to Increase the Accuracy of Net Delay Estimation	20-23
Enabling Net Promotion to Multiple Layers	20-24
Working With Nondefault Routing Rules	20-24
Defining Nondefault Routing Rules	20-25
Applying Nondefault Routing Rules	20-27
Reporting Nondefault Routing Rules	20-28
Removing Nondefault Routing Rules	20-28
Managing Nondefault Routing Rules in the Design Flow	20-29
Automatic Assignment of Nondefault Routing Rules on Nets	20-30
Enabling the Physical Guidance Flow	20-31
Enabling the Physical Guidance Incremental Flow	20-32
Using Physical Guidance in IC Compiler	20-34
Using the Design Compiler Graphical and IC Compiler Hierarchical Flow	20-35
Improved Timing Correlation Between Design Compiler Graphical and IC Compiler II	20-35
Incremental ASCII Flow With a Third-Party DFT Flow Example	20-36
Exporting the Design	20-37
Saving in Binary Format	20-38
Saving in ASCII Format	20-38
Saving in ASCII Format for IC Compiler II	20-39
Reporting Physical Guidance Information	20-42

Physical Guidance Limitations	20-43
Floorplan Exploration Floorplanning With IC Compiler	20-43
Floorplan Exploration Overview	20-43
Enabling Floorplan Exploration	20-44
Running Floorplan Exploration	20-46
Using the Floorplan Exploration GUI	20-47
Creating and Editing Floorplans	20-48
Analyzing the Data Flow for Macro Placement	20-49
Saving the Floorplan or Discarding Updates	20-49
Saving the Floorplan into a Tcl Script File or DEF File	20-50
Exiting the Session	20-50
Incremental or Full Synthesis After Floorplan Changes	20-51
Using Floorplan Exploration With a dc_shell Script	20-52
Black Box Support	20-54
Handling Physical Hierarchies and Block Abstractions	20-54
Floorplan Exploration Floorplanning With IC Compiler II	20-55
License Requirements	20-56
Prerequisites for Floorplan Exploration	20-56
Command Summary	20-57
Running the Floorplan Exploration Flow	20-58
In Interactive Mode	20-59
In Batch Mode	20-60
Data Transfer to IC Compiler II	20-61
Working With Via Ladders	20-62
Reading In Via Ladders Defined in the Technology File	20-62
Creating, Querying, Removing, and Reporting Via Ladders	20-62
Performing What-If Analysis With Via Ladders	20-63
Performing Via-Ladder-Aware Optimization	20-64
Assigning of Via Ladders on Nets	20-65
Working With Must-Join Pins	20-65
Querying and Reporting the pattern_must_join Attribute	20-66
Checking the Inconsistency of pattern_must_join Attributes on Library Pins	20-67
Inbound Cell Optimization	20-68
Optimizing Multicorner-Multimode Designs	20-70

Multicorner-Multimode Concepts	20-71
Multicorner-Multimode Feature Support	20-71
Unsupported Features for Multicorner-Multimode Designs	20-72
Basic Multicorner-Multimode Flow	20-72
Creating a Scenario	20-74
Concurrent Multicorner-Multimode Optimization and Timing Analysis	20-75
Power Optimization in Multicorner-Multimode Designs	20-75
Setting Up the Design for a Multicorner-Multimode Flow	20-77
Specifying TLUPlus Files	20-78
Specifying Operating Conditions	20-78
Specifying Constraints	20-79
Handling Libraries in the Multicorner-Multimode Flow	20-79
Using Link Libraries That Have the Same PVT Nominal Values	20-80
Using Unique PVT Names to Prevent Linking Problems	20-82
Unsupported k-factors	20-83
Automatic Detection of Driving Cell Library	20-84
Defining Minimum Libraries	20-84
Scenario Management Commands	20-85
Creating Scenarios	20-86
Defining Active Scenarios	20-86
Scenario Reduction	20-86
Specifying Scenario Options	20-87
Removing Scenarios	20-87
Reporting Commands for Multicorner-Multimode Designs	20-88
report_scenarios Command	20-88
report_scenario_options Command	20-89
Reporting Commands That Support the -scenario Option	20-89
Commands That Report the Current Scenario	20-90
Reporting Examples	20-91
Supported SDC Commands for Multicorner-Multimode Designs	20-95
Multicorner-Multimode Script Example	20-96
Using Block Abstractions in Multicorner-Multimode Designs	20-97
Methodology for Using Block Abstractions With Scenarios at the Top Level	20-98

21. Using Design Compiler NXT

Performing Buffering-Aware Placement With the IC Compiler II Link	21-2
Performing Automatic Timing Control With the IC Compiler II Link	21-2

Performing Congestion-Driven Restructuring With the IC Compiler II Link	21-3
Performing RC Extraction With the RC Model	21-4
Applying Required Settings for High Performance Cores.	21-4
22. Using Hierarchical Models	
Overview of Hierarchical Models	22-2
Information Used in Hierarchical Models	22-4
Using Hierarchical Models in a Multicorner-Multimode Flow	22-5
Viewing Hierarchical Models in the GUI	22-6
Block Abstraction Hierarchical Flow	22-6
Creating and Saving Block Abstractions	22-8
Information Used in Block Abstractions	22-9
Block Abstractions for Multicorner-Multimode Usage	22-10
Setting Top-Level Implementation Options	22-10
Transparent Interface Optimization	22-11
Creating Block Abstractions for Nested Blocks	22-13
Resetting Implementation Options.	22-14
Controlling the Extent of Logic Loaded for Block Abstractions	22-14
Loading Block Abstractions	22-16
Reporting Implementation Options	22-17
Reporting Block Abstractions.	22-17
Reporting Block Abstraction Area	22-19
Querying Block Abstractions	22-19
Checking Block Abstractions	22-20
Ignoring Timing Paths That Are Entirely Within Block Abstractions	22-20
Performing Top-Level Synthesis	22-20
Saving Optimized Block Abstractions After Top-Level Synthesis	22-21
Limitations	22-21
23. Incremental ASCII Flow With a Third-Party DFT Flow Example	
Performing Initial Compilation Using the <code>compile.tcl</code> Script.	23-3
Performing Incremental Compilation Using the <code>compile_incr.tcl</code> Script.	23-4
Using the Netlist SCANDEF Flow.	23-5
Identifying the Scan Chain Structure	23-6
Performing Scan Chain Reordering in Design Compiler Graphical	23-9

Writing Out the New Scan Chain Structure	23-10
--	-------

24. Analyzing and Resolving Design Problems

Resolving Bus Versus Bit-Blasted Mismatches Between the RTL and Macros	24-3
Fixing Errors Caused by New Unsupported Technology File Attributes	24-3
Using Register Replication to Solve Timing QoR, Congestion, and Fanout Problems	24-4
Assessing Design and Constraint Feasibility in Mapped Designs	24-4
Checking for Design Consistency	24-5
Checking Designs and Libraries Before Synthesis	24-7
Analyzing Your Design During Optimization Using the Compile Log	24-7
Analyzing Design Problems	24-9
Analyzing Area	24-10
Analyzing Timing	24-11
Reporting Quality of Results	24-12
Measuring Quality of Results	24-12
Analyzing Quality of Results	24-13
Displaying Quality of Results	24-17
Reporting Infeasible Paths	24-18
Reporting Infeasible Paths in an HTML Categorized Timing Report	24-19
Reporting Infeasible Paths in a Timing Report	24-20
Debugging Cells and Nets With <code>dont_touch</code>	24-21
Reporting <code>dont_touch</code> Cells and Nets	24-21
Creating a Collection of <code>dont_touch</code> Cells and Nets	24-23
Reporting <code>size_only</code> Cells	24-23
RTL Cross-Probing in the GUI	24-25
RTL Cross-Probing on the Command Line	24-26
Repeating RTL Cross-Probing	24-28
RTL Cross-Probing Report Example	24-29

25. Verifying Functional Equivalence

Setting Up SVF Generation	25-2
-------------------------------------	------

The guide_hier_map (GHM) Flow	25-2
The Formality Tool	25-3
Verification Guidance	25-4
Adjusting Optimization for Successful Verification	25-5
Using the set_verification_priority Command	25-5
Using Single-Pass Verification	25-6
Using Third-Party Formal Verification Tools	25-7

Appendix A. Design Example

Design Description	A-2
Setup File	A-9
Default Constraints File	A-10
Read Script	A-11
Compile Scripts	A-11

Appendix B. Basic Commands

Commands for Defining Design Rules	B-2
Commands for Defining Design Environments	B-2
Commands for Setting Design Constraints	B-3
Commands for Analyzing and Resolving Design Problems	B-4

Appendix C. Predefined Attributes

Appendix D. Latch-Based Design Code Examples

SR Latch	D-2
VHDL and Verilog Code Examples for SR Latch	D-2
Inference Report for an SR Latch	D-3
Synthesized Design for an SR Latch	D-3
D Latch	D-4
VHDL Code for a D Latch	D-4
Inference Report for a D Latch	D-5
Synthesized Design for a D Latch	D-6

D Latch With Asynchronous Reset	D-6
VHDL and Verilog Code for a D Latch With Asynchronous Reset	D-6
Inference Report for a D Latch With Asynchronous Reset	D-8
Synthesized Design for a D Latch With Asynchronous Reset	D-8
D Latch With Asynchronous Set and Reset	D-9
VHDL and Verilog Code for a D Latch With Asynchronous Set and Reset	D-9
Inference Report for a D Latch With Asynchronous Set and Reset	D-11
Synthesized Design for a D Latch With Asynchronous Set and Reset	D-12
D Latch With Enable (Avoiding Clock Gating)	D-12
VHDL and Verilog Code for a D Latch With Enable	D-13
Inference Report for a D Latch With Enable	D-14
Synthesized Design for a D Latch With Enable	D-15
Inferring Gated Clocks	D-15
Case 1	D-15
Case 2	D-16
Synthesized Design With Enable and Gated Clock	D-17
D Latch With Enable and Asynchronous Reset	D-18
VHDL and Verilog Code for a D Latch With Enable and Asynchronous Reset	D-18
Synthesized Design for a D Latch With Enable and Asynchronous Reset	D-20
D Latch With Enable and Asynchronous Set	D-20
VHDL and Verilog Code for a D Latch With Enable and Asynchronous Set	D-20
Synthesized Design for D Latch With Enable and Asynchronous Set	D-23
D Latch With Enable and Asynchronous Set and Reset	D-23
VHDL and Verilog Code for D Latch With Enable and Asynchronous Set and Reset	D-24
Synthesized Design for D Latch With Enable and Asynchronous Set and Reset	D-26

Preface

This preface includes the following sections:

- [About This Manual](#)
- [Customer Support](#)

About This Manual

The Synopsys Design Compiler® tool provides basic synthesis information for users of the Design Compiler tools. This manual describes synthesis concepts and commands, and presents examples for basic synthesis strategies.

This manual does not cover asynchronous design, I/O pad synthesis, test synthesis, simulation, or back-annotation of physical design information.

The information presented here supplements the Synopsys synthesis reference manuals but does not replace them. See other Synopsys documentation for details about topics not covered in this manual.

This manual supports the Synopsys synthesis tools, whether they are running under the UNIX operating system or the Linux operating system. The main text of this manual describes UNIX operation.

Audience

This manual is intended for logic designers and engineers who use the Synopsys synthesis tools with the VHDL or Verilog hardware description language (HDL). Before using this manual, you should be familiar with the following topics:

- High-level design techniques
- ASIC design principles
- Timing analysis principles
- Functional partitioning techniques

Related Publications

For additional information about the Design Compiler tool, see the documentation on the Synopsys SolvNet® online support site at the following address:

<https://solvnet.synopsys.com/DocsOnWeb>

You might also want to see the documentation for the following related Synopsys products:

- Design Vision™
- DesignWare® components
- DFT Compiler and DFTMAX™
- DC Explorer

- HDL Compiler™
- IC Compiler™
- Power Compiler™
- PrimeTime®

Release Notes

Information about new features, changes, enhancements, known limitations, and resolved Synopsys Technical Action Requests (STARs) is available in the *Design Compiler Release Notes* in SolvNet.

To see the *Design Compiler Release Notes*,

1. Go to the Download Center on SolvNet located at the following address:
<https://solvnet.synopsys.com/DownloadCenter>
2. Select Design Compiler, and then select a release in the list that appears.

Conventions

The following conventions are used in Synopsys documentation.

Convention	Description
Courier	Indicates syntax, such as <code>write_file</code> .
<i>Courier italic</i>	Indicates a user-defined value in syntax, such as <code>write_file design_list</code> .
Courier bold	Indicates user input—text you type verbatim—in examples, such as <code>prompt> write_file top</code>
[]	Denotes optional arguments in syntax, such as <code>write_file [-format fmt]</code>
...	Indicates that arguments can be repeated as many times as needed, such as <code>pin1 pin2 ... pinN</code>
	Indicates a choice among alternatives, such as <code>low medium high</code>
Ctrl+C	Indicates a keyboard combination, such as holding down the Ctrl key and pressing C.
\	Indicates a continuation of a command line.
/	Indicates levels of directory structure.
Edit > Copy	Indicates a path to a menu command, such as opening the Edit menu and choosing Copy.

Customer Support

Customer support is available through SolvNet online customer support and through contacting the Synopsys Technical Support Center.

Accessing SolvNet

SolvNet includes a knowledge base of technical articles and answers to frequently asked questions about Synopsys tools. SolvNet also gives you access to a wide range of Synopsys online services including software downloads, documentation, and technical support.

To access the SolvNet site, go to the following address:

<https://solvnet.synopsys.com>

If prompted, enter your user name and password. If you do not have a Synopsys user name and password, follow the instructions to register with SolvNet.

If you need help using the SolvNet site, click HELP in the top-right menu bar.

Contacting the Synopsys Technical Support Center

If you have problems, questions, or suggestions, you can contact the Synopsys Technical Support Center in the following ways:

- Open a support case to your local support center online by signing in to the SolvNet site at <https://solvnet.synopsys.com>, clicking Support, and then clicking “Open A Support Case.”
- Send an e-mail message to your local support center.
 - E-mail support_center@synopsys.com from within North America.
 - Find other local support center e-mail addresses at
<http://www.synopsys.com/Support/GlobalSupportCenters/Pages>
- Telephone your local support center.
 - Call (800) 245-8005 from within North America.
 - Find other local support center telephone numbers at
<http://www.synopsys.com/Support/GlobalSupportCenters/Pages>

1

Design Compiler Introduction

The Design Compiler product is the core of the Synopsys synthesis products. Design Compiler optimizes designs to provide the smallest and fastest logical representation of a given function. It comprises tools that synthesize your HDL descriptions into optimized, technology-dependent, gate-level designs. It supports a wide range of flat and hierarchical design styles and can optimize both combinational and sequential designs for speed, area, and power.

For an introduction to logic- and gate-level designs, the Design Compiler product, the design flow, and the Design Compiler family of products, see the following topics:

- [About Design Compiler](#)
- [The Design Compiler Family](#)
- [Design Compiler in the Design Flow](#)
- [High-Level Design Flow Tasks](#)
- [Design Terminology](#)
- [Selecting and Using a Compile Strategy](#)
- [Optimization Basics](#)

About Design Compiler

The Design Compiler product consists of the following:

- [Design Compiler NXT](#)
 - [Design Compiler Graphical](#)
 - [DC Ultra](#)
 - [DC Expert](#)
-

Design Compiler NXT

Design Compiler NXT provides all the Design Compiler Graphical, DC Ultra, and DC Expert features plus additional features. Design Compiler NXT provides

- Better QoR through advanced optimizations
- Enhanced physical guidance to IC Compiler II, with improved RC and timing correlation
- Faster runtime, with improved multithreading synthesis for better scalability
- Plug-and-play user interface and script compatibility with Design Compiler Graphical

The following Design Compiler NXT features are provided in addition to the Design Compiler Graphical, DC Ultra, and DC Expert features:

- Buffering-aware placement
- Automatic timing control
- Congestion-driven restructuring
- Better QOR with high performance cores

Running Design Compiler NXT requires a Design-Compiler-NXT license.

To invoke Design Compiler NXT, run the `dcnxt_shell` command with the `-topographical_mode` option in the UNIX or Linux shell. For information about using topographical mode, see [Design Compiler Modes](#). To perform synthesis, use the `compile_ultra` command with the `-spg` option.

See Also

- [Using Design Compiler NXT](#)

Design Compiler Graphical

Design Compiler Graphical provides all the DC Ultra and DC Expert features plus additional features. Design Compiler Graphical

- Optimizes multicorner-multimode designs
- Reduces routing congestion
- Improves correlation with IC Compiler and IC Compiler II
- Improves runtime in IC Compiler and IC Compiler II by using Synopsys physical guidance
- Allows you to create and modify floorplans

In addition, Design Compiler Graphical lets you create and modify floorplans using floorplan exploration.

The following Design Compiler Graphical features are provided in addition to the DC Expert and DC Ultra features:

- Optimization for multicorner-multimode designs
- Reduction of routing congestion during synthesis
- Improved area and timing correlation with IC Compiler
- Improved runtime and routability in IC Compiler
- Ability to create and modify floorplans using floorplan exploration
- Physical guidance technology, which includes enhanced placement and the capability to pass seed placement to IC Compiler to improve quality of results (QoR), correlation, and routability

Running Design Compiler Graphical requires a DC Ultra license and a Design Compiler Graphical license.

To invoke Design Compiler Graphical, run the `dc_shell` command with the `-topographical_mode` option in the UNIX or Linux shell. For information about using topographical mode, see [Design Compiler Modes](#). To perform synthesis, use the `compile_ultra` command with the `-spg` option.

See Also

- [Using Design Compiler Graphical](#)

DC Ultra

At the core of the Synopsys RTL synthesis solution is DC Ultra. DC Ultra provides all the DC Expert features plus additional features. It provides concurrent optimization of timing, area, power, and test for today's high performance designs. DC Ultra provides advanced delay and arithmetic optimization, automatic leakage power optimization, advanced timing analysis, register retiming, and more. By default, DC Ultra runs in wire load mode, which uses wire load models for delay estimation.

In addition, DC Ultra provides topographical technology, which allows you to accurately predict post-layout timing, area, and power during RTL synthesis without the need for timing approximations based on wire load models. This ensures better correlation with the final physical design. Topographical technology generates a better starting point for place and route, eliminating costly iterations.

The following list is an overview of the DC Ultra advanced features that are provided in addition to the DC Expert basic features:

- Infrastructure to support multicore execution for faster runtimes
- Advanced arithmetic optimization
- Integrated datapath partitioning and synthesis capabilities
- Advanced timing analysis
- Advanced delay optimization algorithms
- Automatic leakage power optimization
- Register retiming, the process by which the tool moves registers through combinational gates to improve timing

The following additional DC Ultra features are available when you run the tool in topographical mode:

- Support for multivoltage and multiple supply designs
- Concurrent multicorner-multimode optimization, which reduces iterations and provides faster time-to-results
- Placement and optimization technologies that are shared with the IC Compiler or IC Compiler II place and route tools to drive accurate timing and area prediction within synthesis, ensuring a better starting point for physical implementation

Running DC Ultra requires a DC Ultra license and a DesignWare Foundation license. Using topographical mode requires additional licensing.

To invoke DC Ultra, run the `dc_shell` command in the UNIX or Linux shell. To run the tool in topographical mode, you must also specify the `-topographical_mode` option. For

information about using topographical mode, see [Design Compiler Modes](#). To perform synthesis using DC Ultra in wire load mode or topographical mode, use the `compile_ultra` command.

See Also

- [The `compile_ultra` Command](#)
- [Overview of Topographical Technology](#)

DC Expert

DC Expert optimizes designs for area, timing, and power using wire load models for delay estimation. The tool provides the smallest and fastest logical representation of a given function. It comprises tools that synthesize your HDL descriptions into optimized, technology-dependent, gate-level designs. It supports a wide range of flat and hierarchical design styles and can optimize both combinational and sequential designs.

The following list provides an overview of the DC Expert features:

- Hierarchical compile (top down or bottom up)
- Full and incremental compile techniques
- Sequential optimization for complex flip-flops and latches
- Time borrowing for latch-based designs
- Timing analysis
- Command-line interface and graphical user interface

Running DC Expert requires a DC Expert license. To invoke DC Expert, run the `dc_shell` command in the UNIX or Linux shell. To perform synthesis, use the `compile` command.

The Design Compiler Family

The Design Compiler family of products provide an integrated RTL synthesis solution to address today's challenging IC designs.

Features provided by the Design Compiler family of tools include

- Early RTL exploration, which leads to a better starting point for RTL synthesis
- Minimized power consumption at the RTL and gate level, and concurrent timing, area, power, and test optimizations

- Advanced low-power methodologies with power intent defined by the standardized IEEE 1801 Unified Power Format (UPF)
- Transparent implementation of design-for-test capabilities without interfering with functional, timing, signal integrity, or power requirements
- The ability to predict, visualize, and alleviate routing congestion
- Floorplan exploration, which allows you to create and modify design floorplans

To learn about the Design Compiler family of products, see the following topics:

- [About DC Explorer](#)
- [About HDL Compiler](#)
- [About Library Compiler](#)
- [About Power Compiler](#)
- [About DFT Compiler and DFTMAX](#)
- [About Design Vision](#)

About DC Explorer

Developing new RTL and integrating it with third-party IP and many legacy RTL blocks can be a time-consuming process when designers lack a fast and efficient way to explore and improve the data, fix design issues, and create a better starting point for RTL synthesis. DC Explorer enables you to perform early RTL exploration, leading to a better starting point for RTL synthesis and accelerating design implementation.

DC Explorer provides the following features:

- Efficiently performs what-if analysis of various design configurations early in the design cycle, even with incomplete design data, to speed the development of high quality RTL description and constraints and drive a faster, more convergent design flow
- Generates an early netlist, which can be used to begin physical exploration in the IC Compiler or IC Compiler II tool
- Creates and modifies floorplans very early in the design cycle with access to IC Compiler or IC Compiler II design planning
- Performs preliminary synthesis quickly compared to full synthesis, yet gives you timing and area results typically within ten percent of the final results produced by Design Compiler in topographical mode

See Also

- The *DC Explorer User Guide*
-

About HDL Compiler

The HDL Compiler tool translates Verilog or VHDL descriptions into a generic technology (GTECH) netlist, which is used by Design Compiler to create an optimized netlist. After the design meets functionality, timing, power, and other design goals, you can read the gate-level netlist into IC Compiler and begin physical implementation.

See Also

- [HDL Coding for Synthesis](#)
 - The HDL Compiler documentation
-

About Library Compiler

Library Compiler reads the description of an ASIC library from a text file and compiles the description into either an internal database (.db file format) or into VHDL libraries. The compiled database supports synthesis tools. The VHDL libraries support VHDL simulation tools.

See Also

- The Library Compiler documentation

About Power Compiler

The Power Compiler tool offers a complete methodology for power, including analyzing and optimizing designs for static and dynamic power consumption. It performs RTL and gate-level power optimization and gate-level power analysis. By applying the power reduction techniques available in the Power Compiler tool, including clock-gating and multivoltage threshold group leakage power optimization, you can achieve power savings, area optimization, and timing optimization in synthesis.

See Also

- [Leakage Power and Dynamic Power Optimization](#)
 - [The Power Compiler User Guide](#)
-

About DFT Compiler and DFTMAX

The DFT Compiler tool is the Synopsys advanced test synthesis solution. It enables transparent implementation of design-for-test capabilities into the Synopsys synthesis flow without interfering with functional, timing, signal integrity, or power requirements.

DFTMAX compression provides synthesis-based scan compression to lower the cost of testing complex designs, particularly when fabricated with advanced process technologies. Designs using advanced process technologies can have subtle manufacturing defects that are only detected by applying at-speed and bridging tests, in addition to stuck-at tests. The extra patterns needed to achieve high test quality for these designs can increase both the test time and the test data, resulting in higher test costs. DFTMAX compression reduces these costs by delivering 10-100x test data and test time reduction with very low silicon area overhead. DFTMAX enables compressed scan synthesis in Design Compiler and compressed scan pattern generation in TetraMAX ATPG.

See Also

- [The DFT Compiler and DFTMAX documentation](#)
-

About Design Vision

The Design Vision tool is the graphical user interface (GUI) for the Synopsys logic synthesis environment and provides analysis tools for viewing and analyzing designs at the generic technology (GTECH) and gate level. The Design Vision main window provides menus and dialog boxes for running frequently used Design Compiler commands. It also provides graphical displays, such as histograms and schematics for visual analysis.

When you start the tool in topographical mode, the Design Vision layout window lets you analyze physical constraints, timing, and congestion in your floorplan. A layout view displays floorplan constraints, critical timing paths, and congested areas in a single, flat view of the physical design. This information can help you to guide later optimization operations in Design Compiler and other Synopsys tools.

Design Vision Help is available in the GUI from the Help menu. The Help system contains topics that explain the details of tasks that you can perform.

See Also

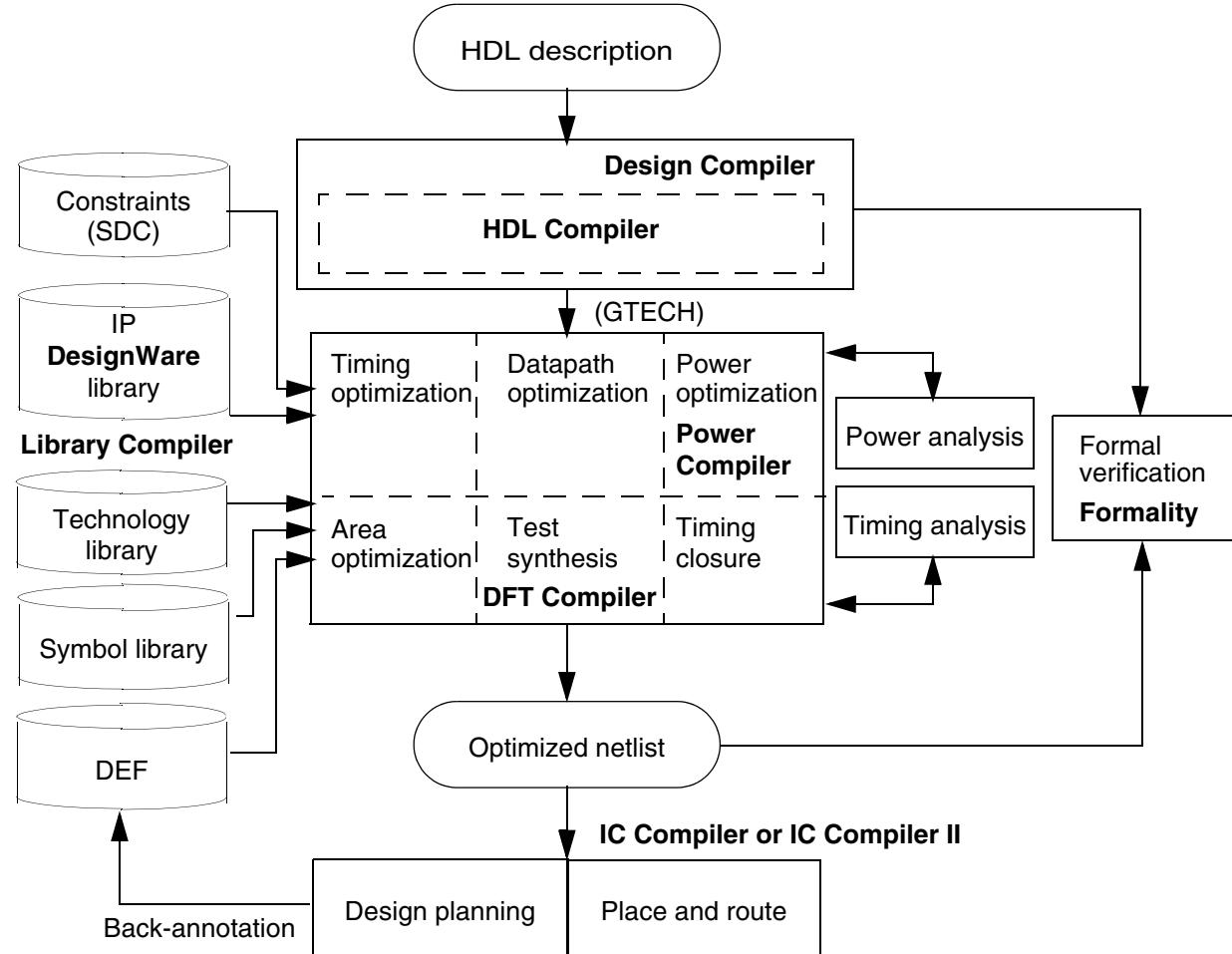
- The *Design Vision User Guide*
- Design Vision Help

Design Compiler in the Design Flow

You use Design Compiler for logic synthesis, which converts a design description written in a hardware description language, such as Verilog or VHDL, into an optimized gate-level netlist mapped to a specific logic library. When the synthesized design meets functionality, timing, power, and other design goals, you can pass the design to the IC Compiler or IC Compiler II tool for physical implementation.

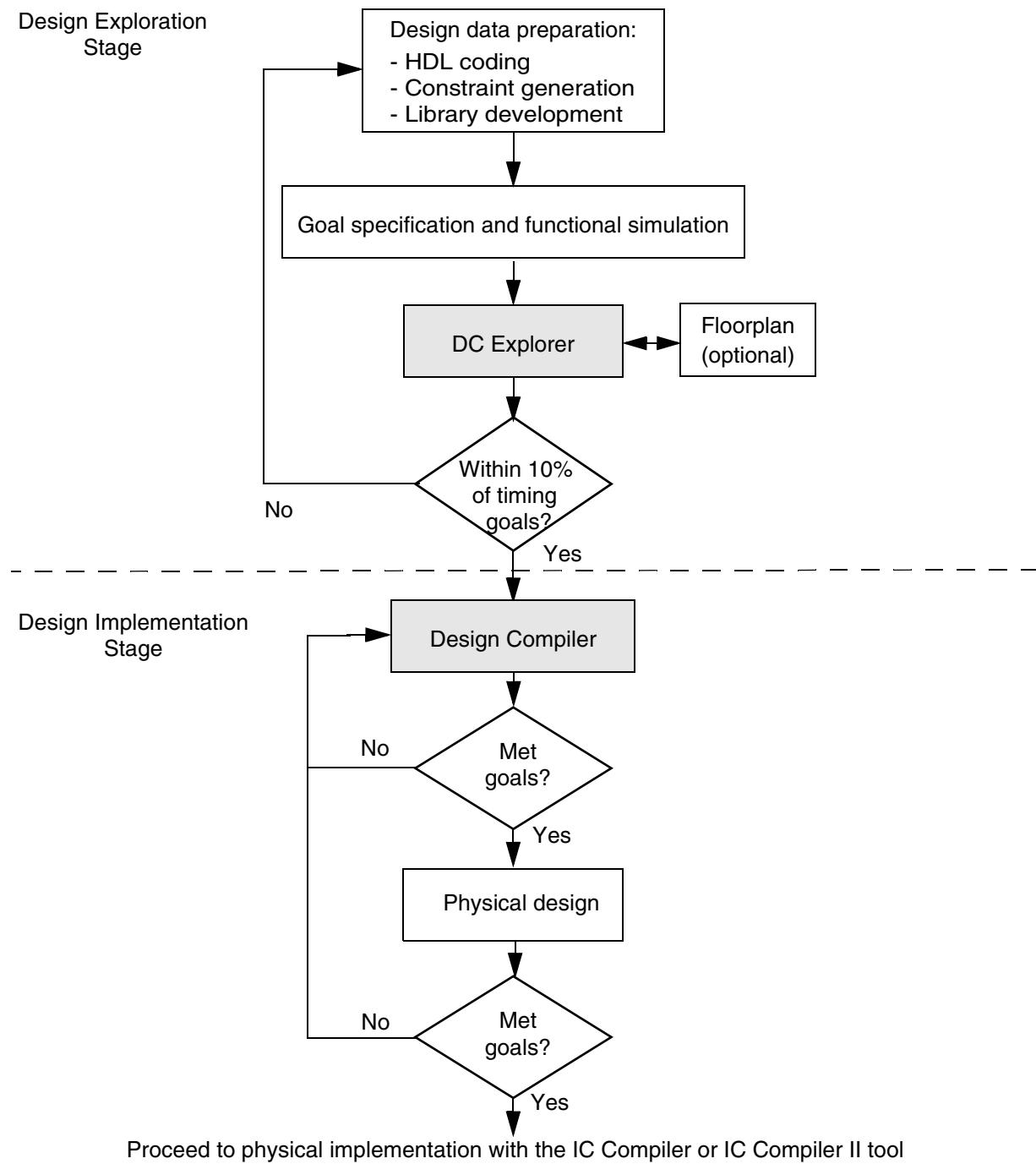
[Figure 1-1](#) shows an overview of how Design Compiler fits into the design flow.

Figure 1-1 Design Compiler in the Design Flow



High-Level Design Flow Tasks

[Figure 1-2](#) shows the high-level design flow from HDL coding to physical implementation in IC Compiler and IC Compiler II. The shaded areas indicate where the design exploration and synthesis tasks occur in the flow.

Figure 1-2 High-Level Design Flow

The synthesis design flow consists of the design exploration stage and the final design implementation stage. In the design exploration stage, you use DC Explorer to perform what-if analysis of various design configurations early in the design cycle to speed the

development of high-quality RTL and constraints and drive a faster, more convergent design flow. In the design implementation stage, you use the full power of Design Compiler to synthesize the design.

Using the high-level design flow shown in [Figure 1-2](#), you perform the following tasks:

1. Write an HDL description of your design in Verilog or VHDL. Use good coding practices to facilitate successful Design Compiler synthesis of the design.
2. Perform design exploration and functional simulation in parallel.
 - In design exploration, use DC Explorer to (a) implement specific design goals, such as design rules and optimization constraints, (b) detect mismatches and missing constraints, and (c) resolve mismatches and design data inconsistencies.

You can also create and modify floorplans early in the design cycle with floorplan exploration.

 - If design exploration fails to meet timing goals by more than 10 percent, modify your design goals and constraints, or improve the HDL code. Then repeat both design exploration and functional simulation.
 - In functional simulation, determine whether the design performs the desired functions by using an appropriate simulation tool.
 - If the design does not function as required, you must modify the HDL code and repeat both design exploration and functional simulation.
 - Continue performing design exploration and functional simulation until the design is functioning correctly and is within 10 percent of the timing goals.

3. Perform design implementation synthesis by using Design Compiler to meet design goals.

After synthesizing the design into a gate-level netlist, verify that the design meets your goals. If the design does not meet your goals, generate and analyze various reports to determine the techniques you might use to correct the problems.

4. After the design meets functionality, timing, power, and other design goals, proceed to the physical implementation stage in the IC Compiler or IC Compiler II tool.
5. Analyze the physical design's performance by using back-annotated data. If the results do not meet design goals, resolve them in IC Compiler or return to step 3. If the results meet your design goals, you are finished with the design cycle.

See Also

- [Design Compiler in the Design Flow](#)
- [The Synthesis Flow](#)

Design Terminology

Even though the following terms have slightly different meanings, they are often used synonymously in the Design Compiler documentation:

- *Synthesis* is the process that generates a gate-level netlist for an IC design that has been defined with a hardware description language (HDL). Synthesis includes reading the HDL source code and optimizing the design created from that description.
- *Optimization* is the step in the synthesis process that implements a combination of library cells that best meet the functional, timing, area, and power requirements of the design.
- *Compile* is the Design Compiler process that executes the synthesis and optimization steps. After you read in the design and perform other necessary tasks, you run the `compile_ultra` or `compile` command to generate a gate-level netlist for the design.

Different companies use different terminology for designs and their components. The following topics describe the terminology used in the Synopsys synthesis tools and the relationship between design instances and references.

- [Designs](#)
 - [Design Objects](#)
 - [Relationship Between Designs, Instances, and References](#)
-

Designs

Designs are circuit descriptions that perform logical functions. Designs are described in various design formats, such as VHDL or Verilog HDL. Logic-level designs are represented as sets of Boolean equations. Gate-level designs, such as netlists, are represented as interconnected cells.

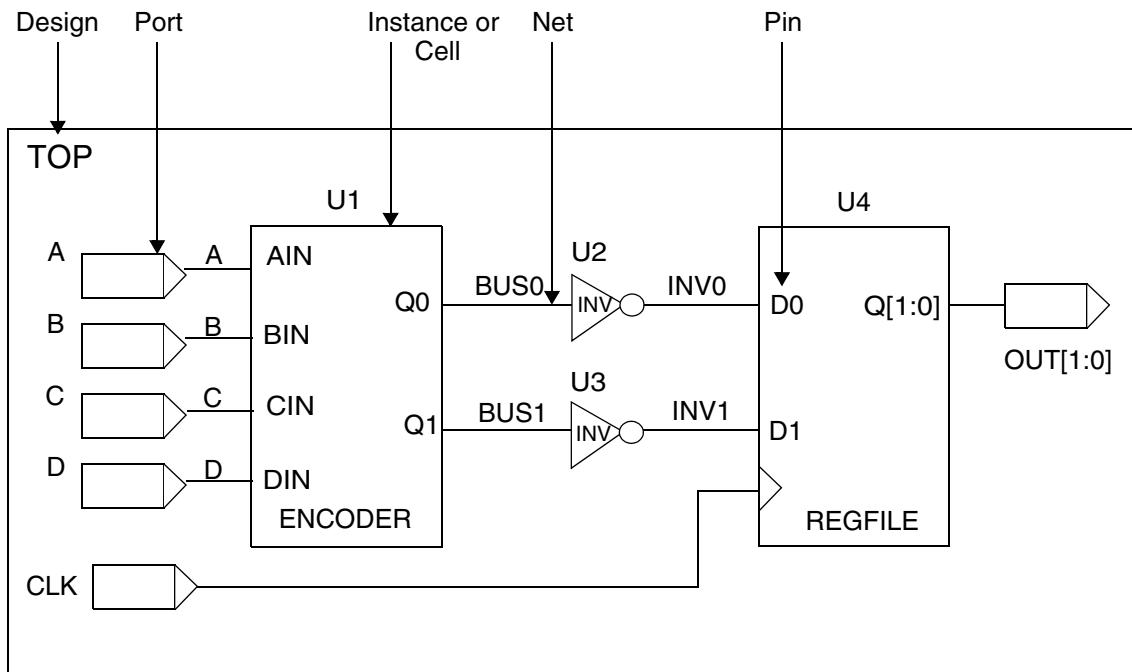
Designs can exist and be compiled independently of one another, or they can be used as subdesigns in larger designs. Designs are flat or hierarchical:

- **Flat Designs**
Flat designs contain no subdesigns and have only one structural level. They contain only library cells.
- **Hierarchical Designs**
Hierarchical designs contains one or more designs as subdesigns. Each subdesign can further contain subdesigns, creating multiple levels of design hierarchy. Designs that contain subdesigns are called parent designs.

Design Objects

Synopsys commands, attributes, and constraints are directed toward the specific design objects described in this topic. [Figure 1-3](#) shows the design objects in a design called TOP.

Figure 1-3 Design Objects



Design: {TOP, ENCODER, REGFILE}

Reference: {ENCODER, REGFILE, INV}

Instance: {U1, U2, U3, U4}

Design

A design consists of instances, nets, ports, and pins. It can contain subdesigns and library cells. In [Figure 1-3](#), the designs are TOP, ENCODER, and REGFILE. The active design (the design being worked on) is called the current design. Most commands are specific to the current design, that is, they operate within the context of the current design.

Reference

A reference is a library component or design that can be used as an element in building a larger circuit. The structure of the reference can be a simple logic gate or a more complex design (a RAM core or CPU). A design can contain multiple occurrences of a reference; each occurrence is an instance.

References enable you to optimize every cell (such as a NAND gate) in a single design without affecting cells in other designs. The references in one design are independent of the same references in a different design. In [Figure 1-3](#), the references are INV, ENCODER, and REGFILE.

Instance or Cell

An instance is an occurrence in a circuit of a reference (a library component or design) loaded in memory; each instance has a unique name. A design can contain multiple instances; each instance points to the same reference but has a unique name to distinguish it from other instances. An instance is also known as a cell.

A unique instance of a design within another design is called a hierarchical instance. A unique instance of a library cell within a design is called a leaf cell. Some commands work within the context of a hierarchical instance of the current design. The current instance defines the active instance for these instance-specific commands. In [Figure 1-3](#), the instances are U1, U2, U3, and U4.

Ports

Ports are the inputs and outputs of a design. The port direction is designated as input, output, or inout.

Pins

Pins are the input and output of cells (such as gates and flip-flops) within a design. The ports of a subdesign are pins within the parent design.

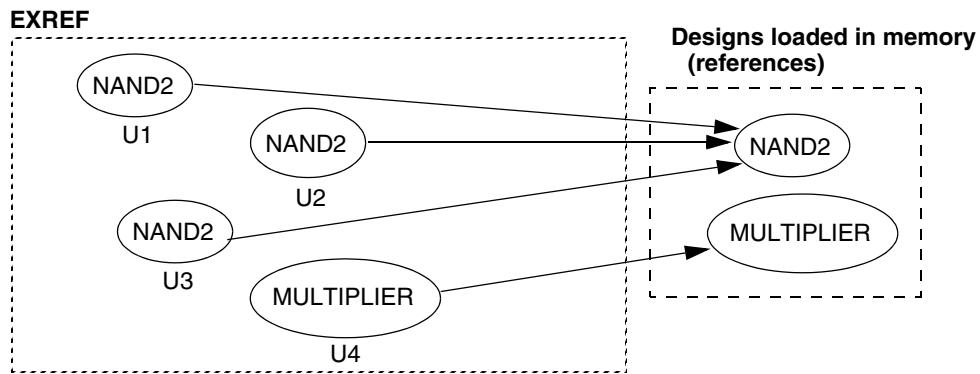
Nets

Nets are the wires that connect ports to pins and pins to each other.

Relationship Between Designs, Instances, and References

Figure 1-4 shows the relationships among designs, instances, and references.

Figure 1-4 Instances and References



The EXREF design contains two references: NAND2 and MULTIPLIER. NAND2 is instantiated three times, and MULTIPLIER is instantiated one time.

The names given to the three instances of NAND2 are U1, U2, and U3. The references of NAND2 and MULTIPLIER in the EXREF design are independent of the same references in different designs.

See Also

- [Linking Designs](#)
Provides information about resolving references
- [Working With Designs in Memory](#)
Provides information about changing designs, such as grouping or ungrouping subdesigns or changing subdesign references

Selecting and Using a Compile Strategy

To compile a design, you use the `compile` command in DC Expert or the `compile_ultra` command in DC Ultra or DC Graphical. Command options allow you to customize and control optimization. When you compile a design, Design Compiler reads the HDL source code and optimizes the design created from that description. The tool uses heuristics to implement a combination of library cells that meets the functional, speed, and area requirements of the design according to the attributes and constraints placed on it. The optimization process trades off timing and area constraints to provide the smallest possible circuit that meets the specified timing requirements.

Design Compiler supports the following compile flows:

- Full compile

During a full compile, Design Compiler maps and optimizes the entire design, resulting in a gate-level netlist of the design. Any mapped cells are unmapped to their logic function first; no attempt is made to preserve existing netlist structures.

- Incremental compile

During an incremental compile, Design Compiler can improve quality of results (QoR) by improving the structure of your design after the initial compile. Incremental mapping uses the existing gates from an earlier compilation as a starting point for the mapping process. It improves the existing design cost by focusing on the areas of the design that do not meet constraints and affects the second pass of compile. The existing structure is preserved where all constraints are already met. Mapping optimizations are accepted only if they improve the circuit speed or area.

In topographical mode, you can perform a second-pass, incremental compile to enable topographical-based optimization for post-topographical-based synthesis flows such as retiming, design-for-test (DFT), DFTMAX, and minor netlist edits. The primary focus in Design Compiler topographical mode is to maintain QoR correlation; therefore, only limited changes to the netlist can be made.

Different pieces of your design require different compilation strategies, such as a top-down hierarchical compile or bottom-up compile. You need to develop a compilation strategy before you compile.

You can use various strategies to compile, depending on your design, and you can mix strategies. The following strategies compile hierarchical designs in either wire load mode or topographical mode:

- Top-down compile

The top-level design and all its subdesigns are compiled together.

- Bottom-up compile

The individual subdesigns are compiled separately, starting from the bottom of the hierarchy and proceeding up through the levels of the hierarchy until the top-level design is compiled.

- Mixed compile

You can take advantage of the benefits of the top-down and the bottom-up compile strategies by using both strategies.

- Use the top-down compile strategy for small hierarchies of blocks.

- Use the bottom-up compile strategy to tie small hierarchies together into larger blocks.

See Also

- [Full and Incremental Compilation](#)

Provides more information about full and incremental compilation

- [Compile Strategies](#)

Provides detailed steps for top-down, bottom-up, and mixed compile strategies

- [Compile Flows in Topographical Mode](#)

Provides more information about performing an incremental compile and steps for performing a bottom-up (hierarchical) compile in topographical mode

Optimization Basics

Optimization is the Design Compiler synthesis step that maps the design to an optimal combination of specific target logic library cells, based on the design's functional, speed, and power requirements. You use the `compile` or the `compile_ultra` command to start the synthesis and optimization processes. Optimization transforms the design into a technology-specific circuit based on the attributes and constraints you place on the design.

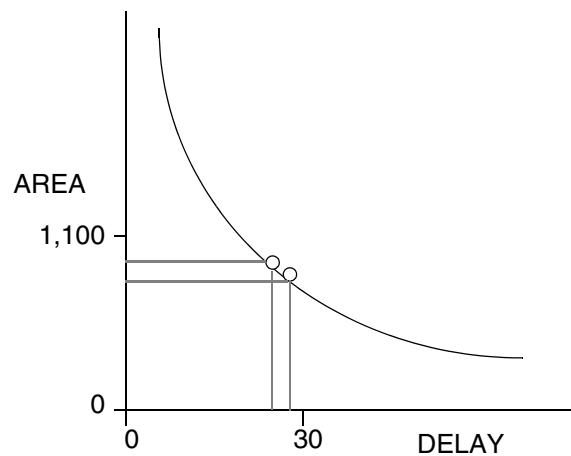
During optimization, Design Compiler uses heuristics to meet the constraints you have set on the design. Design Compiler's optimization algorithms use costs to determine if a design change is an improvement. Design Compiler calculates two cost functions: one for design rule constraints and one for optimization constraints. Optimization accepts a change if it decreases the cost of one component without increasing more-important costs. By default, the *design rule constraints* (transition, fanout, capacitance, and cell degradation) have a higher priority than the *optimization constraints* (delay and area).

Design Compiler performs the following levels of optimization in the following order:

1. [Architectural Optimization](#)
2. [Logic-Level Optimization](#)
3. [Gate-Level Optimization](#)

Experimenting with speed and area to get the smallest or fastest design is called *exploring the design space*. Using Design Compiler, you can examine different implementations of the same design in a relatively short time. [Figure 1-5](#) shows a design space curve. The shape of the curve demonstrates the tradeoff between area-efficient and speed-efficient circuits.

Figure 1-5 Design Space Curve



See Also

- [Optimization Constraints](#)
- [Optimizing the Design](#)

2

Working With Design Compiler

Design Compiler offers the following interfaces for synthesis and timing analysis: the dcnxt_shell and the dc_shell command-line interfaces, and the Design Vision user interface (GUI). The dcnxt_shell or dc_shell command-line interfaces are a text-only environments in which you enter commands at the command-line prompt. Design Vision is the GUI for the Synopsys logic synthesis environment; use it for visualizing design data and analyzing results.

To learn how to use the Design Compiler tools to run a synthesis flow, see the following topics:

- [Running Design Compiler](#)
- [The Synthesis Flow](#)
- [A Design Compiler Session Example](#)
- [Using Multicore Technology](#)
- [Runtime Improvement Techniques Outside of Synthesis](#)

Running Design Compiler

To learn how to run Design Compiler using dcnxt_shell or dc_shell, see the following topics:

- [Design Compiler Modes](#)
- [Working With Licenses](#)
- [The Setup Files](#)
- [Starting the Tool in Wire Load Mode](#)
- [Starting the Tool in Topographical Mode](#)
- [Design Compiler Startup Tasks](#)
- [Entering dcnxt_shell or dc_shell Commands](#)
- [Redirecting the Output of Commands](#)
- [Interrupting or Terminating Command Processing](#)
- [Finding Session Information in the Log Files](#)
- [Using Script Files](#)
- [Getting Help on the Command Line](#)
- [Saving Designs and Exiting Design Compiler](#)

You can also use the GUI to perform many of the tasks in this topic. For more information, see the *Design Vision User Guide*.

Design Compiler Modes

You can use Design Compiler in the following modes. Wire load mode and topographical mode are tool modes. When you start Design Compiler, you must choose either wire load mode or topographical mode. Multimode and UPF mode are not tool modes. Multimode allows you to operate the tool under multiple operating conditions and multiple modes, such as test mode and standby mode. UPF mode allows you to specify advanced low-power methodologies. Multimode and UPF mode are available only in topographical mode.

- [Wire Load Mode \(Default\)](#)
- [Topographical Mode](#)
- [Multimode](#)
- [UPF Mode](#)

Wire Load Mode (Default)

When you invoke Design Compiler with the `dcnxt_shell` or `dc_shell` command, the tool runs in the default mode, which uses wire load models for delay estimation:

```
% dcnxt_shell
```

or

```
% dc_shell
```

In this mode, you can run the following compilation commands:

- `compile`

The `compile` command runs DC Expert. DC Expert synthesizes your HDL descriptions into optimized, technology-dependent, gate-level designs. It supports a wide range of flat and hierarchical design styles and can optimize both combinational and sequential designs for area, timing, and power.

- `compile_ultra`

The `compile_ultra` command in wire load mode runs DC Ultra without the topographical mode features. DC Ultra provides concurrent optimization of timing, area, power, and test for high performance designs. It also provides advanced delay and arithmetic optimization, advanced timing analysis, automatic leakage power optimization, and register retiming. For more information, see [The compile_ultra Command](#).

See Also

- [DC Expert](#)
- [DC Ultra](#)

Topographical Mode

If you are using physical constraints on your design, you must use Design Compiler in topographical mode. Topographical mode allows you to accurately predict post-layout timing, area, and power during synthesis without the need for timing approximations based on wire load models. It uses placement and optimization technologies to drive accurate timing prediction within synthesis and automatically performs leakage power optimization, ensuring better correlation with the final physical design.

To invoke Design Compiler NXT in topographical mode, run the `dcnxt_shell` command with the `-topographical_mode` option:

```
% dcnxt_shell -topographical_mode
```

To invoke Design Compiler Graphical in topographical mode, run the `dc_shell` command with the `-topographical_mode` option:

```
% dc_shell -topographical_mode
```

In the topographical mode, you can run the following compilation commands:

- `compile_ultra`

Uses the DC Ultra tool with the topographical mode features enabled. Topographical technology derives a “virtual layout” of the design so the tool can accurately predict and use real net capacitances instead of statistical net approximations based on wire load models. This generates a better starting point for place and route, eliminating costly iterations. Topographical technology ensures better correlation with the final physical design.

- `compile_ultra -spg`

Uses the Design Compiler NXT and Design Compiler Graphical tools, which are the extension of DC Ultra topographical technology.

The Design Compiler NXT tool provides

- Better QoR through advanced optimizations
- Enhanced physical guidance to IC Compiler II, with improved RC and timing correlation
- Faster runtime, with improved multithreading synthesis for better scalability
- Plug-and-play user interface and script compatibility with Design Compiler Graphical
- Buffering-aware placement
- Automatic timing control
- Congestion-driven restructuring
- Better QOR with high performance cores

The Design Compiler Graphical tool

- Optimizes multicorner-multimode designs
- Reduces routing congestion
- Improves correlation with IC Compiler and IC Compiler II
- Improves runtime in IC Compiler and IC Compiler II by using Synopsys physical guidance
- Allows you to create and modify floorplans

See Also

- [DC Ultra](#)
- [Design Compiler Graphical](#)
- [Design Compiler NXT](#)

Multimode

Designs are often required to operate under multiple modes, such as test or standby mode, and under multiple operating conditions, sometimes referred to as corners. Such designs are known as multicorner-multimode designs. The Design Compiler NXT and Design Compiler Graphical tools can analyze and optimize designs across multiple modes and corners concurrently.

The multicorner-multimode feature in the Design Compiler NXT and Design Compiler Graphical tools provides compatibility between Design Compiler and IC Compiler flows.

For details about defining modes and corners and setting up multicorner-multimode analysis, see [Optimizing Multicorner-Multimode Designs](#).

UPF Mode

You can use IEEE 1801—also known as Unified Power Format (UPF)—commands in a UPF file to specify low-power design intent for multivoltage designs when you use the Power Compiler tool. The UPF set of Tcl-like commands provide the ability to specify the power intent early in the design process. In addition, UPF is supported throughout the design flow.

For information about multivoltage design concepts and using UPF mode to synthesize your multivoltage designs in Power Compiler, see the *Power Compiler User Guide*.

Working With Licenses

You need to determine which licenses are in use and know how to obtain and release licenses. You can use `dcnxt_shell` or `dc_shell` commands or the GUI to view, check out, and release licenses.

To learn about working with licenses, see the following topics:

- [License Requirements](#)
- [Enabling License Queuing](#)
- [Listing the Licenses in Use](#)
- [Checking Out Licenses](#)

- [Releasing Licenses](#)
- [Checking DesignWare Licenses](#)

License Requirements

You need the appropriate licenses to run Design Compiler in wire load or topographical mode. Design Compiler NXT, Design Compiler Graphical, DC Ultra, and DC Expert require their own licenses. Some features require additional licenses or add-ons to the Design Compiler NXT, Design Compiler Graphical, DC Ultra, and DC Expert licenses. To determine which licenses you need for your design flow, contact your Synopsys representative.

When you start the tool, Design Compiler automatically checks out the necessary license or licenses. When you exit from Design Compiler, the licenses are automatically checked in, allowing others at your site to use them.

Enabling License Queuing

Design Compiler has a license queuing functionality that allows your application to wait for licenses to become available if all licenses are in use. To enable this functionality, set the `SNPSLMD_QUEUE` environment variable to `true` before you start the Design Compiler tool. When you invoke the tool, Design Compiler displays the following message:

Information: License queuing is enabled. (DCSH-18)

When you have enabled the license queuing functionality, you might run into a situation where you hold license L1 while waiting for license L2, and another user holds license L2 while waiting for license L1. In that case, both you and the other user might wait indefinitely, unless more licenses become available.

To prevent such situations, use the `SNPS_MAX_WAITTIME` and the `SNPS_MAX_QUEUEETIME` environment variables. You must set the `SNPSLMD_QUEUE` environment variable to `true` before using these two variables.

- The `SNPS_MAX_WAITTIME` variable specifies the maximum wait time in seconds for the first license that you require.
- The `SNPS_MAX_QUEUEETIME` variable specifies the maximum wait time in seconds for checking out subsequent licenses within the same `dcnxt_shell` or `dc_shell` process. You use this variable after you have successfully checked out the first license to start `dcnxt_shell` or `dc_shell`.

Consider the following scenario: You have already started Design Compiler and are running a command that requires a DC-Ultra-Features license. The queuing functionality attempts to check out the license within the specified wait time. The default is 28,800 seconds (or eight hours). If the license is still not available after the predefined time, you might see a message similar to the following:

Information: Timeout while waiting for feature 'DC-Ultra-Features.'
(DCSH-17)

When you run your design through the synthesis flow, the queuing functionality might display other status messages as follows:

Information: Started queuing for feature 'HDL-Compiler'. (DCSH-15)
Information: Still waiting for feature 'HDL-Compiler'. (DCSH-16)
Information: Successfully checked out feature 'HDL-Compiler'. (DCSH-14)

Listing the Licenses in Use

To view the licenses that you currently have checked out using Design Compiler in wire load or topographical mode, use the following command:

```
prompt> list_licenses
```

```
Licenses in use:  
    DC-Extension  
    Design-Compiler-NXT  
    DC-Expert (3)  
    DC-Ultra-Features (3)  
    DC-Ultra-Opt (3)  
    Design-Compiler  
    DesignWare
```

```
1
```

To display which licenses are already checked out, use the following command:

```
prompt> license_users
```

```
bill@eng1 Design-Compiler  
matt@eng2 Design-Compiler, DC-Ultra-Opt  
2 users listed.  
1
```

Checking Out Licenses

When you invoke Design Compiler in topographical or wire load mode, the Synopsys Common Licensing software automatically checks out the appropriate license. For example, if you read in an HDL design description, Synopsys Common Licensing checks out a license for the appropriate HDL compiler.

If you know the tools and interfaces you need, you can use the `get_license` command to check out those licenses. This ensures that each license is available when you are ready to use it. By default, only one license is checked out for each feature. After a license is checked out, it remains checked out until you release it or exit `dcnxt_shell` or `dc_shell`.

If multiple licenses are required for a multicore run, use the `-quantity` option to specify the total number of licenses needed. If licenses have already been checked out, Design

Compiler acquires only the additional licenses needed to bring the total to the specified quantity.

In the following example, Design Compiler in topographical mode checks out a license for the multivoltage feature:

```
prompt> get_license Galaxy-MV
```

In the following example, Design Compiler in topographical mode checks out two of the licenses required to run a multicore `compile_ultra` command:

```
prompt> get_license -quantity 2 \
           {DC-Expert DC-Ultra-Opt DC-Ultra-Features}
```

The tool checks out a copy of the license if one is available or displays an error message if all the licenses are already taken.

Checking DesignWare Licenses

If any DesignWare component set in the `synthetic_library` variable requires a DesignWare license, Design Compiler checks for this license. You do not need to specify the standard synthetic library, `standard.sldb`, that implements the built-in HDL operators. Design Compiler automatically uses this library.

You can force Design Compiler to wait for a DesignWare license by setting the `synlib_wait_for_design_license` variable to DesignWare as follows:

```
prompt> set_app_var synlib_wait_for_design_license "DesignWare"
```

See Also

- [DesignWare Libraries](#)

Releasing Licenses

To release a license that is checked out to you in topographical or wire load mode, use the following command:

```
prompt> remove_license HDL-Compiler
```

For multicore runs, where multiple licenses might be required for certain features, use the `-keep` option to specify how many licenses should be retained for each feature after the command has completed.

The following example removes some, but not all, of the licenses required for a multicore `compile_ultra` run in Design Compiler:

```
prompt> list_licenses
Licenses in use:
    DC-Extension
```

```
Design-Compiler-NXT
DC-Expert (4)
DC-Ultra-Features (4)
DC-Ultra-Opt (4)
Design-Compiler
1
prompt> remove_license -keep 2 \
           {DC-Expert DC-Ultra-Opt DC-Ultra-Features}

prompt> list_licenses
Licenses in use:
    DC-Extension
    Design-Compiler-NXT
    DC-Expert (2)
    DC-Ultra-Features (2)
    DC-Ultra-Opt (2)
    Design-Compiler
1
```

The Setup Files

Before starting Design Compiler, make sure your `$SYNOPSYS` variable is set, and the path to the bin directory is included in your `$PATH` variable. Be sure to specify the absolute path to indicate the Synopsys root that contains the Design Compiler installation, as shown:

```
/tools/synopsys/2014.09/bin/
```

If you use a relative path `(..)`, as shown, Design Compiler cannot access the libraries that are located in the root directory:

```
../../../../2014.09/bin/
```

When you invoke Design Compiler in wire load or topographical mode, it automatically executes commands in three setup files. These files have the same file name, `.synopsys_dc.setup`, but reside in different directories. The files can contain commands that initialize parameters and variables, declare design libraries, and so on.

Design Compiler reads the setup files from three directories in the following order:

1. The Synopsys root directory (`$SYNOPSYS/admin/setup`)

This system-wide setup file contains system variables defined by Synopsys and general Design Compiler setup information for all users at your site. Only the system administrator can modify this file.

2. Your home directory

This user-defined setup file can contain variables that define your preferences for the Design Compiler working environment. The variables in this file override the corresponding variables in the system-wide setup file.

3. The current working directory (the directory from which you start Design Compiler)

This design-specific setup file can contain project-specific or design-specific variables that affect the optimizations of all designs in this directory. To use the file, you must invoke Design Compiler from this directory. Variables defined in this file override the corresponding variables in the user-defined and system-wide setup files.

[Example 2-1](#) shows a .synopsys_dc.setup file.

Example 2-1 .synopsys_dc.setup File

```
# Define the target logic library, symbol library,
# and link libraries
set_app_var target_library lsi_10k.db
set_app_var symbol_library lsi_10k.sdb
set_app_var synthetic_library dw_foundation.sldb
set_app_var link_library "* $target_library $synthetic_library"
set_app_var search_path [concat $search_path ./src]
set_app_var designer "Your Name"

# Define aliases
alias h history
alias rc "report_constraint -all_violators"
```

Some ASIC and EDA vendors have a program that creates a .synopsys_dc.setup file that includes the appropriate commands to convert names to their conventions. For an example of a naming rules section in a .synopsys_dc.setup file, see [Example 2-2](#).

See Also

- [*Synthesis Tools Installation Notes*](#)
Provides information about defining the \$PATH and \$SYNOPSYS variables
- [*Naming Rules Section of the .synopsys_dc.setup File*](#)

Naming Rules Section of the .synopsys_dc.setup File

[Example 2-2](#) shows sample naming rules created by a specific layout tool vendor. These naming rules do the following:

- Limit object names to alphanumeric characters
- Change DesignWare cell names to valid names (changes “*cell*” to “U” and “*-return” to “RET”)

Your vendor might use different naming conventions. Check with your vendor to determine the naming conventions you need to follow. If you need to change any net or port names, use the `define_name_rules` and `change_names` commands.

Example 2-2 Naming Rules Section of .synopsys_dc.setup File

```
define_name_rules simple_names -allowed "A-Za-z0-9_" \
    -last_restricted "_" \
    -first_restricted "U" \
    -map {"{*cell*","U"}, {"*-return","RET"} } }
```

Starting the Tool in Wire Load Mode

You start Design Compiler by entering the following commands in a UNIX or Linux shell:

- The `dcnxt_shell` command.

```
% dcnxt_shell
```

Be sure to specify the absolute path to indicate the Synopsys root that contains the Design Compiler NXT installation, as shown:

```
% /tools/synopsys/2019.03/bin/dcnxt_shell
```

If you use a relative path `(..)`, as shown, Design Compiler NXT cannot access the libraries that are located in the root directory:

```
% ../../2019.03/bin/dcnxt_shell
```

- The `dc_shell` command.

```
% dc_shell
```

Be sure to specify the absolute path to indicate the Synopsys root that contains the Design Compiler installation, as shown:

```
% /tools/synopsys/2019.03/bin/dc_shell
```

If you use a relative path `(..)`, as shown, Design Compiler cannot access the libraries that are located in the root directory:

```
% ../../2019.03/bin/dc_shell
```

To execute a script file before displaying the initial `dcnxt_shell` or `dc_shell` prompt, specify the `-f` option with the `dcnxt_shell` or `dc_shell` command. To include any command statements that you want to be executed automatically at startup, specify the `-x` option.

You can also open the GUI when you start Design Compiler by specifying the `-gui` option with the `dcnxt_shell` and `dc_shell` commands. You must have a Design Vision license to use the GUI from a `dcnxt_shell` and `dc_shell` session.

You can open the GUI at any time from the Design Compiler command-line interface by entering the `gui_start` command at the `dcnxt_shell` or `dc_shell` prompt:

```
prompt> gui_start
```

For more information about using the GUI, see the *Design Vision User Guide*.

For information about the options you can use when you start the tool, see the `dcnxt_shell` and `dc_shell` man pages.

When you start the command-line interface, the `dcnxt_shell` or `dc_shell` prompt appears in the UNIX or Linux shell:

```
dcnxt_shell>
```

or

```
dc_shell>
```

Starting the Tool in Topographical Mode

If you are using Design Compiler in topographical mode, following the instructions in [Starting the Tool in Wire Load Mode](#) and specify the `-topographical_mode` option with the `dcnxt_shell` or `dc_shell` command:

For Design Compiler NXT,

- % `dcnxt_shell -topographical_mode`

The resulting command prompt is

```
dcnxt_shell-topo>
```

For Design Compiler Graphical,

- % `dc_shell -topographical_mode`

The resulting command prompt is

```
dc_shell-topo>
```

Design Compiler Startup Tasks

At startup, `dcnxt_shell` or `dc_shell` does the following tasks:

1. Creates a command log file.
2. Reads and executes the `.synopsys_dc.setup` files, as described in [The Setup Files](#).
3. Executes any script files or commands specified by the `-f` and `-x` options, respectively, on the command line.
4. Displays the program header and `dcnxt_shell` or `dc_shell` prompt in the window from which you invoked `dcnxt_shell` or `dc_shell`. The program header lists all features for which your site is licensed.

Entering dcnxt_shell or dc_shell Commands

Design Compiler shell (dcnxt_shell or dc_shell) commands are based on the tool command language (Tcl), including certain command extensions needed to implement specific Design Compiler functionality. The Design Compiler command language provides capabilities similar to UNIX command shells, including variables, conditional execution of commands, and control flow commands. You can

- Enter individual commands interactively at the dcnxt_shell or dc_shell prompt
- Run one or more Tcl command scripts, which are text files that contain dcnxt_shell or dc_shell commands

You enter commands in dcnxt_shell or dc_shell the same way you enter commands in a standard UNIX or Linux shell. When entering a command, option, or file name, you can minimize your typing by pressing the Tab key when you have typed enough characters to specify a unique name; Design Compiler completes the remaining characters. If the characters you typed could be used for more than one name, Design Compiler lists the qualifying names from which you can select by using the arrow keys and the Enter key.

To get a list of all dcnxt_shell or dc_shell commands on the command line, enter

```
prompt> help
```

You can reuse a command from the output for a command-line interface by copying and pasting it to the dcnxt_shell or dc_shell command line.

When the GUI is open, you can enter commands on the console command line and use the commands available through the menu interface. You can select commands in the console history view and either rerun them or copy them to the command line, where you can edit them. For more information, see the “Viewing the Command History” topic in Design Vision Help.

See Also

- *Using Tcl With Synopsys Tools*

Redirecting the Output of Commands

You can redirect or append the output of the commands to a file you can review. This way, you can archive runtime messages for future reference.

Table 2-1 Redirecting the Command Output

To do this	Use this
Divert command output to a file.	> (redirection operator)
Append command output to a file.	>> (append operator)
Redirect command output to a file.	The <code>redirect</code> command

Note:

The pipe character (|) has no meaning in the `dcnxt_shell` or `dc_shell` interface.

Interrupting or Terminating Command Processing

If you enter the wrong options for a command or enter the wrong command, you can interrupt command processing and remain in `dcnxt_shell` or `dc_shell`. To interrupt or terminate a command, press **Ctrl+C**.

Some commands and processes, such as the `update_timing` command, cannot be interrupted. To stop these commands or processes, you must terminate `dcnxt_shell` or `dc_shell` at the system level. When you terminate a process or the shell, no data is saved.

When you press **Ctrl+C**, remember the following points:

- If a script file is being processed and you interrupt one of its commands, the script processing is interrupted and no further script commands are processed.
- If you press **Ctrl+C** three times before a command responds to your interrupt, `dcnxt_shell` or `dc_shell` is interrupted and exits with the following message:

Information: Process terminated by interrupt.

This behavior has a few exceptions, which are documented in the man pages for the applicable commands.

Finding Session Information in the Log Files

You can find session information, such as the `dcnxt_shell` or `dc_shell` commands that were processed and the files that were accessed, in the following log files:

- [Command Log Files](#)
- [Compile Log Files](#)
- [File Name Log Files](#)

Command Log Files

The command log file records the `dcnxt_shell` or `dc_shell` commands processed by Design Compiler, including setup file commands and variable assignments. By default, Design Compiler writes the command log to a file called `command.log` in the directory from which you invoked `dcnxt_shell` or `dc_shell`.

You can change the name of the `command.log` file by setting the `sh_command_log_file` variable in the `.synopsys_dc.setup` file. You should make any changes to this variable before you start Design Compiler. If your user-defined or project-specific `.synopsys_dc.setup` file does not define the variable, Design Compiler automatically creates the `command.log` file.

Each Design Compiler session overwrites the existing command log file. To save a command log file, move or rename it. You can use the command log file to

- Produce a script for a particular synthesis strategy
- Record the design exploration process
- Document any problems you are having

Compile Log Files

Each time you compile a design, Design Compiler creates the following compile log files:

- ASCII log file

The log displays the output, such as the commands that are processed and the error messages for each Design Compiler run, on the screen for quick viewing and debugging.

For example, each time Design Compiler begins a step, it prints a message in the compile log, indicating its progress:

```
prompt> compile_ultra
Beginning Mapping Optimizations (Medium effort)
```

Trials	Area	Delta delay	Total neg slack	Design rule cost
3	1296477.2	7.58	3468.1	2.9
1	1296538.9	7.48	3382.7	2.9

- **HTML log file**

The file resides under the current working directory. It contains the complete contents of the ASCII log but in HTML format. At the end of this file, a summary table provides an overview of all occurrences of messages grouped by the message ID. You can click the message ID link to display the message.

To generate the HTML log file in dcnxt_shell or dc_shell, set the `html_log_enable` variable to `true` before reading in the design. For example,

```
prompt> set_app_var html_log_enable true
```

By default, the `html_log_enable` variable is set to `false`, and only the ASCII log file is generated. The contents of the ASCII and HTML log files are identical. However, the way the contents are displayed is different. In the HTML log file, you can control the level of detail that is displayed by clicking the plus (+) or minus (-) buttons to expand or collapse multiple lines of messages.

By default, the HTML file name is `default.html`. To specify a different file name, set the `html_log_filename` variable. For example,

```
prompt> set_app_var html_log_filename my_HTML_log.html
```

Important:

You must have the Python programming language installed to generate an HTML log file. For download information, go to the following address:

<http://www.python.org/download>

See Also

- [Analyzing Your Design During Optimization Using the Compile Log](#)

File Name Log Files

By default, Design Compiler lists the names of the files that it has read to a log file in the directory from which you invoked dcnxt_shell or dc_shell. You can use the log file to identify data files needed to reproduce an error if Design Compiler terminates abnormally. To specify the name of the log file, set the `filename_log_file` variable in the `.synopsys_dc.setup` file.

Using Script Files

Scripts are used to accomplish repetitive routines, such as setting constraints or defining other design attributes. You can use your existing Tcl scripts in the Design Compiler command-line interface and GUI.

You can create a script file by placing a sequence of dcnxt_shell or dc_shell commands in a text file. You can also define scripts in your setup files. Any dcnxt_shell or dc_shell command can be executed within a script file.

In Tcl, a pound sign (#) at the beginning of a line denotes a comment:

```
# This is a comment.
```

To execute a script file before displaying the initial dcnxt_shell or dc_shell prompt, specify the `-f` option with the `dcnxt_shell` or `dc_shell` command. For example,

```
% dc_shell -f script_file_name
```

To execute a script file within the tool, use the `source` command at the command prompt and specify the file name:

```
prompt> source script_file_name
```

For example, the following command executes the `dc.tcl` top-down compile script in topographical mode that is shown in [Example 2-3](#):

```
prompt> source dc.tcl
```

See Also

- [Using Tcl With Synopsys Tools](#)
- The “Using Tcl Scripts” topic in Design Vision Help

Getting Help on the Command Line

Design Compiler provides a variety of user-assistance tools. The following online information resources are available while you are using the Design Compiler tool:

- Command help lists the options and arguments used with a specified dcnxt_shell or dc_shell command and displays them in the Design Compiler shell and also in the console log view when the GUI is open.

To get information about the options available for a specific dcnxt_shell or dc_shell command, enter the command name with the `-help` option:

```
prompt> command_name -help
```

- Man pages are displayed in the Design Compiler shell and also in the console log view when the GUI is open.

To get the man page for a specific dcnxt_shell or dc_shell command or variable, enter

```
prompt> man command_or_variable_name
```

- The man page viewer displays command, variable, and error message man pages that you request while using the GUI.

To open the man page viewer, choose Help > Man Pages.

For information about using the GUI to get command help, display man pages, and access the Design Vision Help system, which contains topics that explain tasks that you can perform, see the *Design Vision User Guide*.

Saving Designs and Exiting Design Compiler

You can exit Design Compiler at any time and return to the operating system. By default, dcnxt_shell and dc_shell save the session information in the command.log file. However, if you change the name of the log file using the `sh_command_log_file` variable after you start the tool, session information might be lost.

Also, dcnxt_shell and dc_shell do not automatically save the designs loaded in memory. To save these designs before exiting, use the `write_file` command. For example,

```
prompt> write_file -format ddc -hierarchy -output my_design.ddc
```

To exit dcnxt_shell or dc_shell, do one of the following:

- If you are in the command-line interface, enter `quit` or `exit`.
- If you are running Design Compiler in interactive mode, and the tool is busy, press `Ctrl+D`.

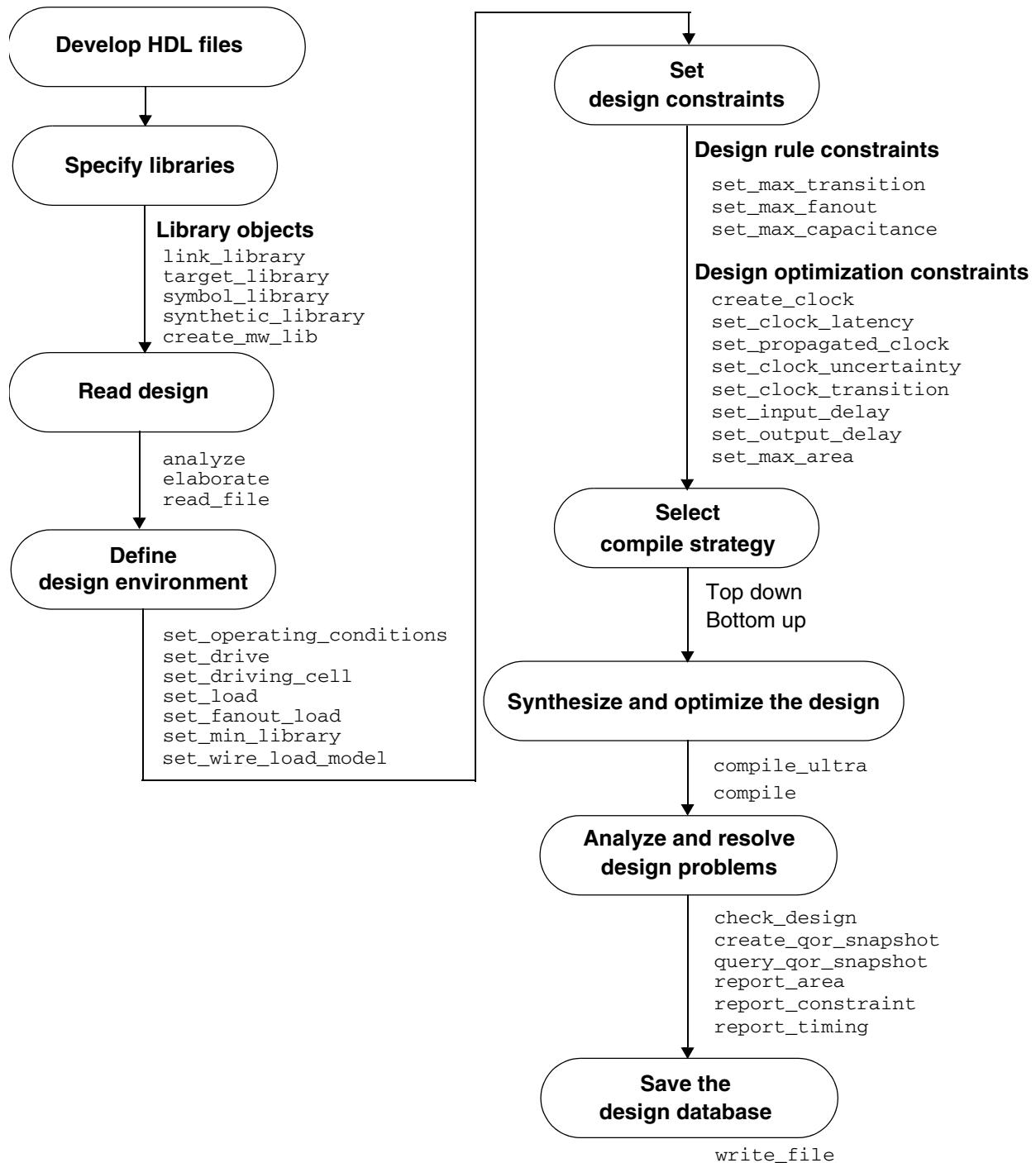
The Synthesis Flow

[Figure 2-1](#) shows a basic synthesis flow. You can use this synthesis flow during design exploration and design implementation.

The figure lists the basic DC Expert and DC Ultra commands that are commonly used in each step of the flow. All the commands shown in the figure can accept options, but no options are shown in the figure.

Note:

In the “Select Compile Strategy” step, top down and bottom up are not commands. They refer to two commonly used compile strategies that use different combinations of commands.

Figure 2-1 Synthesis Flow

The following steps provide an overview of the synthesis flow:

1. Develop the HDL files.

The input design files for Design Compiler are written using a hardware description language (HDL) such as Verilog or VHDL. When preparing the HDL code, you need to consider design data management, design partitioning, and your HDL coding style.

For details, see [Preparing for Synthesis](#) and the HDL Compiler documentation.

2. Specify the libraries.

Specify the link, target, symbol, synthetic, and physical libraries.

For details, see [Setting Up and Working With Libraries](#).

3. Read the design.

Design Compiler can read both RTL designs and gate-level netlists. Design Compiler uses HDL Compiler to read Verilog and VHDL RTL designs and Verilog and VHDL gate-level netlists. You can also read gate-level netlists in .ddc format.

For details, see [Working With Designs in Memory](#).

For information about the recommended reading methods, see the HDL Compiler documentation.

4. Define the design environment.

Design Compiler requires that you model the environment of the design to be synthesized. This model comprises the external operating conditions (manufacturing process, temperature, and voltage), loads, drives, fanouts, and so on. It directly influences design synthesis and optimization results. If you are not using topographical mode, you need to specify wire load models to estimate the effect of wire length on design performance.

For details, see [Defining the Design Environment](#).

5. Set the design constraints.

You define these constraints by using commands such as those listed under this step in [Figure 2-1](#).

Note:

Design constraint settings are influenced by the compile strategy you choose. Flow steps 5 and 6 are interdependent. Compile strategies are discussed in step 6.

For details, see [Defining Design Constraints](#) and [Using Floorplan Physical Constraints](#)

6. Select the compile strategy.

If your design is hierarchical, you must choose a compile strategy. You can use a top-down or bottom-up strategy. Both strategies have advantages and disadvantages, depending on your particular designs and design goals. You can use either strategy to

process the entire design, or you can use a mix of strategies, using the most appropriate strategy for each subdesign.

For details, see [Compile Strategies](#).

7. (Optional) Provide the floorplan information.

If you are using topographical mode, you can provide a floorplan or floorplan constraints. If you do not specify a floorplan, Design Compiler creates one for you. However, specifying floorplan constraints ensures more accurate placement area and improved timing correlation with the post-place-and-route design.

For details, see [Using Floorplan Physical Constraints](#) and [Performing a Bottom-up Hierarchical Compile](#).

8. Synthesize and optimize the design with the `compile_ultra` or `compile` command.

For details, see [Optimizing the Design](#) and [Using Topographical Technology](#).

9. Insert scan chains.

Run the `insert_dft` command to insert scan chains.

For details, see the *DFT Compiler User Guide*.

10. Perform incremental synthesis.

Perform an incremental compile by using the `compile_ultra -incremental -scan` command or the `compile -incremental_mapping -scan` command.

The main goal for `compile_ultra -incremental` is to enable topographical-based optimization for post-topographical-based synthesis flows such as retiming, design-for-test (DFT), DFTMAX, and minor netlist edits.

For details, see [Performing an Incremental Compile](#).

11. (Optional) Visually inspect the floorplan and placement results.

In topographical mode, use the GUI layout window to verify your floorplan and placement results. You can

- Examine the placement and orientation of objects such as macro cells, port locations, and physical constraints
- Examine the placement of critical timing path objects
- Analyze floorplan-related congestion and identify the causes of congestion hotspots

For details, see [Using Design Compiler Graphical](#) and the Design Vision Help.

12. Analyze and resolve design problems.

Design Compiler can generate numerous reports, such as area, constraint, and timing reports, on the synthesis and optimization results. You use reports to analyze and resolve any design problems or to improve synthesis results. You can use the

`check_design` command to check the synthesized design for consistency. Other `check_*` commands are available.

You can also create a categorized timing report in HTML format by using the `create_qor_snapshot` and `query_qor_snapshot` commands. The report lets you quickly find paths with certain problems, such as large fanouts or transition degradation. You can then modify the constraints and generate a new report based on the constraints you specified.

For details, see [Analyzing and Resolving Design Problems](#).

13. Save the design.

Use the `write_file` command to save the synthesized design. Design Compiler does not automatically save designs before exiting. You can write out the design in .ddc, Milkyway, or Verilog format.

For details, see [Working With Designs in Memory](#). If you are using topographical mode, see [Inputs and Outputs in Design Compiler Topographical Mode](#).

You can also save the design attributes and constraints used during synthesis in a script file. Script files are ideal for managing your design attributes and constraints. For details, see the information about using script files in [Using Tcl With Synopsys Tools](#).

See Also

- [Compile Flows in Topographical Mode](#)

A Design Compiler Session Example

[Example 2-3](#) shows a Tcl script that performs a top-down compile run. It uses the basic synthesis flow in topographical mode. The script contains comments that identify each step in the flow. Some of the script command options and arguments have not yet been explained in this manual. Nevertheless, from the previous discussion of the basic synthesis flow, you can begin to understand this example of a top-down compile.

Note:

Only the `set_driving_cell` command is not discussed in the section on basic synthesis design flow. The `set_driving_cell` command is an alternative way to set the external drives on the ports of the design to be synthesized.

Example 2-3 Top-Down Compile Script in Topographical Mode

```
# Specify the libraries
set_app_var search_path "$search_path ./libraries"
set_app_var link_library "* max.lib.db"
set_app_var target_library "max.lib.db"
create_mw_lib -technology $mw_tech_file \
-mw_reference_library $mw_reference_library $mw_lib_name
```

```
open_mw_lib $mw_lib_name

# Read the design
read_verilog rtl.v

# Define the design environment
set_load 2.2 sout
set_load 1.5 cout
set_driving_cell -lib_cell FD1 [all_inputs]

# Set the optimization constraints
create_clock clk -period 10
set_input_delay -max 1.35 -clock clk {ain bin}
set_input_delay -max 3.5 -clock clk cin
set_output_delay -max 2.4 -clock clk cout
extract_physical_constraints def_file_name

# Map and optimize the design
compile_ultra

# Analyze and debug the design
report_timing

change_names -rules verilog -hierarchy

# Save the design database
write_file -format ddc -hierarchy -output top_synthesized.ddc
write_file -format verilog -hierarchy -output netlist.v
write_sdf sdf_file_name
write_parasitics -output parasitics_file_name
write_sdc sdc_file_name
write_floorplan -all phys_cstr_file_name.tcl
```

You can execute these commands in any of the following ways:

- Enter `dcnxt_shell` or `dc_shell` and type each command in the order shown in the example.
- Enter `dcnxt_shell` or `dc_shell` and execute a script file by using the `source` command.

For example, if you are running Design Compiler and the script is in a file called `run.scr`, you can execute the script file by entering the following command:

```
prompt> source run.scr
```

- Run the script from the Linux command line by using the `dcnxt_shell` or `dc_shell` command with the `-f` option.

For example, the following command invokes Design Compiler in topographical mode and executes the `run.scr` script file from the Linux prompt:

```
% dc_shell -topographical_mode -f run.scr
```

Using Multicore Technology

The multicore technology in Design Compiler allows you to use multiple cores to improve the tool runtime. During synthesis, multicore functionality divides large optimization tasks into smaller tasks for processing on multiple cores. All `compile_ultra` command options support the use of multiple cores for optimization.

Multicore technology is supported in DC Ultra in wire load mode and topographical mode, Design Compiler NXT, and Design Compiler Graphical. It is not supported in DC Expert.

Multicore processing requires one Design Compiler license for every eight cores.

To enable multicore processing and report multicore runtime speedup, see

- [Enabling Multicore Functionality](#)
 - [Measuring Runtime](#)
-

Enabling Multicore Functionality

To enable multicore functionality in Design Compiler, use the `set_host_options` command. For example, the following command enables the tool to use six cores to run your processes:

```
prompt> set_host_options -max_cores 6
```

Design Compiler automatically checks for the maximum number of CPU cores available on the execution host. If you specify a higher number of CPU cores than are available, the tool limits the CPU cores to the maximum cores available and issues a warning message.

Design Compiler also checks the load of the multiple cores on multicore machines. If most of the cores are overloaded, the tool limits the number of cores based on the load, and it issues a warning message.

If you are in multicore mode, the log file contains an information message similar to the following message:

```
Information: Running optimization using a maximum of 8 cores. (OPT-1500)
```

Measuring Runtime

When you measure the runtime speedup using multicore optimization, use the wall clock time of the process. The CPU time does not correctly account for multicore runtime speedup.

To report the overall compile wall clock time, run the `report_qor` command, as shown in the following example. The command reports the combined wall clock time, combining the `compile_ultra` and `compile_ultra -incremental` command runs.

```
prompt> report_qor
*****
Report : qor
...
*****
...
Hostname: machine
Compile CPU Statistics
-----
Resource Sharing:           21.54
Logic Optimization:         182.63
Mapping Optimization:       230.79
-----
Overall Compile Time:       631.32
Overall Compile Wall Clock Time: 288.11
```

You can also check the wall clock time using the clock commands shown in the following example:

```
prompt> set_host_options -max_cores 2
prompt> set pre_compile_clock [clock seconds]
prompt> compile_ultra
prompt> set post_compile_clock [clock seconds]
prompt> set diff_clock \
          [expr $post_compile_clock - $pre_compile_clock]
```

See Also

- [Displaying Quality of Results](#)

Runtime Improvement Techniques Outside of Synthesis

Executing checking or reporting commands serially in a script can consume a significant portion of the overall runtime. To improve runtime, you can use multiple cores to run checking and reporting commands in the following ways:

- Run a command or a list of commands in the background while other commands run in the foreground
- Run commands in parallel in the foreground or in the background

For details, see the following topics:

- [Running Commands in the Background](#)
- [Enabling Parallel Command Execution](#)

Note:

Multicore technology is supported in DC Ultra in both wire load mode and topographical mode, and it is supported in Design Compiler NXT and Design Compiler Graphical. It is not supported in DC Expert.

Running Commands in the Background

To improve runtime in the Design Compiler NXT and Design Compiler Graphical tools, you can run commands in the background while other commands run in the foreground by using the `redirect` command with the `-bg` option. You can run a single command, source a Tcl script to run several commands in parallel, or run a Tcl procedure.

You must specify the number of cores to use to run the background jobs. The number of cores available to `redirect -bg` is deducted from the cores set by the `set_host_options` command. When you use the `redirect` command to run a background job, `dc_shell` returns immediately to execute the next command.

The following example redirects a Tcl script to run in the background:

```
set_host_options -max_cores 8
read_verilog
read_sdc
...
compile_ultra -spg
redirect -bg -max_cores 3 -file y.out {source y.tcl}
remote_execute_source1.log {source bg_script.tcl}
    Information: redirect -bg with max_cores 3 started. The maximum number
    of cores available in parent is reduced to 5. (BGE-004)
compile_ultra -spg -incremental
```

There can be only two background jobs active at a time. If you redirect a third command or script, the command waits until a previous redirect command is completed before starting the new job.

Supported commands include reporting, analysis, and design export commands only. Commands that change the netlist or other data are not supported.

You can also run the `parallel_execute` command inside the `redirect -bg` command to run a list of checking and reporting commands in parallel. For details, see [Enabling Parallel Command Execution](#).

Verification Guidance Support

If you want to include the `change_names` guidance from the background job in your netlist for formal verification, you must stop recording Formality setup information before running the `redirect -bg` command and use the `set_svf` command to include an .svf file in the script executed by the `redirect -bg` command.

In the following example, the `y.tcl` script includes the `set_svf be.svf` command to generate verification guidance from the `redirect -bg` command that you can use in your netlist for formal verification in the Formality tool.

```
set_host_options -max_cores 8
set_svf compile.svf
read_verilog
read_sdc
...
compile_ultra -spg
set_svf -off
redirect -bg -max_cores 3 -file y.out {source y.tcl} \
remote_execute_source1.log {source bg_script.tcl}
Information: redirect -bg with max_cores 3 started. The maximum number
of cores available in parent is reduced to 5. (BGE-004)
set_svf post_compile.svf
compile_ultra -spg -incremental
```

Be sure to use the correct .svf file when you run the Formality tool. For information about using the Formality tool for formal verification, see [Verifying Functional Equivalence](#).

Reporting Jobs Submitted To Run in the Background

To report all the jobs that you have submitted to run in the background by the `redirect -bg` command, use the `report_background_jobs` command.

The `report_background_jobs` command reports the completed jobs and the jobs currently running in the background, as shown in the following example:

```
prompt> report_background_jobs

JOB 'redirect -bg -file {background.log} source bg_script.tcl
-max_cores 4' completed
JOB(pid:13010) 'redirect -bg -file {background_1.log} source
bg_script_1.tcl -max_cores 3' is running
```

To omit the completed jobs, use the `-reset` option with the `report_background_jobs` command.

Enabling Parallel Command Execution

You can run checking and reporting commands in parallel in the following ways:

- Specify the list the commands you want to run in parallel by running the `parallel_execute` command.
The tool blocks `dc_shell` until the longest running command in the parallel execution list is completed. For details, see [Running Commands in Parallel in the Foreground](#).
- Specify the list of commands you want to run in parallel in the background. In this case, you must specify the `parallel_execute` command and the list of commands to run in parallel using the `redirect -bg` command.
When you use the `redirect -bg` command to run a background job, `dc_shell` returns immediately to execute the next command. For details, see [Running Commands in the Background](#).

Running Commands in Parallel in the Foreground

You must specify the number of cores by using the `-max_cores` option with the `set_host_options` command before running the `parallel_execute` command. Parallel command execution can use up to eight cores. If you specify a number larger than eight, the tool issues a warning message and limits the maximum cores to the number of available cores.

The following example specifies ten cores, but the tool limits parallel command execution to eight cores:

```
prompt> set_host_options -max_cores 10
prompt> parallel_execute [list \
"report_cell" "report_timing" "report_area"]
Warning: you specified 10 cores to use but 'parallel_execute' can use
only up to 8 cores. parallel_execute will override and use only 8 cores.
(RPT-110)
```

If any of the listed reporting commands need updated timing information and the `update_timing` command is not specified before the `parallel_execute` command, the tool automatically invokes the `update_timing` command before parallel command execution. For example, the following script updates the timing information and then executes the `report_timing`, `report_qor`, `report_cell`, and `report_area` commands in parallel using eight cores:

```
set_host_options -max_cores 8
update_timing
parallel_execute [list \
    "report_timing > $mylogfile" \
    "report_qor >> $mylogfile" \
    "report_cell" \
    "report_area"]
```

The following script uses variables, including `$MAX`, `$NWORST`, `$cstr_log`, and `$qor_log` for the option arguments and log files in the command strings:

```
set MAX 10000; set_app_var NWORST 10
set rpt_const_options "-all_violators"
set cstr_log "rpt_cstr.log"
set qor_log "rpt_qor.log"
parallel_execute [list \
    "report_timing -max $MAX -nworst $NWORST > rpt_tim.log" \
    "report_constraints $rpt_const_options > $cstr_log" \
    "report_qor > $qor_log"]
```

Parallel command execution with multicore processing has the same license requirements as serial command execution. By default, you need one Design Compiler license for every eight cores.

Supported Commands for Parallel Execution

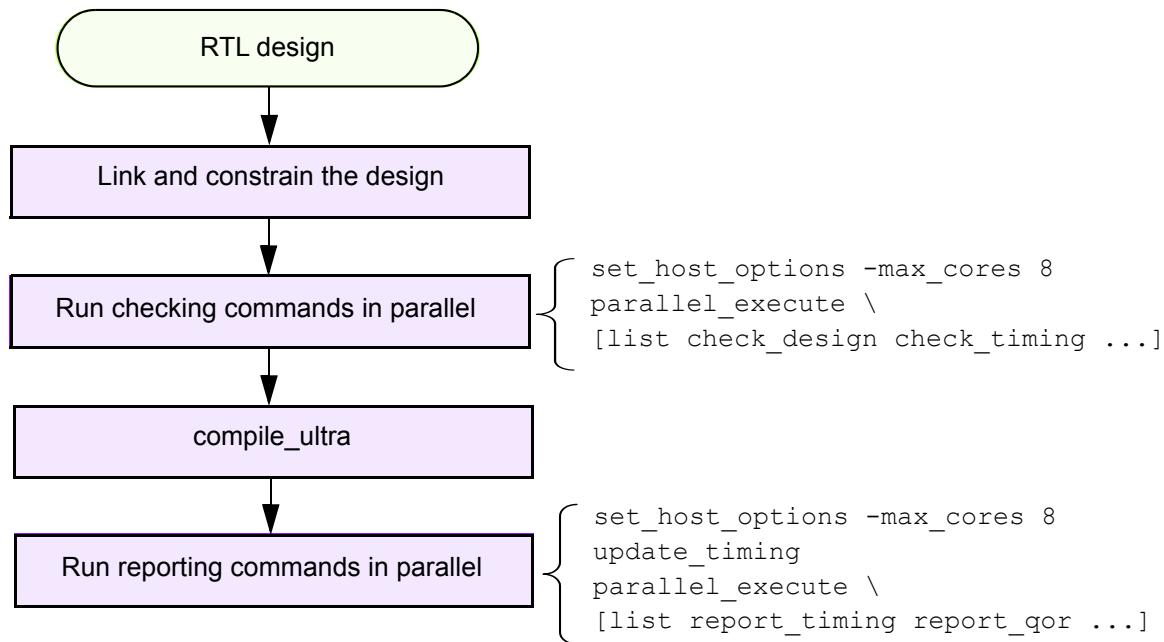
Use parallel command execution for reporting and checking commands only. To list the supported commands, specify the `-list_all` option with the `parallel_execute` command. If you specify an unsupported command, the tool skips the command and issues a warning message similar to the following:

```
prompt> parallel_execute [list report_libcell_subset]
Warning: 'report_libcell_subset' report command can't run in
parallel_execute mode. (RPT-106)
```

Parallel Command Execution Design Flow

Figure 2-2 shows how to run checking and reporting commands in parallel to improve runtime in the Design Compiler design flow.

Figure 2-2 Parallel Command Execution in the Design Compiler Design Flow



See Also

- [Checking for Design Consistency](#)
- [Analyzing and Resolving Design Problems](#)

3

Preparing for Synthesis

Designs (design descriptions) are stored in design files. Design files must have unique names. If a design is hierarchical, each subdesign refers to another design file, which must also have a unique name. However, different design files can contain subdesigns with identical names.

As you prepare your design for synthesis, it is important that you develop a strategy for managing design files so that data is not lost. It is important also to consider HDL coding strategies. HDL coding is the foundation for synthesis because it implies the initial structure of the design. In addition, consider how constraints will impact your design. Constraints are declarations that define the design's goals in measurable circuit characteristics. They can be specified interactively on the command line or specified in a script file.

To learn about preparing your design and constraints for synthesis, see the following topics:

- [Managing the Design Data](#)
- [Partitioning for Synthesis](#)
- [HDL Coding for Synthesis](#)
- [Performing Design Exploration](#)
- [Creating Constraints](#)

Managing the Design Data

Use systematic organizational methods to manage the design data. Design data control and data organization, as described in the following topics, are two basic elements of managing design data.

- [Controlling the Design Data](#)
- [Organizing the Design Data](#)

See Also

- [Creating Constraints](#)

Controlling the Design Data

As new versions of your design are created, you must maintain some archival and record-keeping method that provides a history of the design evolution and that lets you restart the design process if data is lost. Establishing controls for data creation, maintenance, overwriting, and deletion is a fundamental design management issue. Establishing file-naming conventions is one of the most important rules for data creation.

[Table 3-1](#) lists the recommended file name extensions for each design data type.

Table 3-1 File Name Extensions

Design data type	Extension	Description
Design source code	.v	Verilog
	.vhd	VHDL
Synthesis scripts	.con	Constraints
	.scr	Script
Reports and logs	.rpt	Report
	.log	Log
Design database	.ddc	Synopsys internal database format

Organizing the Design Data

Establishing and adhering to a method of organizing data is more important than the method you choose. After you place the essential design data under a consistent set of controls, you can organize the data in a meaningful way. To simplify data exchanges and data searches, you should adhere to this data organization system.

You can use a hierarchical directory structure to address data organization issues. Your compile strategy will influence your directory structure. [Figure 3-1](#) shows directory structures based on the top-down compile strategy, and [Figure 3-2](#) shows directory structures based on the bottom-up compile strategy.

Figure 3-1 Top-Down Compile Directory Structure

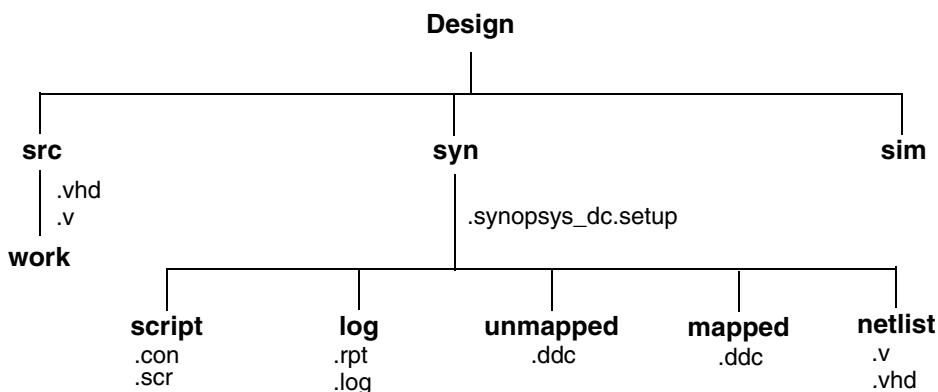
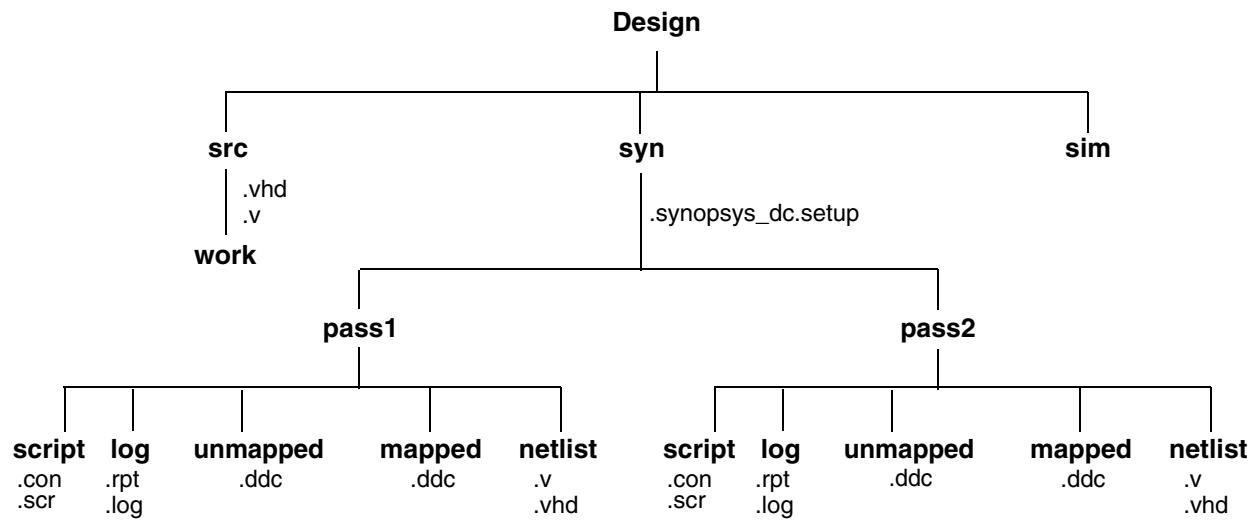


Figure 3-2 Bottom-Up Compile Directory Structure

See Also

- [Compile Strategies](#)

Provides information about top-down, bottom-up, and mixed compile strategies

Partitioning for Synthesis

Partitioning a design effectively can enhance the synthesis results, reduce compile time, and simplify the constraint and script files.

Partitioning affects block size, and although Design Compiler has no inherent block size limit, you should be careful to control block size. If you make blocks too small, you can create artificial boundaries that restrict effective optimization. If you create very large blocks, compile runtimes can be lengthy.

Use the following strategies to partition your design and improve optimization and runtimes:

- [Partitioning for Design Reuse](#)
- [Keeping Related Combinational Logic Together](#)
- [Registering Block Outputs](#)
- [Partitioning by Design Goal](#)
- [Partitioning by Compile Technique](#)
- [Keeping Sharable Resources Together](#)

- [Keeping User-Defined Resources With the Logic They Drive](#)
- [Isolating Special Functions](#)

Partitioning for Design Reuse

Design reuse decreases time-to-market by reducing the design, integration, and testing effort. When reusing existing designs, partition the design to enable instantiation of the designs.

To enable designs to be reused, follow these guidelines during partitioning and block design:

- Thoroughly define and document the design interface.
- Standardize interfaces whenever possible.
- Parameterize the HDL code.

Keeping Related Combinational Logic Together

Dividing related combinational logic into separate blocks introduces artificial barriers that restrict logic optimization.

For best results, apply these strategies:

- Group related combinational logic and its destination register together.

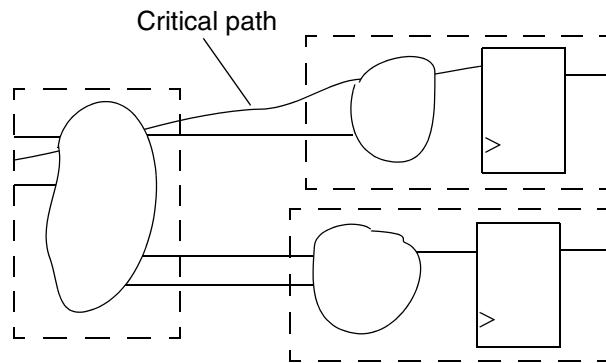
When working with the complete combinational path, Design Compiler has the flexibility to merge logic, resulting in a smaller, faster design. Grouping combinational logic with its destination register also simplifies the timing constraints and enables sequential optimization.

- Eliminate glue logic.

Glue logic is the combinational logic that connects blocks. Moving this logic into one of the blocks improves synthesis results by providing Design Compiler with additional flexibility. Eliminating glue logic also reduces compile time, because Design Compiler has fewer logic levels to optimize.

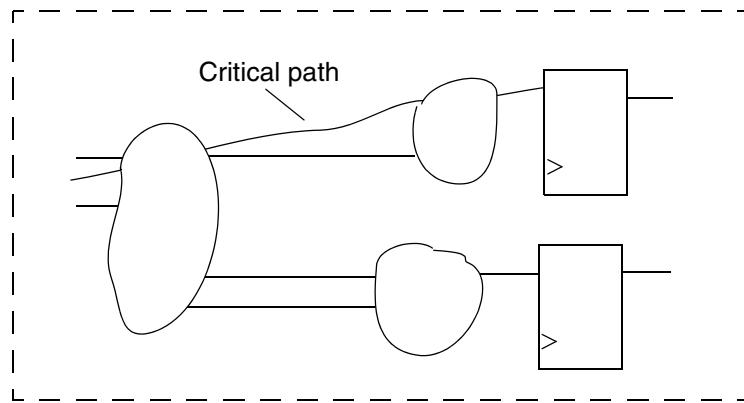
For example, assume that you have a design containing three combinational clouds on or near the critical path. [Figure 3-3](#) shows poor partitioning of this design. Each of the combinational clouds occurs in a separate block, so Design Compiler cannot fully exploit its combinational optimization techniques.

Figure 3-3 Poor Partitioning of Related Logic



[Figure 3-4](#) shows the same design with no artificial boundaries. In this design, Design Compiler has the flexibility to combine related functions in the combinational clouds.

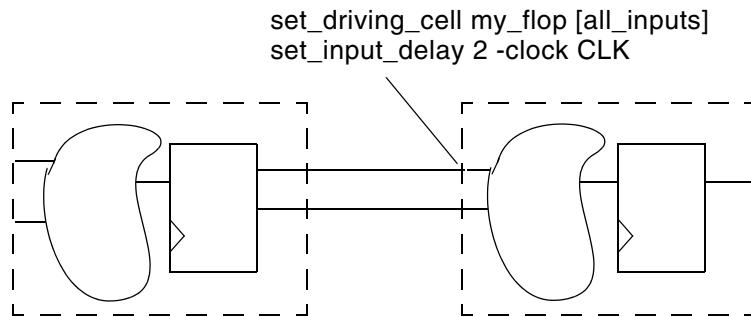
Figure 3-4 Keeping Related Logic in the Same Block



Registering Block Outputs

To simplify the constraint definitions, make sure that registers drive the block outputs, as shown in [Figure 3-5](#).

Figure 3-5 Registering All Outputs



This method enables you to constrain each block easily because

- The drive strength on the inputs to an individual block always equals the drive strength of the average input drive
- The input delays from the previous block always equal the path delay through the flip-flop

Because no combinational-only paths exist when all outputs are registered, time budgeting the design and using the `set_output_delay` command are easier. Given that one clock cycle occurs within each module, the constraints are simple and identical for each module.

This partitioning method can improve simulation performance. With all outputs registered, a module can be described with only edge-triggered processes. The sensitivity list contains only the clock and, perhaps, a reset pin. A limited sensitivity list speeds simulation by having the process triggered only one time in each clock cycle.

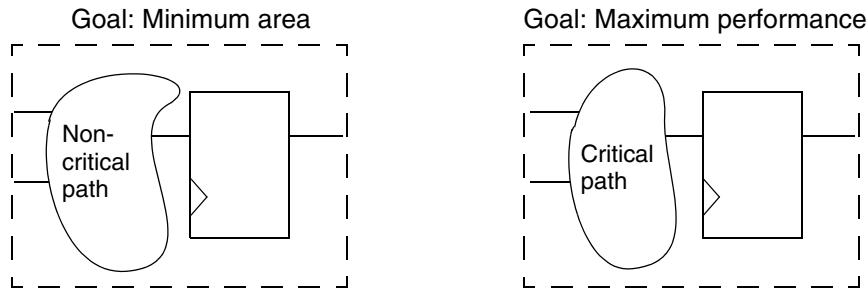
Partitioning by Design Goal

Partition logic with different design goals into separate blocks. Use this method when certain parts of a design are more area and timing critical than other parts.

To achieve the best synthesis results, isolate the noncritical speed constraint logic from the critical speed constraint logic. By isolating the noncritical logic, you can apply different constraints, such as a maximum area constraint, on the block.

[Figure 3-6](#) shows how to separate logic with different design goals.

Figure 3-6 Blocks With Different Constraints



Partitioning by Compile Technique

Partition logic that requires different compile techniques into separate blocks. Use this method when the design contains highly structured logic along with random logic.

- Highly structured logic, such as error detection circuitry, which usually contains large exclusive OR trees, is better suited to structuring.
- Random logic is better suited to flattening.

See Also

- [Logic-Level Optimization](#)

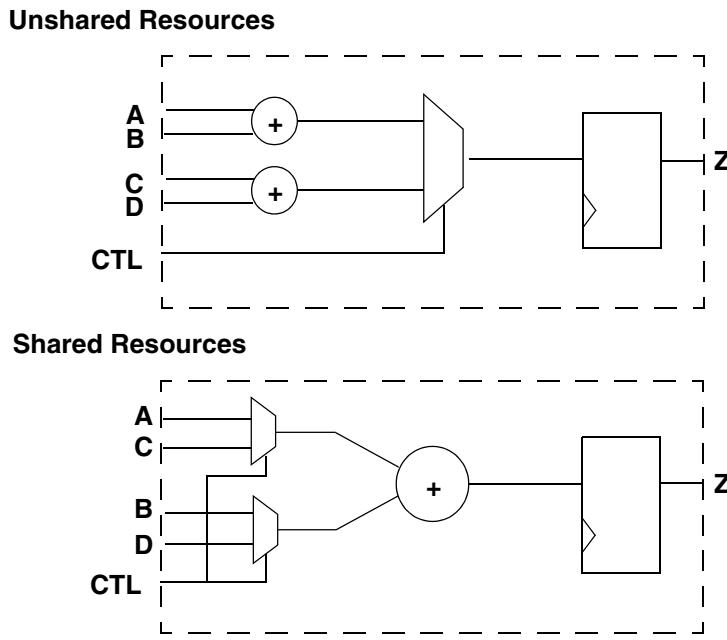
Provides information about structuring and flattening

Keeping Sharable Resources Together

Design Compiler can share large resources, such as adders or multipliers, but resource sharing can occur only if the resources belong to the same VHDL process or Verilog always block.

For example, if two separate adders have the same destination path and have multiplexed outputs to that path, keep the adders in one VHDL process or Verilog always block. This approach allows Design Compiler to share resources (using one adder instead of two) if the constraints allow sharing. [Figure 3-7](#) shows possible implementations of a logic example.

Figure 3-7 Keeping Sharable Resources in the Same Process



See Also

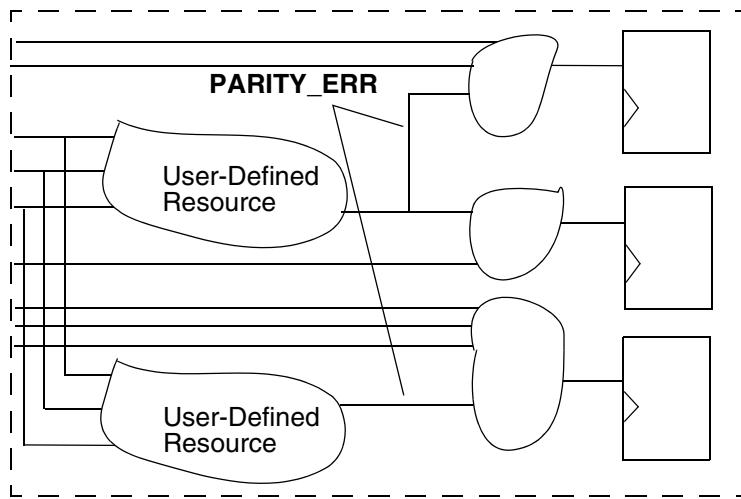
- [Resource Sharing](#)
- The HDL Compiler documentation

Keeping User-Defined Resources With the Logic They Drive

User-defined resources are user-defined functions, procedures, or macro cells, or user-created DesignWare components. Design Compiler cannot automatically share or create multiple instances of user-defined resources. Keeping these resources with the logic they drive, however, gives you the flexibility to split the load by manually inserting multiple instantiations of a user-defined resource if timing goals cannot be achieved with a single instantiation.

[Figure 3-8](#) illustrates splitting the load by multiple instantiation when the load on the signal PARITY_ERR is too heavy to meet constraints.

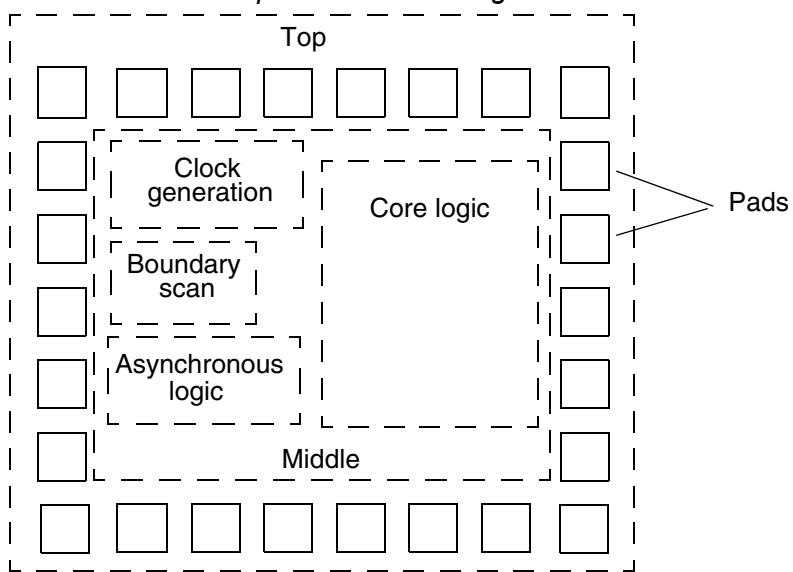
Figure 3-8 Duplicating User-Defined Resources



Isolating Special Functions

Isolate special functions, such as I/O pads, clock generation circuitry, boundary-scan logic, and asynchronous logic from the core logic. [Figure 3-9](#) shows the recommended partitioning for the top level of the design.

Figure 3-9 Recommended Top-Level Partitioning



The top level of the design contains the I/O pad ring and a middle level of hierarchy that contains submodules for the boundary-scan logic, the clock generation circuitry, the asynchronous logic, and the core logic. The middle level of hierarchy exists to allow the flexibility to instantiate I/O pads. Isolation of the clock generation circuitry enables instantiation and careful simulation of this module. Isolation of the asynchronous logic helps confine testability problems and static timing analysis problems to a small area.

HDL Coding for Synthesis

HDL coding is the foundation for synthesis because it implies the initial structure of the design. When writing your HDL source code, always consider the hardware implications of the code. A good coding style can generate smaller and faster designs.

The quality of optimization results depends on how the HDL description is written. In particular, the partitioning of the hierarchy in the HDL, if done well, can enhance optimization.

To learn how to write efficient code so that you can achieve your design target in the shortest possible time, see

- [Writing Technology-Independent HDL](#)
- [Using HDL Constructs](#)
- [Writing Effective Code](#)

See Also

- [Instantiating RTL PG Pins](#)

Writing Technology-Independent HDL

The goal of high-level design that uses a completely automatic synthesis process is to have no instantiated gates or flip-flops. If you meet this goal, you will have readable, concise, and portable high-level HDL code that can be transferred to other vendors or to future processes.

In some cases, the HDL Compiler tool requires compiler directives to provide implementation information while still maintaining technology independence. In Verilog, compiler directives begin with the characters // or /*. In VHDL, compiler directives begin with two hyphens (--) followed by `pragma` or `synopsys`.

To learn various methods for keeping your HDL code technology independent, see

- [Inferring Components](#)
- [The HDL Compiler documentation](#)

Inferring Components

HDL Compiler can infer multiplexers, registers, three-state drivers, and multibit components.

To learn about these inference capabilities, see

- [Inferring Multiplexers](#)
- [Inferring Registers](#)
- [Mixing Register Types](#)
- [Inferring Registers Without Control Signals](#)
- [Inferring Registers With Control Signals](#)
- [Inferring Three-State Drivers](#)
- [Inferring Multibit Components](#)

Inferring Multiplexers

HDL Compiler can infer a generic multiplexer cell (MUX_OP) from case statements in your HDL code. If your target logic library contains at least a 2-to-1 multiplexer cell, Design Compiler maps the inferred MUX_OPs to multiplexer cells in the target logic library. Design Compiler determines the MUX_OP implementation during compile based on the design constraints. For information about how Design Compiler maps MUX_OPs to multiplexers, see [Multiplexer Mapping and Optimization](#).

Use the `infer_mux` compiler directive to control multiplexer inference. When attached to a block, the `infer_mux` directive forces multiplexer inference for all case statements in the block. When attached to a case statement, the `infer_mux` directive forces multiplexer inference for that specific case statement.

Inferring Registers

Register inference allows you to specify technology-independent sequential logic in your designs. A register is a simple, 1-bit memory device, either a latch or a flip-flop. A latch is a level-sensitive memory device. A flip-flop is an edge-triggered memory device.

HDL Compiler infers a D latch whenever you do not specify the resulting value for an output under all conditions, as in an incompletely specified if or case statement. HDL Compiler can also infer SR latches and master-slave latches. For examples of designs that use various types of latches, see [Latch-Based Design Code Examples](#).

HDL Compiler infers a D flip-flop whenever the sensitivity list of a Verilog always block or VHDL process includes an edge expression (a test for the rising or falling edge of a signal). HDL Compiler can also infer JK flip-flops and toggle flip-flops.

Mixing Register Types

For best results, restrict each Verilog always block or VHDL process to a single type of register inferencing: latch, latch with asynchronous set or reset, flip-flop, flip-flop with asynchronous set or reset, or flip-flop with synchronous set or reset.

Be careful when mixing rising- and falling-edge-triggered flip-flops in your design. If a module infers both rising- and falling-edge-triggered flip-flops and the target logic library does not contain a falling-edge-triggered flip-flop, Design Compiler generates an inverter in the clock tree for the falling-edge clock.

Inferring Registers Without Control Signals

For inferring registers without control signals, make the data and clock pins controllable from the input ports or through combinational logic. If a gate-level simulator cannot control the data or clock pins from the input ports or through combinational logic, the simulator cannot initialize the circuit, and the simulation fails.

Inferring Registers With Control Signals

You can initialize or control the state of a flip-flop by using either an asynchronous or a synchronous control signal.

For inferring asynchronous control signals on latches, use the `async_set_reset` compiler directive (attribute in VHDL) to identify the asynchronous control signals. HDL Compiler automatically identifies asynchronous control signals when inferring flip-flops.

For inferring synchronous resets, use the `sync_set_reset` compiler directive (attribute in VHDL) to identify the synchronous controls.

Inferring Three-State Drivers

Assign the high-impedance value (1'bz in Verilog, 'Z' in VHDL) to the output pin to have Design Compiler infer three-state gates. Three-state logic reduces the testability of the design and makes debugging difficult. Where possible, replace three-state buffers with a multiplexer.

Never use high-impedance values in a conditional expression. HDL Compiler always evaluates expressions compared to high-impedance values as false, which can cause the gate-level implementation to behave differently from the RTL description.

For additional information about three-state inference, see the HDL Compiler documentation.

Inferring Multibit Components

Multibit inference allows you to map registers to regularly structured logic or multibit library cells. Using multibit components can have the following results:

- Smaller area and delay, due to shared transistors and optimized transistor-level layout
- Reduced clock skew in sequential gates
- Lower power consumption by the clock in sequential banked components
- Improved regular layout of the data path

Multibit components might not be efficient in the following instances:

- As state machine registers
- In small bused logic that would benefit from single-bit design

You must weigh the benefits of multibit components against the loss of optimization flexibility when deciding whether to map to multibit or single-bit components.

Attach the `infer_multibit` compiler directive to bused signals to infer multibit components. You can also change between a single-bit and a multibit implementation after optimization by using the `create_multibit` and `remove_multibit` commands.

For more information about how Design Compiler handles multibit components, see [Optimizing Multibit Registers](#).

Using HDL Constructs

For information and guidelines about HDL constructs, see

- [General HDL Constructs](#)
- [Using Verilog Macro Definitions](#)
- [Using VHDL Port Definitions](#)

General HDL Constructs

The following information applies to both Verilog and VHDL:

- [Sensitivity Lists](#)
- [Value Assignments](#)
- [if Statements](#)

- [case Statements](#)
- [Constant Definitions](#)

Sensitivity Lists

You should completely specify the sensitivity list for each Verilog always block or VHDL process. Incomplete sensitivity lists (shown in the following examples) can result in simulation mismatches between the HDL and the gate-level design.

Example 3-1 Incomplete Sensitivity List (Verilog)

```
always @ (A)
    C <= A | B;
```

Example 3-2 Incomplete Sensitivity List (VHDL)

```
process (A)
    C <= A or B;
```

Value Assignments

Both Verilog and VHDL support the use of immediate and delayed value assignments in the RTL code. The hardware generated by immediate value assignments—implemented by Verilog blocking assignments (=) and VHDL variables (:=)—is dependent on the ordering of the assignments. The hardware generated by delayed value assignments—implemented by Verilog nonblocking assignments (<=) and VHDL signals (<=)—is independent of the ordering of the assignments.

For correct simulation results,

- Use delayed (nonblocking) assignments within sequential Verilog always blocks or VHDL processes
- Use immediate (blocking) assignments within combinational Verilog always blocks or VHDL processes

if Statements

When an if statement used in a Verilog always block or VHDL process as part of a continuous assignment does not include an else clause, Design Compiler creates a latch. The following examples show if statements that generate latches during synthesis.

Example 3-3 Incorrect if Statement (Verilog)

```
if ((a == 1) && (b == 1))
    z = 1;
```

Example 3-4 Incorrect if Statement (VHDL)

```
if (a = '1' and b = '1') then
  z <= '1';
end if;
```

case Statements

If your if statement contains more than three conditions, consider using the case statement to improve the parallelism of your design and the clarity of your code. The following examples use the case statement to implement a 3-bit decoder.

Example 3-5 Using the case Statement (Verilog)

```
case ({a, b, c})
  3'b000: z = 8'b00000001;
  3'b001: z = 8'b00000010;
  3'b010: z = 8'b00000100;
  3'b011: z = 8'b00001000;
  3'b100: z = 8'b00010000;
  3'b101: z = 8'b00100000;
  3'b110: z = 8'b01000000;
  3'b111: z = 8'b10000000;
  default: z = 8'b00000000;
endcase
```

Example 3-6 Using the case Statement (VHDL)

```
case_value := a & b & c;
CASE case_value IS
  WHEN "000" =>
    z <= "00000001";
  WHEN "001" =>
    z <= "00000010";
  WHEN "010" =>
    z <= "00000100";
  WHEN "011" =>
    z <= "00001000";
  WHEN "100" =>
    z <= "00010000";
  WHEN "101" =>
    z <= "00100000";
  WHEN "110" =>
    z <= "01000000";
  WHEN "111" =>
    z <= "10000000";
  WHEN OTHERS =>
    z <= "00000000";
END CASE;
```

An incomplete case statement results in the creation of a latch. VHDL does not support incomplete case statements. In Verilog you can avoid latch inference by using either the default clause or the full_case compiler directive.

Although both the full_case directive and the default clause prevent latch inference, they have different meanings. The full_case directive asserts that all valid input values have been specified and no default clause is necessary. The default clause specifies the output for any undefined input values.

For best results, use the default clause instead of the full_case directive. If the unspecified input values are don't care conditions, using the default clause with an output value of x can generate a smaller implementation.

If you use the full_case directive, the gate-level simulation might not match the RTL simulation whenever the case expression evaluates to an unspecified input value. If you use the default clause, simulation mismatches can occur only if you specified don't care conditions and the case expression evaluates to an unspecified input value.

Constant Definitions

Use the Verilog `define statement or the VHDL constant statement to define global constants. Keep global constant definitions in a separate file. Use parameters (Verilog) or generics (VHDL) to define local constants.

[Example 3-7](#) shows a Verilog code fragment that includes a global `define statement and a local parameter. [Example 3-8](#) shows a VHDL code fragment that includes a global constant and a local generic.

Example 3-7 Using Macros and Parameters (Verilog)

```
// Define global constant in def_macro.v
`define WIDTH 128

// Use global constant in reg128.v
reg regfile[WIDTH-1:0];

// Define and use local constant in module my_module
module my_module (a, b, c);
    parameter WIDTH=128;
    input [WIDTH-1:0] a, b;
    output [WIDTH-1:0] c;
```

Example 3-8 Using Global Constants and Generics (VHDL)

```
-- Define global constant in synthesis_def.vhd
constant WIDTH : INTEGER := 128;

-- Include global constants
library my_lib;
USE my_lib.synthesis_def.all;

-- Use global constant in entity my_design
entity my_design is
    port (a,b : in std_logic_vector(WIDTH-1 downto 0);
          c: out std_logic_vector(WIDTH-1 downto 0));
end my_design;

-- Define and use local constant in entity my_design
entity my_design is
    generic (WIDTH_VAR : INTEGER := 128);
    port (a,b : in std_logic_vector(WIDTH-1 downto 0);
          c: out std_logic_vector(WIDTH-1 downto 0));
end my_design;
```

Using Verilog Macro Definitions

In Verilog, macros are implemented using the `define statement. Follow these guidelines for `define statements:

- Use `define statements only to declare constants.
- Keep `define statements in a separate file.
- Do not use nested `define statements.

Reading a macro that is nested more than twice is difficult. To make your code readable, do not use nested `define statements.

- Do not use `define inside module definitions.

When you use a `define statement inside a module definition, the local macro and the global macro have the same reference name but different values. Use parameters to define local constants.

Using VHDL Port Definitions

When defining ports in VHDL source code, observe these guidelines:

- Use the STD_LOGIC and STD_LOGIC_VECTOR packages.

By using STD_LOGIC, you avoid the need for type conversion functions on the synthesized design.

- Do not use the buffer port mode.

When you declare a port as a buffer, the port must be used as a buffer throughout the hierarchy. To simplify synthesis, declare the port as an output, then define an internal signal that drives the output port.

Writing Effective Code

For guidelines for writing efficient, readable HDL source code for synthesis, see

- [Guidelines for Identifiers](#)
- [Guidelines for Expressions](#)
- [Guidelines for Functions](#)
- [Guidelines for Modules](#)

Guidelines for Identifiers

A good identifier name conveys the meaning of the signal, the value of a variable, or the function of a module; without this information, the hardware descriptions are difficult to read.

Observe the following naming guidelines to improve the clarity of your HDL source code:

- Ensure that the signal name conveys the meaning of the signal or the value of a variable without being verbose.

For example, assume that you have a variable that represents the floating point opcode for rs1. A short name, such as frs1, does not convey the meaning to the reader. A long name, such as floating_pt_opcode_rs1, conveys the meaning, but its length might make the source code difficult to read. Use a name such as fpop_rs1, which meets both goals.

- Use a consistent naming style for capitalization and to distinguish separate words in the name.

Commonly used styles include C, Pascal, and Modula.

- C style uses lowercase names and separates words with an underscore, for example, packet_addr, data_in, and first_grant_enable.
- Pascal style capitalizes the first letter of the name and first letter of each word, for example, PacketAddr, DataIn, and FirstGrantEnable.
- Modula style uses a lowercase letter for the first letter of the name and capitalizes the first letter of subsequent words, for example, packetAddr, dataIn, and firstGrantEnable.

Choose one convention and apply it consistently.

- Avoid confusing characters.
Some characters (letters and numbers) look similar and are easily confused, for example, O and 0 (zero); I and 1 (one).
- Avoid reserved words.
- Use the noun or noun followed by verb form for names, for example, AddrDecode, DataGrant, PCI_interrupt.
- Add a suffix to clarify the meaning of the name.

[Table 3-2](#) shows common suffixes and their meanings.

Table 3-2 Signal Name Suffixes and Their Meanings

Suffix	Meaning
_clk	Clock signal
_next	Signal before being registered
_n	Active low signal
_z	Signal that connects to a three-state output
_f	Register that uses an active falling edge
_xi	Primary chip input
_xo	Primary chip output
_xod	Primary chip open drain output
_xz	Primary chip three-state output
_xbio	Primary chip bidirectional I/O

Guidelines for Expressions

Observe the following guidelines for expressions:

- Use parentheses to indicate precedence.

Expression operator precedence rules are confusing, so you should use parentheses to make your expression easy to read. Unless you are using DesignWare resources, parentheses have little effect on the generated logic. An example of a logic expression without parentheses that is difficult to read is

```
bus_select = a ^ b & c^^d|b^~e&^f[1:0];
```

- Replace repetitive expressions with function calls or continuous assignments.
If you use a particular expression more than two or three times, consider replacing the expression with a function or a continuous assignment that implements the expression.

Guidelines for Functions

Observe these guidelines for functions:

- Do not use global references within a function.

In procedural code, a function is evaluated when it is called. In a continuous assignment, a function is evaluated when any of its declared inputs changes.

Avoid using references to global names within a function because the function might not be reevaluated if the global value changes. This can cause a simulation mismatch between the HDL description and the gate-level netlist.

For example, the following Verilog function references the global name byte_sel:

```
function byte_compare;
    input [15:0] vector1, vector2;
    input [7:0] length;

    begin
        if (byte_sel)
            // compare the upper byte
        else
            // compare the lower byte
        ...
    end
endfunction // byte_compare
```

- Be aware that the local storage for tasks and functions is static.

Formal parameters, outputs, and local variables retain their values after a function has returned. The local storage is reused each time the function is called. This storage can be useful for debugging, but storage reuse also means that functions and tasks cannot be called recursively.

- Be careful when using component implication.

You can map a function to a specific implementation by using the map_to_module and return_port_name compiler directives. Simulation uses the contents of the function. Synthesis uses the gate-level module in place of the function. When you are using component implication, the RTL model and the gate-level model might be different. Therefore, the design cannot be fully verified until simulation is run on the gate-level design.

The following functionality might require component instantiation or functional implication:

- Clock-gating circuitry for power savings
- Asynchronous logic with potential hazards

This functionality includes asynchronous logic and asynchronous signals that are valid during certain states.

- Data-path circuitry

This functionality includes large multiplexers; instantiated wide banks of multiplexers; memory elements, such as RAM or ROM; and black box macro cells.

For more information about component implication, see the HDL Compiler documentation.

Guidelines for Modules

Observe these guidelines for modules:

- Avoid using logic expressions when you pass a value through ports.
The port list can include expressions, but expressions complicate debugging. In addition, isolating a problem related to the bit field is difficult, particularly if that bit field leads to internal port quantities that differ from external port quantities.
- Define local references as generics (VHDL) or parameters (Verilog). Do not pass generics or parameters into modules.

Instantiating RTL PG Pins

Design Compiler can accept RTL designs containing a small number of PG pin connections on macros. The tool does not support a full PG netlist for a block. For example, the tool only supports designs that contain a small number of analog macros that have PG pins.

To instantiate PG pins in your RTL design, set the `dc_allow_rtl_pg` variable to `true`. The default is `false`. To preserve the PG connections in a Verilog output, execute the `write_file -pg -format verilog` command. To preserve the PG connections in a `.ddc` format output, execute the `write_file -format ddc` command. Note that when saving the design in `.ddc` format, you do not need to use the `-pg` option. When reading the `.ddc` file back into Design Compiler, make sure that the `dc_allow_rtl_pg` variable is set to `true`; otherwise, the tool issues a DDC-21 error:

Error: The feature used to generate this DDC file is not supported by this tool or is not enabled in the current session. (DDC-21)

Information: This `.ddc` file contains RTL PG data. Set the `dc_allow_rtl_pg` variable to `true` before reading the file back into Design Compiler.

To preserve the PG connections in a Milkyway database, use the `write_milkyway` command. To pass the design netlist to IC Compiler, PrimeTime, or Formality, you can use a Verilog output, .ddc file, or Milkyway database.

To use PG pins in your RTL design, observe the following guidelines:

- PG libraries are required
- FRAM must always have correct PG information
- The tool will not display the PG nets and pins; the `get_pins` command will not show PG pins
- The RTL design must represent all PG pins as wires, not as supply0, supply1, and so on
- The RTL design must instance PG pins by name, such as `ref U1 (.pin(net), ...)`;
- PG nets can only connect to the following:
 - Macro cells
 - PG pins on leaf power management cells (power switches, level shifters, and isolation cells)
 - Hierarchical ports
- PG nets should reach the top level, but do not have to connect to top-level ports
- The tool will mark cells with PG pins with the `dont_touch` attribute
- If you use UPF with your design, you must convert the PG connections from RTL to UPF by using the `convert_pg` command before you compile the design

[Example 3-9](#) shows Verilog RTL code that instantiates two PG pins: `my_vdd` and `my_vss`.

Example 3-9 Coding PG Pins in the RTL Design

```
module my_design(a, b, c, my_vdd, my_vss);
  input a, b, my_vdd, my_vss;
  output c;
  my_macro U1(.a(a), .b(b), .c(c), .VDD(my_vdd), .VSS(my_vss));
endmodule
```

Performing Design Exploration

Developing new RTL and integrating it with third-party IP and many legacy RTL blocks can be a time-consuming process when designers lack a fast and efficient way to explore and improve the data, fix design issues, and create a better starting point for RTL synthesis. The DC Explorer tool allows you to perform early RTL exploration, leading to a better starting point for RTL synthesis and accelerating design implementation.

Use DC Explorer to do the following:

- Implement specific design goals, such as design rules and optimization constraints.
- Detect mismatches and missing constraints.
- Resolve mismatches and design data inconsistencies.

You can also create and modify floorplans early in the design cycle with floorplan exploration.

If design exploration fails to meet timing goals by more than 10 percent, modify your design goals and constraints, or improve the HDL code. Then, repeat both design exploration and functional simulation. If the design does not function as required, you must modify the HDL code and repeat both design exploration and functional simulation. Continue this process until the design is functioning correctly and is within 10 percent of the timing goals.

See Also

- [High-Level Design Flow Tasks](#)
- The DC Explorer documentation

Creating Constraints

Constraints are declarations that define the design's goals in measurable circuit characteristics, such as timing, area, power, and capacitance. The logic library defines implicit design rule constraints. These constraints are required for a design to function correctly. You can also define explicit optimization constraints. In topographical mode, physical constraints improve timing correlation with post-place-and-route tools, such as IC Compiler and IC Compiler II, by considering floorplanning information during optimization. Design Compiler needs these constraints to effectively optimize the design.

You can specify constraints interactively on the command line or specify them in a script file. If you specify constraints in a script file, use the .con extension.

To learn about the concepts and tasks necessary for defining design constraints, see [Defining Design Constraints](#) and [Using Floorplan Physical Constraints](#).

See Also

- [Using Script Files](#)

4

Setting Up and Working With Libraries

Design Compiler uses logic, symbol, and DesignWare libraries to implement design function and display synthesis results graphically. The logic libraries that Design Compiler maps to during optimization are called target libraries. Target libraries contain the cells used to generate the netlist and definitions for the design's operating conditions. The target libraries are the subset of the link libraries that are used to compile or translate a design. Link libraries define the delay models that are used to calculate timing values and path delays.

In DC Ultra, you can characterize your target logic library and create a pseudolibrary called ALIB, which maps Boolean functional circuits to actual gates from the target library. ALIB files provide Design Compiler with greater flexibility and a larger solution space to explore tradeoffs between area and delay during optimization.

In topographical mode, Design Compiler uses logic, symbol, and DesignWare libraries, plus physical libraries to obtain physical design information. In addition, you can use TLUPlus files for RC estimation and black box cells during synthesis.

To learn about libraries, how to set up libraries in wire load mode and topographical mode, and how to restrict the optimization of certain cells to a subset of the target library, see the following topics:

- [Selecting a Semiconductor Vendor](#)
- [Library Requirements](#)
- [Specifying Logic Libraries](#)
- [Specifying Physical Libraries](#)

- Using TLUPplus Files for RC Estimation
- Working With Libraries
- Target Library Subsets
- Library Subsets for Sequential Cells and Instantiated Combinational Cells
- Link Library Subsets
- Library-Aware Mapping and Synthesis
- Analyzing Multithreshold Voltage Library Cells
- Handling Black Boxes

Selecting a Semiconductor Vendor

One of the first things you must do when designing a chip is to select the semiconductor vendor and technology you want to use. Consider the following issues during the selection process:

- Maximum frequency of operation
- Physical restrictions
- Power restrictions
- Packaging restrictions
- Clock tree implementation
- Floorplanning
- Back-annotation support
- Design support for libraries, megacells, and RAMs
- Available IP cores
- Available test methods and scan styles

Library Requirements

In wire load mode, Design Compiler uses logic, symbol, and DesignWare libraries. To learn about these libraries, see the following topics:

- [Logic Libraries](#)
- [Symbol Libraries](#)
- [DesignWare Libraries](#)

In topographical mode, Design Compiler uses logic, symbol, and DesignWare libraries, plus physical libraries. Though not required, you can also use TLUPlus files for RC estimation and black boxes. To learn about physical libraries, see the following topic:

- [Physical Libraries](#)

Logic Libraries

Logic libraries, which are maintained and distributed by semiconductor vendors, contain information about the characteristics and functions of each cell, such as cell names, pin names, area, delay arcs, and pin loading. They also define the conditions that must be met, for example, the maximum transition time for nets. These conditions are called design rule constraints. In addition, a logic library specifies the operating conditions and wire load models for a specific technology.

Design Compiler supports logic libraries that use nonlinear delay models (NLDMs), Composite Current Source (CCS) models (either compact or noncompact), or both NLDM and CCS models. Design Compiler automatically selects the type of timing model to use based on the contents of the logic library. If a library contains both NLDM and CCS models, Design Compiler uses the CCS models. During logic synthesis and preroute optimization, the tool might not use all the available CCS data to save runtime.

Design Compiler requires the logic libraries to be in .db format. In most cases, your semiconductor vendor provides you with .db-formatted libraries. If you are provided with only library source code, see the Library Compiler documentation for information about generating logic libraries in .db format. To set up logic libraries, see [Specifying Logic Libraries](#).

Design Compiler uses logic libraries for the following purposes:

- Implementing the design function

The logic libraries that Design Compiler maps to during optimization are called target libraries. Target libraries contain the cells used to generate the netlist and definitions for the design's operating conditions. The target libraries are the subset of the link libraries that are used to compile or translate a design. Design Compiler saves this information in the design's `local_link_library` attribute.

- Resolving cell references

The logic libraries that Design Compiler uses to resolve cell references are called link libraries. Link libraries contain the descriptions of library cells and subdesigns in a mapped netlist and can also contain design files. Link libraries include local link libraries defined in the `local_link_library` attribute and system link libraries specified by the `link_library` variable.

- Calculating timing values and path delays

Link libraries define the delay models that are used to calculate timing values and path delays. For information about the various delay models, see the Library Compiler documentation.

- Calculating power consumption

For information about calculating power consumption, see the *Power Compiler User Guide*.

Design Compiler uses the first logic library found in the `link_library` variable as the main library. It uses the main library to obtain default values and settings used in the absence of explicit specifications for operating conditions, wire load selection group, wire load mode, and net delay calculation. If other libraries have measurement units different from the main library units, Design Compiler converts all units to those specified in the main library. Design Compiler obtains the following default values and settings from the main library:

- Unit definitions
- Operating conditions
- K-factors
- Wire load model selection
- Input and output voltage
- Timing ranges
- RC slew trip points
- Net transition time degradation tables

The logic library setup contains target libraries and link libraries:

- [Target Libraries](#)
- [Link Libraries](#)

Target Libraries

Design Compiler selects functionally correct gates from the target libraries to build a circuit during mapping. It also calculates the timing of the circuit by using the vendor-supplied timing data for these gates.

To specify the target libraries, use the `target_library` variable. You should specify only the standard cell libraries that you want Design Compiler to use for mapping the standard cells in your design, such as combinational logic and registers. You should not specify any DesignWare libraries or macro libraries, such as pads or memories.

For information about specifying target libraries, see [Specifying Logic Libraries](#).

Link Libraries

For a design to be complete, all cell instances in the design must be linked to the library components and designs that are referenced. This process is called linking the design or resolving references. To resolve references, Design Compiler uses the link libraries set by the following variables and attribute:

- The `link_library` application variable lists the libraries and design files that Design Compiler uses to resolve references.

Design Compiler searches the files listed in the `link_library` variable from left to right, and it stops searching when it finds a reference. Specifying an asterisk in the `link_library` variable means that Design Compiler searches loaded libraries in memory for the reference. For example, if you set the `link_library` variable to `{"**" lsi_10k.db}`, Design Compiler searches for the reference in memory first and then in the `lsi_10k` library.
- The `local_link_library` attribute lists the design files and libraries added to the beginning of the `link_library` variable during the link process. Design Compiler searches files in the `local_link_library` attribute first when it resolves references. You can set this attribute by using the `set_local_link_library` command.
- The `search_path` variable specifies a list of directory paths that the tool uses to find logic libraries and other files when you specify a plain file name without a path. It also sets the paths where Design Compiler can continue the search for unresolved references after it searches the link libraries.

If Design Compiler does not find the reference in the link libraries, it searches in the directories specified by the `search_path` variable, as described in [Specifying a Library Search Path](#).

Symbol Libraries

Symbol libraries contain definitions of the graphic symbols that represent library cells in design schematics. Semiconductor vendors maintain and distribute the symbol libraries.

Design Compiler uses symbol libraries to generate schematic views. You must use Design Vision to view the schematic. When you generate a schematic, Design Compiler performs a one-to-one mapping of cells in the netlist to cells in the symbol library.

Each Design Compiler installation includes a default symbol library file, `generic.sdb`, located in the `$SYNOPSYS_ROOT/libraries/syn` directory. This file contains generic symbols and all the Verilog standard logic gate symbols.

To load the schematic symbols, Design Vision first checks the symbol library files that you specify with the `symbol_library` variable. For example, a `symbol_library` value of `tech.sdb` prompts the tool to search for the symbols in the file named `tech.sdb`.

If the specified libraries do not contain the symbols, the tool searches the generic.sdb file in the \$SYNOPSYS_ROOT/libraries/syn directory. If no match is found, the tool displays the cell instance as a rectangle, which is the default representation.

DesignWare Libraries

A DesignWare library is a collection of reusable circuit-design building blocks (components) that are tightly integrated into the Synopsys synthesis environment. During synthesis, Design Compiler selects the component with the best speed and area optimization from the DesignWare library.

DesignWare components that implement many of the built-in HDL operators are provided by Synopsys. These operators include +, -, *, <, >, <=, >=, and the operations defined by if and case statements.

You can develop additional DesignWare libraries at your site by using DesignWare Developer, or you can license DesignWare libraries from Synopsys or from third parties. To use licensed DesignWare components, you need a license key for the components.

If any DesignWare component set in the `synthetic_library` variable requires a DesignWare license, Design Compiler checks for this license. You do not need to specify the standard synthetic library, `standard.sldb`, that implements the built-in HDL operators. Design Compiler automatically uses this library.

By default, if the `dw_foundation.sldb` library is not in the synthetic library list but the DesignWare license has been successfully checked out, the `dw_foundation.sldb` library is automatically added to the synthetic library list. This behavior applies to the current command only. The user-specified synthetic library and link library lists are not affected.

All DesignWare hierarchies are, by default, unconditionally ungrouped in the second pass of the compile. You can prevent this ungrouping by setting the `compile_ultra_ungroup_dw` variable to `false`. The default is `true`.

See Also

- [Specifying DesignWare Libraries](#)
- The DesignWare Library documentation

Physical Libraries

If you want to use Design Compiler in topographical mode, you need to specify physical libraries in addition to logic libraries. You use the Milkyway design library to specify physical libraries and save designs in Milkyway format.

The inputs required to create a Milkyway design library are the Milkyway reference library and the Synopsys technology file:

- Milkyway reference library

The Milkyway reference library contains the physical representation of standard cells and macros. In topographical mode, the Milkyway reference library uses the FRAM abstract view to store information. The reference library also defines the placement unit tile (the width and height of the smallest placeable instance and the routing directions).

- Synopsys technology file

The Milkyway technology file (.tf), contains technology-specific information required to route a design. Design Compiler automatically derives routing layer directions if your Milkyway library file is missing this information. Derived routing layer directions are saved in the .ddc file. To override the derived routing layer direction, use the `set_preferred_routing_direction` command. To report all routing directions, use the `report_preferred_routing_direction` command.

If you have only an IC Compiler II reference library with frame views, you can use it to generate Milkyway FRAM views. For more information, contact your Synopsys support representative.

Because multivoltage designs use power domains, these designs usually require certain library cells to be marked as always-on cells and certain library cell pins to be marked as always-on library cell pins. These always-on attributes are necessary to establish any always-on relationships between power domains. For more information, see the *Power Compiler User Guide*.

Note:

Occasionally, new attributes are added to the Synopsys technology file that are not yet supported in the Design Compiler tool. In these cases, the Design Compiler tool issues a TFCHK-009 error message. If the new attributes are safe to ignore, set the `ignore_tf_error` variable to `true`, which changes the error condition to a warning. For details, see [Fixing Errors Caused by New Unsupported Technology File Attributes](#).

See Also

- [Specifying Physical Libraries](#)
- [Using a Milkyway Database](#)

Specifying Logic Libraries

[Table 4-1](#) lists the variables that control library reading for each library type and the typical file name. You use these variables to specify logic libraries and DesignWare libraries.

Table 4-1 Library Variables

Library type	Variable	Default	File extension
Target library	target_library	{your_library.db}	.db
Link library	link_library	{* your_library.db}	.db
Symbol library	symbol_library	{your_library.sdb}	.sdb
DesignWare library	synthetic_library	""	.sldb

To set up access to the logic libraries, you must specify the target libraries and link libraries.

You can also use the Application Setup dialog box in the GUI to view, set, or change the library and search path variables for the current session. For more information, see the *Design Vision User Guide* and the “Setting Library Locations” topic in the Design Vision Help.

In the following example, the target library is the first link library. To simplify the link library definition, the example includes the `additional_link_lib_files` user-defined variable for libraries such as pads and macros.

```
prompt> set_app_var target_library [list_of_standard_cell_libraries]
prompt> set_app_var synthetic_library [list_of_sldb_files_for_DesignWare]
prompt> set additional_link_lib_files [additional_libraries]
prompt> set_app_var link_library [list * $target_library \
                                $additional_link_lib_files $synthetic_library]
```

If you are performing technology translation, add the standard cell library for the existing mapped gates to the link libraries and the standard cell library being translated to the target library.

Specifying DesignWare Libraries

You do not need to specify the standard synthetic library, standard.sldb, which implements the built-in HDL operators. The Design Compiler tool automatically uses this library.

However, if you are using additional DesignWare libraries, you must specify these libraries by using the `synthetic_library` variable for optimization and the `link_library` variable to resolve cell references:

```
prompt> set_app_var synthetic_library {dw_foundation.sldb}
prompt> set_app_var link_library [list * $target_library \
    $additional_link_lib_files $synthetic_library]
```

See Also

- [DesignWare Libraries](#)
 - The DesignWare Library documentation
-

Specifying a Library Search Path

You can specify the library location by using either the complete path or only the file name. If you specify only the file name, Design Compiler uses the search path defined in the `search_path` variable to locate the library files. By default, the search path includes the current working directory and `$SYNOPSYS/libraries/syn`, where `$SYNOPSYS` is the path to the installation directory. Design Compiler looks for the library files, starting with the leftmost directory specified in the `search_path` variable, and uses the first matching library file it finds.

For example, assume that you have logic libraries named `my_lib.db` in both the `lib` directory and the `vhdl` directory. Design Compiler uses the `my_lib.db` file found in the `lib` directory because it finds the `lib` directory first:

```
prompt> set_app_var search_path "lib vhdl default"
```

To see the order of the library files as they are found by Design Compiler, use the `which` command:

```
prompt> which my_lib.db
/usr/lib/my_lib.db, /usr/vhdl/my_lib.db
```

You can also use the Application Setup dialog box in the GUI to view, set, or change the library search path for the current session. For more information about using the GUI, see the *Design Vision User Guide* and the Design Vision Help.

Setting Minimum Timing Libraries

If you are performing simultaneous minimum and maximum timing analysis, the logic libraries specified by the `link_library` variable are used for both maximum and minimum timing information. To specify a separate minimum timing library, use the `set_min_library` command. The `set_min_library` command associates minimum timing libraries with the maximum timing libraries specified in the `link_library` variable. For example,

```
prompt> set_app_var link_library {* maxlib.db}
prompt> set_min_library maxlib.db -min_version minlib.db
```

To find out which libraries have been set to be the minimum and maximum libraries, use the `list_libs` command. In the generated report, the lowercase letter m appears next to the minimum library and the uppercase letter M appears next to the maximum library.

See Also

- [Library Requirements](#)

Specifying Physical Libraries

The inputs required to create a Milkyway design library are the Milkyway reference library and the Milkyway technology file.

To create a Milkyway design library,

1. Create the Milkyway design library by using the `create_mw_lib` command.

For example,

```
dc_shell-topo> create_mw_lib -technology $mw_tech_file \
    -mw_reference_library $mw_reference_library $mw_design_library_name
```

2. Open the Milkyway library that you created by using the `open_mw_lib` command.

For example,

```
dc_shell-topo> open_mw_lib $mw_design_library_name
```

3. (Optional) Attach the TLUPlus files (which provide more accurate capacitance and resistance data) by using the `set_tlu_plus_files` command.

For example,

```
dc_shell-topo> set_tlu_plus_files -max_tluplus $max_tlu_file \
    -min_tluplus $min_tlu_file -tech2itf_map $sprs_map_file
```

4. In subsequent sessions, use the `open_mw_lib` command to open the Milkyway library. If you are using the TLUPlus files for RC estimation, use the `set_tlu_plus_files` command to attach these files.

For example,

```
dc_shell-topo> open_mw_lib $mw_design_library_name
dc_shell-topo> set_tlu_plus_files -max_tluplus $max_tlu_file \
    -min_tluplus $min_tlu_file -tech2itf_map $prs_map_file
```

Note:

The tool does not link physical libraries automatically when you use the `analyze` and `elaborate` commands to read your designs. It might issue the LINK-5 error and the OPT-1428 warning messages. To manually link the physical libraries, use the `enable_phys_lib_during_elab true` variable.

The following Milkyway library commands are also supported: `copy_mw_lib`, `close_mw_lib`, `report_mw_lib`, `current_mw_lib`, and `check_tlu_plus_files`.

See Also

- [Library Requirements](#)
- [Using a Milkyway Database](#)
- [Using TLUPlus Files for RC Estimation](#)

Using TLUPlus Files for RC Estimation

If TLUPlus files are available or are used in your back-end flow, you can use them for RC estimation in Design Compiler topographical mode. TLUPlus files contain resistance and capacitance look-up tables and model ultra deep submicron (UDSM) process effects. TLUPlus files provide more accurate capacitance and resistance data, thereby improving correlation with back-end results. Using TLUPlus files is available only in topographical mode.

To specify the TLUPlus files used for virtual route and postroute extraction, use the `set_tlu_plus_files` command. At a minimum, you must specify the maximum TLUPlus file and the mapping file. The mapping file maps layer names between the Milkyway technology file and the process Interconnect Technology Format (ITF) file. For example,

```
dc_shell-topo> set_tlu_plus_files -max_tluplus max.tlup \
    -tech2itf_map layer_map.map
```

If the layer names are the same in your technology file and ITF file, you do not need to specify the mapping file; however, if the layer names are different and you do not specify the

mapping file, an error will occur later in the flow. For more information about the mapping file, see the Milkyway documentation.

To check TLUPlus settings, use the `check_tlu_plus_files` command. To verify that you are using the TLUPlus files, check the `compile_ultra` log for the following message:

```
*****  
Information: TLU Plus based RC computation is enabled.  
(RCEX-141)  
*****
```

You can read nxtgrd files for extraction as an alternative to TLUPlus files. You specify an nxtgrd file the same way you specify a TLUPlus file, by using the `set_tlu_plus_files` command. Supporting nxtgrd files aligns Design Compiler in topographical mode with the StarRC and PrimeTime tools, eliminating the need to maintain a separate TLUPlus file.

The following example sets the `NXTGRD_file_max.nxtgrd` file for the maximum condition calculation of the resistance and capacitance:

```
dc_shell-topo> set_tlu_plus_files -max_tluplus NXTGRD_file_max.nxtgrd \  
-tech2itf_map layer_map.map
```

You must generate the nxtgrd file using the grdgenxo tool version N-2017.12 or later.

To read nxtgrd files for 5 nm and smaller technology nodes, the DC-Synth-AdvGeo license is required. If the license is not available, the tool issues an error message.

If you used the `set_tlu_plus_files` command to specify the TLUPlus technology files, the tool performs extraction based on TLUPlus technology. Otherwise, the tool performs extraction using the extraction parameters in your physical library. Use the `set_extraction_options` command to specify the parameters that influence extraction and the `report_extraction_options` command to report the parameters that influence the post-route extraction.

To perform RC extraction for the virtual routing of design nets, use the `extract_rc` command. The command calculates delays based on the Elmore delay model and can update back-annotated delay and capacitance numbers on nets. This command is available only in Design Compiler in topographical mode. Use the command after editing the netlist.

Working With Libraries

You can perform the following tasks by using simple library commands:

- [Loading Libraries](#)
- [Listing Libraries](#)
- [Reporting Library Contents](#)

- [Specifying Library Objects](#)
- [Excluding Cells From the Target Libraries](#)
- [Verifying Library Consistency](#)
- [Removing Libraries From Memory](#)
- [Saving Libraries](#)

Loading Libraries

Design Compiler uses binary libraries, .db format for logic libraries and .sdb format for symbol libraries, and automatically loads these libraries when needed. To manually load a binary library, use the `read_file` command:

```
prompt> read_file my_lib.db
prompt> read_file my_lib.sdb
```

If your library is not in the appropriate binary format, use the `read_lib` command to compile the library source. The `read_lib` command requires a Library-Compiler license.

Listing Libraries

Design Compiler refers to a library loaded in memory by its name. The library statement in the library source defines the library name. To list the names of the libraries loaded in memory, use the `list_libs` command:

```
prompt> list_libs
Logical Libraries:
Library          File          Path
-----          ----          ----
my_lib           my_lib.db    /synopsys/libraries
my_symbol_lib   my_lib.sdb   /synopsys/libraries
```

Reporting Library Contents

To report the contents of a library, use the `report_lib` command. The command reports the following information:

- Library units
- Operating conditions
- Wire load models
- Cells (including cell exclusions, preferences, and other attributes)

Specifying Library Objects

Library objects are the vendor-specific cells and their pins. To specify library objects, use the following naming convention:

```
[file:]library/cell[/pin]
```

where *file* is the file name of a logic library, *library* is the name of a library loaded in memory, *cell* is a library cell, and *pin* is a cell's pin. If you have multiple libraries loaded in memory with the same name, you must specify the file name.

For example, to set the `dont_use` attribute on the AND4 cell in the `my_lib` library, enter

```
prompt> set_dont_use my_lib/AND4
```

To set the `disable_timing` attribute on the Z pin of the AND4 cell in the `my_lib` library, enter

```
prompt> set_disable_timing [get_pins my_lib/AND4/z]
```

Excluding Cells From the Target Libraries

When Design Compiler maps a design to a logic library, it selects library cells from this library. To specify cells in the target library to be excluded during optimization, use the `set_dont_use` command. For example, to prevent Design Compiler from using the INV_HD high-drive inverter, enter

```
prompt> set_dont_use MY_LIB/INV_HD
```

This command affects only the copy of the library that is currently loaded in memory and has no effect on the version that exists on disk. However, if you save the library, the exclusions are saved, and the cells are permanently excluded.

To remove the `dont_use` attribute set by the `set_dont_use` command, use the `remove_attribute` command. For example,

```
prompt> remove_attribute MY_LIB/INV_HD dont_use  
MY_LIB/INV_HD
```

You can also restrict optimization of a design block by a subset of the target library by using the `set_target_library_subset` command. You can restrict the library cells to be used in a particular block or filter the target library cells on a block-by-block basis.

See Also

- [Target Library Subsets](#)

Verifying Library Consistency

Consistency between the logic library and the physical library is critical to achieving good results. Before you process your design, make sure that your libraries are consistent by running the `check_library` command. If the `check_library` command reports any inconsistencies, you must fix these inconsistencies before you process your design.

```
prompt> check_library
```

The `check_library` command performs the following checks:

- Checks the integrity of individual logical and physical libraries
- Checks consistency between logic libraries
- Checks consistency between logic libraries and physical libraries
- Checks consistency between physical libraries and technology files

By default, the `check_library` command performs consistency checks between the logic libraries specified in the `link_library` variable and the physical libraries in the current Milkyway design library. You can also explicitly specify logic libraries by using the `-logic_library_name` option or Milkyway reference libraries by using the `-mw_library_name` option. If you explicitly specify libraries, these override the default libraries.

You can use the `set_check_library_options` command to set options for the `check_library` command to perform various logic library and physical library checks, such as the bus naming style, area of each cell, and so forth.

To see the enabled library consistency checks, run the `report_check_library_options -logic_vs_physical` command.

See Also

- See the Library Compiler documentation for more information about logic libraries.
- See the Milkyway documentation for more information about physical libraries.

Removing Libraries From Memory

To remove libraries from dc_shell memory, use the `remove_design` command. If you have multiple libraries with the same name loaded into memory, you must specify both the path and the library name. To see the path for each library in memory, use the `list_libs` command.

Saving Libraries

The `write_lib` command saves (writes to disk) a compiled library in the Synopsys database or VHDL format. To write out the shell commands to save the current link library settings for design instances, use the `write_link_library` command.

To optimize tool performance, the `write_lib` command is disabled upon startup of the tool. To enable usage of the `write_lib` command, you must execute the `enable_write_lib_mode` command at the beginning of the tool session. Use this command before any library files have been read in, either explicitly or implicitly. You should use this tool session only for library file generation, not optimization.

You can determine whether the `write_lib` command is enabled by using the `report_write_lib_mode` command.

Target Library Subsets

Design Compiler allows you to restrict optimization of a design block using a subset of the target library. You can check for errors and conflicts introduced by target library subsets, find out which target library subsets have been defined both for the hierarchical cells and at the top level, and remove a target library subset constraint from the design or a design instance:

- [Specifying Target Library Subsets](#)
- [Checking Target Library Subsets](#)
- [Reporting Target Library Subsets](#)
- [Removing Target Library Subsets](#)

See Also

- [Target Libraries](#)

Specifying Target Library Subsets

Design Compiler allows you to restrict optimization of a design block using a subset of the target library. Normally, optimization can select any library cell from the target library. However, you can restrict the library cells to be used in a particular block or filter the target library cells on a block-by-block basis with the `set_target_library_subset` command. For example, you can omit specific double-height cells in some blocks even though they are mixed with other needed library cells in the target library. The subset restriction only applies

to new cells that are created or mapped during optimization. It does not affect cells that are already mapped unless Design Compiler encounters a reason to modify the cells during optimization.

When you use the `set_target_library_subset` command, the subset applies to the specified blocks and their subblocks. Applying the `set_target_library_subset` command to a hierarchical cell or to the top-level design enforces the library restriction on all lower cells in the hierarchy, except for those cells that have a different library subset constraint explicitly set on them. A subset at a lower level overrides any subset specified at a higher level. Design Compiler does not incrementally refine the upper subset.

You cannot ungroup a subdesign if a subset restriction is applied to it by the `set_target_library_subset` command. Similarly, the tool's auto ungrouping capability cannot ungroup the subdesign.

To restrict a list of blocks or top-level cells to a specific target library subset during optimization, use the `-object_list` option. The cells are instances of hierarchical designs, and the subset restriction applies to these cells and their child instances. The target library subset cannot be specified on leaf-level cells. If you do not specify the `-object_list` option, Design Compiler sets the target library subset on the current design.

To specify a list of libraries that are available to optimize the identified design instances, use the `library_list` argument. These libraries must also be specified by the `target_library` variable. If you do not specify the `library_list` argument, all the libraries listed in the `target_library` variable can be used.

To specify library cells that cannot be used for optimization even if they are inside the libraries specified by the `library_list` argument, use the `-dont_use` option. You can specify the library cells by name or by collection in a space-separated list. You can also use a wildcard (*).

Use the `-only_here` option to specify library cells that can be used for optimization within the block but cannot be used in other blocks (unless those blocks also list the library cell in an `-only_here` option). This essentially applies a `-dont_use` option on these cells for all other target library subsets in the design, including the target library subset at the top level, unless the subsets also contain an `-only_here` option for these cells. You can specify the library cells by name or by collection in a space-separated list. You can also use a wildcard (*).

If the `-dont_use` cell list and the `-only_here` cell list include the same cell name after wildcard expansion, the `-only_here` cell list takes precedence, and Design Compiler reports this with an information message.

If you specify a library name with the `-dont_use` and `-only_here` options, Design Compiler ignores the specified library and reports an information message. For example, if you run the following command, Design Compiler issues a message saying that library name `lib1` is irrelevant and is ignored:

```
set_target_library_subset -top -dont_use {AN2 */OR* lib1/NOR2}
```

In this case, Design Compiler restricts the use of all library cells named NOR2.

You cannot override a `dont_use` attribute if it is set in a library. If the `dont_use` attribute is set on a target library cell, it cannot be used even if listed in the `-only_here` cell list. Design Compiler issues a warning if you apply the `-only_here` option to a target library cell that has a `dont_use` attribute set on it. If you want to make target library cells selectively available for use with the `set_target_library_subset` command, you must first remove the `dont_use` attribute from the library with the `remove_attribute` command.

To restrict the scope of the `set_target_library_subset` command to only cells in clock paths, not data paths, specify the `-clock_path` option. This option has priority over any conflicting target library subset specified for the same instances without the `-clock_path` option. You can use this feature to guide the mapping of clock-gating and clock-steering logic in clock paths.

Setting Target Library Subset Examples

The following example shows how to set the target library for the design to the full set of libraries “lib1.db lib2.db” but restrict the library cells used in block u1 to the cells in library lib2.db:

```
dc_shell-topo> set_app_var target_library "lib1.db lib2.db"
dc_shell-topo> set_target_library_subset "lib2.db" -object_list \
    [get_cells u1]
```

The following example restricts the library cells avoid1 and avoid2 so they will not be used in blocks u1 and u2:

```
dc_shell-topo> set_target_library_subset -object_list "u1 u2" \
    -dont_use "avoid1 avoid2"
```

The following example restricts library cell SPECIAL so it can only be used in blocks u1 and u2:

```
dc_shell-topo> set_target_library_subset -object_list "u1 u2" \
    -only_here "SPECIAL"
```

The following example achieves the same effect as the previous example. The second `set_target_library_subset` command gives no restrictions and opens up the entire target library in blocks u1 and u2, including SPECIAL.

```
dc_shell-topo> set_target_library_subset -top -dont_use "SPECIAL"
dc_shell-topo> set_target_library_subset -object_list "u1 u2"
```

The following example restricts the use of the INVX1, MUX1, and NOR2 library cells so they are used on the clock path during optimization:

```
dc_shell-topo> set_target_library_subset clocklib.db \
    -clock_path -use [INVX1 MUX1 NOR2]
```

Checking Target Library Subsets

To check for errors and conflicts introduced by target library subsets, use either the `check_mv_design -target_library_subset` or `check_target_library_subset` command. The commands check for the following conditions:

- Conflicts between target library subsets and the global `target_library` variable
- Conflicts between operating condition and target library subset
- Conflicts between the library cell of a mapped cell and target library subset

The `check_mv_design -target_library_subset` command searches the design for any cells that do not follow the rules for the subset. If encountered, Design Compiler reports them as warnings. They are not reported as errors because they might have already existed in the design before the subset was specified. Using the `check_mv_design -target_library_subset` command can be helpful to identify issues if you run the command before optimization and then run the command again after optimization to see if you get the same results.

Reporting Target Library Subsets

To find out which target library subsets have been defined both for the hierarchical cells and at the top level, use the `report_target_library_subset` command with the appropriate library cell list.

Reports that are generated by reporting commands, such as `report_cell` and `report_timing`, include a `td` attribute, indicating that a cell specified with the `-dont_use` option or the `-only_here` option is used somewhere in the design.

Removing Target Library Subsets

To remove a target library subset constraint from the design or a design instance, use the `remove_target_library_subset` command with the appropriate library list and cell list, or use the `reset_design` command.

Library Subsets for Sequential Cells and Instantiated Combinational Cells

Design Compiler in topographical mode allows you to restrict the mapping and optimization of sequential cells and instantiated combinational cells in a design to a user-specified subset of library cells in a target library.

To learn how to apply, report, and remove library cell subset specifications, see

- [Specifying the Library Cell Subsets](#)
- [Reporting the Library Cell Subsets](#)
- [Removing the Library Cell Subsets](#)

See Also

- [Target Libraries](#)

Specifying the Library Cell Subsets

Design Compiler in topographical mode allows you to restrict the mapping and optimization of sequential cells and instantiated combinational cells in a design to a user-specified subset of library cells in a target library. To define a subset of library cells, use the `define_libcell_subset` command. The command specifies a list of library cells and defines them as a family if they meet the following criteria:

- They must have the same functional identification.
- They must be sequential library cells or instantiated combinational cells.
- They cannot belong to another family.

You can define multiple subsets. However, when library cells are grouped into a subset family, they are not available for general mapping during the compile run. Design Compiler uses the library cells that you specified in the subset exclusively to map the sequential cells or instantiated combinational cells.

After you define the library cell subset, you must list the sequential cells or instantiated combinational cells to be mapped and optimized and then specify the library cell subset by using the `set_libcell_subset` command:

- To specify a list of sequential cells or instantiated combinational cells, use the `-object_list` option. The cells must be either unmapped or instantiated as one of the library cells in the library cell subset.
- To specify the library cell subset to be used during optimization, use the `-family_name` option.

When you specify the `define_libcell_subset` command followed by the `set_libcell_subset` command, Design Compiler restricts all optimizations, including cell sizing and cell swapping, to the cells specified within the library cell subset. The subset restriction applies only to cells that are created, mapped, or modified during optimization. The subset does not affect any unoptimized portions of the design.

In the following example, the `define_libcell_subset` command groups logic library cells SDFLOP1 and SDFLOP2 into a family called “specialflops.” Next, the `set_libcell_subset` command sets the library cell subset for instances reg01 and reg02 to the user-defined cell family, `specialflops`, which consists of the cells SDFLOP1 and SDFLOP2:

```
dc_shell-topo> define_libcell_subset -libcell_list {SDFLOP1 SDFLOP2} \
               -family_name specialflops
dc_shell-topo> set_libcell_subset -object_list [get_cells {reg01 reg02}] \
               -family_name specialflops
```

Reporting the Library Cell Subsets

To report information about a library cell subset that is specified for sequential cells or instantiated combinational cells, use the `report_libcell_subset` command. The command only reports library subset information that is explicitly user-defined. In other words, it reports the library cell subset family specified by the `define_libcell_subset` command.

The following example reports the library cell subset family for the reg01 sequential cell:

```
dc_shell-topo> report_libcell_subset -object_list [get_cells reg01]
*****
Report : library cell subset
Design : chip
Version: ...
Date   : ...
*****
Libcell Family      Libcells
-----
specialflops        sff0 sff1 sff2
-----
Object             Reference       Libcell Family
-----
reg01              sff0           specialflops
-----
1
```

See Also

- [Specifying the Library Cell Subsets](#)

Removing the Library Cell Subsets

To remove a library cell subset constraint from specified sequential cells or instantiated combinational cells or remove a library cell subset that was created by the `define_libcell_subset` command, use the `remove_libcell_subset` command.

A library cell can only belong to one subset; therefore, if you need to change the subset of a library cell, you should remove the existing subset definition first by using the `remove_libcell_subset` command.

In the following example, the library subset constraint that was set by the `set_libcell_subset` command is removed from the reg01 and reg02 instances:

```
dc_shell-topo> remove_libcell_subset -object_list [get_cells {reg01 reg02}]
```

In the following example, the user-defined specialflops library cell subset that was created by the `define_libcell_subset` command is removed:

```
dc_shell-topo> remove_libcell_subset -family_name specialflops
```

See Also

- [Specifying the Library Cell Subsets](#)

Link Library Subsets

Design Compiler allows you to restrict the selection of library cells so they are chosen from a subset of the libraries specified by the `link_library` variable. You can report the link library subset on the design and remove a link library subset from the design or a design instance.

- [Specifying Link Library Subsets](#)
- [Reporting Link Library Subsets](#)
- [Removing Link Library Subsets](#)

See Also

- [Link Libraries](#)

Specifying Link Library Subsets

Design Compiler allows you to restrict the selection of library cells so they are chosen from a subset of the libraries specified by the `link_library` variable. Normally, library cells are selected from any library in the `link_library` variable. However, when you use the

`set_link_library_subset` command, Design Compiler restricts the selection of library cells, which can resolve ambiguity among libraries with the same voltage, temperature, and process.

The command resolves ambiguity among libraries that are characterized for the same voltage, temperature, and process, but that are characterized differently for other known arbitrary parameters. For example, in a multivoltage or multicorner design, the tool would normally determine a cell's timing and power from a library in the `link_library` whose voltage, temperature, and process match the cell. If more than one library satisfies these conditions, the tool issues an "Ambiguous Libraries" warning (MV-086). You can use the `set_link_library_subset` command to identify which library is appropriate for a particular block or scenario.

The `set_link_library_subset` command does not override existing library selection rules; it augments them. The tool still matches voltage, temperature, and process within the libraries listed in the subset. The link library subset provides an additional filter on the libraries to be considered. You could, for example, use the command to make macro libraries unambiguous by specifying just the relevant macro library, without the need to list all the other libraries in the link library.

To specify a list of cells for which the link library subset will be used, use the `set_link_library_subset` command with the `-object_list` option. The cells must be instances of hierarchical designs, macros, or pads. The subset restriction applies to these cells and their child instances.

To specify a subset of libraries belonging to the `link_library` variable, use the `set_link_library_subset` command with the `library_list` argument. For any reference name that appears in a library of the subset, only library cells within the subset are used. For reference names that do not appear in the subset, library cells are selected from any library in the `link_library` variable, as usual.

Note:

The `set_link_library_subset` command does not work the same way as the `set_target_library_subset` command. You do not need to specify anything to make optimization work correctly with the link library subsets. Link library subsets are taken into account when Design Compiler decides which library cells have suitable characterizations. The `set_target_library_subset` command restricts the library cells to be used in a particular block or filters the target library cells on a block-by-block basis. However, the `set_target_library_subset` command does not select individual characterizations.

Setting Link Library Subset Examples

The following example has two libraries, lib1.db and lib2.db, with the same voltage, temperature, and process. Ordinarily this would result in an “Ambiguous Libraries” warning, but it does not because the ambiguity is resolved. Library values are taken from lib1.db everywhere in the design, except for the hierarchy under cell u1, which uses values from lib2.db.

```
dc_shell-topo> set_app_var link_library "* lib1.db lib2.db"
dc_shell-topo> set_link_library_subset "lib1.db" -top
dc_shell-topo> set_link_library_subset "lib2.db" -object_list [get_cells u1]
```

The following example uses two macro libraries, extracted as models by PrimeTime with different parasitics in effect.

```
dc_shell-topo> set_app_var link_library "* stdlib.db macro_corner1.db \
macro_corner2.db"
dc_shell-topo> create_scenario A
dc_shell-topo> set_link_library_subset "macro_corner1.db" \
-object_list [get_cells u1/my_macro]
dc_shell-topo> create_scenario B
dc_shell-topo> set_link_library_subset "macro_corner2.db" \
-object_list [get_cells u1/my_macro]
```

Reporting Link Library Subsets

To report the link library subset on the design based on the specified options, use the `report_link_library_subset` command.

The following example reports the link library subset for the top-level design and for the u1 instance:

```
dc_shell-topo> report_link_library_subset -top -object_list [get_cells u1]
```

Removing Link Library Subsets

To remove a link library subset from the design or a design instance, use the `remove_link_library_subset` or `reset_design` command.

The following example removes the link library subset from the top-level design and from the u1 instance:

```
dc_shell-topo> remove_link_library_subset -object_list [get_cells u1] -top
```

Library-Aware Mapping and Synthesis

Using DC Ultra in wire load mode or topographical mode, you can characterize (or analyze) your target logic library and create a pseudolibrary called ALIB, which maps Boolean functional circuits to actual gates from the target library. Design Compiler reads the ALIB file during compile. The ALIB file provides Design Compiler with greater flexibility and a larger solution space to explore more intelligent tradeoffs between area and delay during optimization. You must use the `compile_ultra` command to get the benefits from the ALIB library.

Library characterization occurs during the initial stage of compile. Because it can take time for Design Compiler to characterize each logic library, it is recommended that you generate the ALIB file when you install Design Compiler, and store it in a single repository so that multiple users can share the library.

Generating the ALIB File

To generate the ALIB library corresponding to your target logic library, use the `alib_analyze_libs` command. The tool creates a release-specific subdirectory in the location specified by the `alib_library_analysis_path` variable and stores the generated ALIB files in this directory. For example, the following sequence of commands creates the `x.db.alib` file for the target library `x.db` and stores it in `/remote/libraries/alib/alib-51`.

```
prompt> set_app_var target_library "x.db"
prompt> set_app_var link_library "x.db"
prompt> set_app_var alib_library_analysis_path "/remote/libraries/alib"
prompt> alib_analyze_libs
```

Design Compiler creates the `alib-51` directory for version control; each release has a different version.

Using the ALIB Library

To load the previously generated ALIB library, use the `alib_library_analysis_path` variable to point to the location of the file. For example,

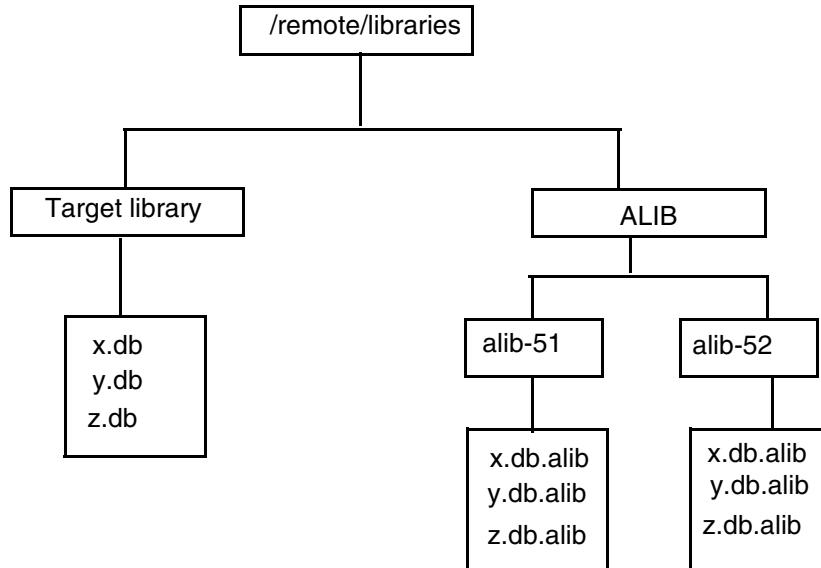
```
prompt> set_app_var alib_library_analysis_path "remote/libraries/alib"
```

During compile, if no pre-generated ALIB libraries exist, Design Compiler performs library characterization automatically. It generates the ALIB library in the location specified by the `alib_library_analysis_path` variable. If you have not set this variable, the ALIB library is stored in the current working directory. The tool uses this ALIB library for subsequent runs.

It is recommended that you generate the ALIB library for your target logic library when you install Design Compiler. Create a directory structure similar to the one you use for storing

libraries as shown in [Figure 4-1](#). If you install a new version of Design Compiler, you must regenerate the ALIB library.

Figure 4-1 Directory Structure for ALIB Library



Analyzing Multithreshold Voltage Library Cells

To compare the leakage power with respect to the timing characteristics of the target library cells belonging to each threshold voltage group, Design Compiler supports the `analyze_library -multi_vth` command. [Example 4-1](#) shows the report generated by the command.

Example 4-1 Report Generated by the analyze_library -multi_vth Command

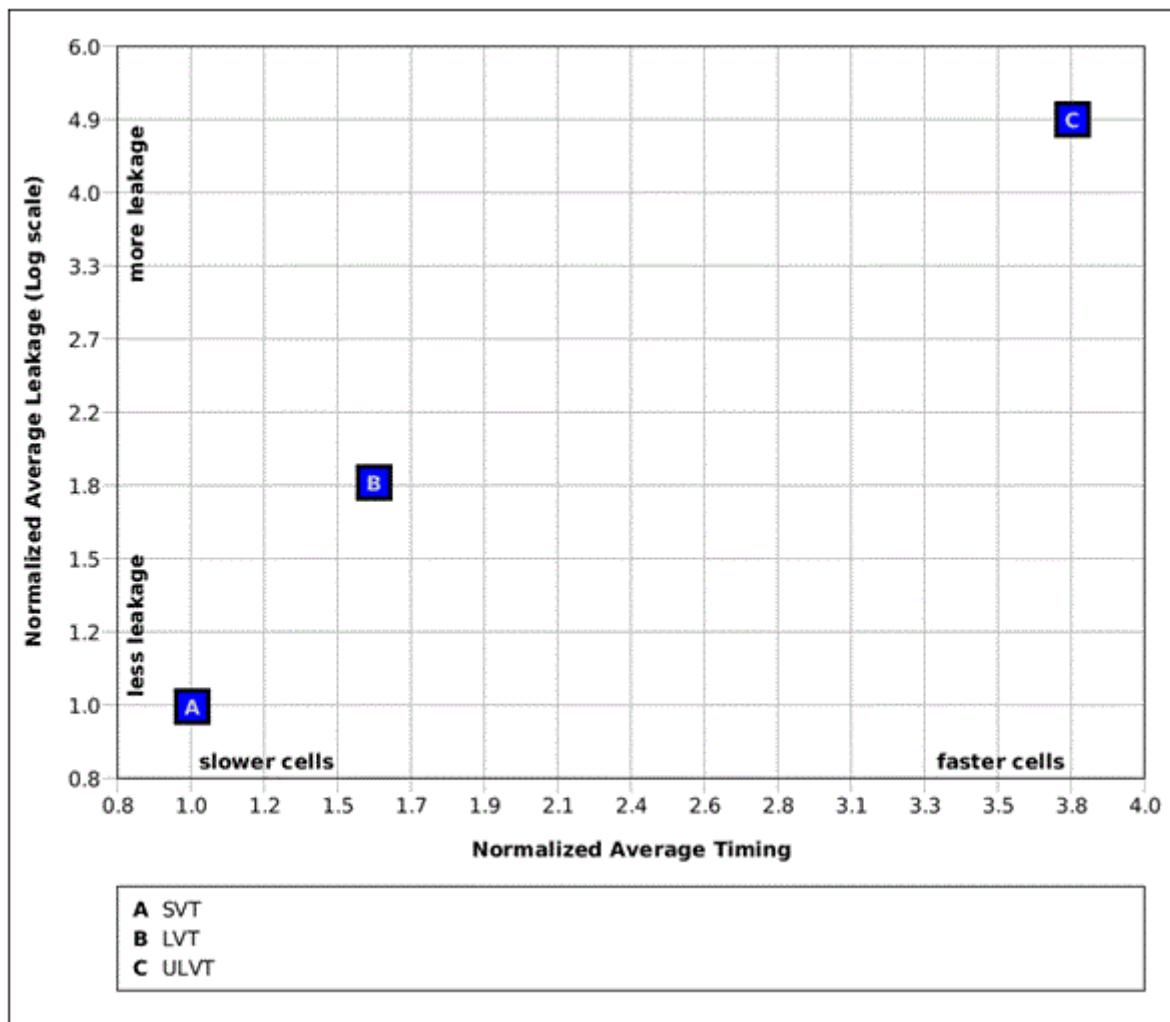
```
*****
Multi-VT Library Analysis Report
*****
Vth Group/Library Name          Avg.          Avg.
(don't use cells/total cells)  Leakage        Timing
*****
SVT (0/998)                    1.00          1.00 (Baseline)
LVT (0/793)                     1.80          1.59
ULVT (0/998)                   4.88          3.75
*****
```

You can also generate a graphical representation of the information using the `-graph` option as shown in [Figure 4-2](#). The results of the `analyze_library` command are shown relative to a baseline library. In this figure, the baseline library is the SVT voltage threshold group. This is the library that the tool considers to have the least leakage; it is also known as an HVt

(high voltage threshold group). The timing and leakage values are measured relative to this baseline. For instance, in [Figure 4-2](#), the ULVT voltage threshold group has 4.88 times more leakage than the baseline SVT group. The ULVT voltage threshold group is also 3.75 times faster in terms of delay than the SVT group. This information is provided so you can see how the tool views the performance and leakage of your voltage threshold groups relative to each other. This information is presented in a design-independent way and does not rely upon any design specific constraints.

The graph in [Figure 4-2](#) can help you determine which library to use for your optimization goals. If your design is power sensitive, you might avoid using the ULVT voltage threshold group to save power. If your design has high-performance targets, you might want to exclude the SVT group in order to achieve your performance goals. In this case, you will sacrifice power for performance.

Figure 4-2 Graphical Output Generated by the analyze_library -graph Command



See Also

- [Leakage Power Optimization Based on Threshold Voltage](#)
-

Handling Black Boxes

Design Compiler in topographical mode supports synthesis with black boxes. Black boxes can be cells where the logic functionality is not known, cells that do not link to a cell in the logic library, or cells that do not have physical representation.

For information about working with black boxes, see

- [Supported Black Boxes](#)
 - [Black Box Flow](#)
 - [Defining Timing in Quick Timing Model Format](#)
 - [Identifying Black Box Cells](#)
-

Supported Black Boxes

The following types of black boxes are supported:

- Functionally unknown black boxes

These are cells where the logic functionality is not known. Examples include the following types of cells:

- Macro cells
- Empty hierarchy cells or black-boxed modules
- Unlinked or unresolved cells

- Logical black boxes

These are cells that do not link to a cell in the logic library. This type of black box is categorized under functionally unknown black box cells. Examples include the following types of cells:

- Empty hierarchy cells or black-boxed modules
- Unlinked or unresolved cells

You can define the timing for logical black box cells by using Design Compiler in wire load mode or topographical mode, as described in [Defining Timing in Quick Timing Model Format](#).

- Physical black boxes

These are cells that do not have physical representation. Examples include the following types of cells:

- Empty hierarchy cells or black-boxed modules
- Unlinked or unresolved cells

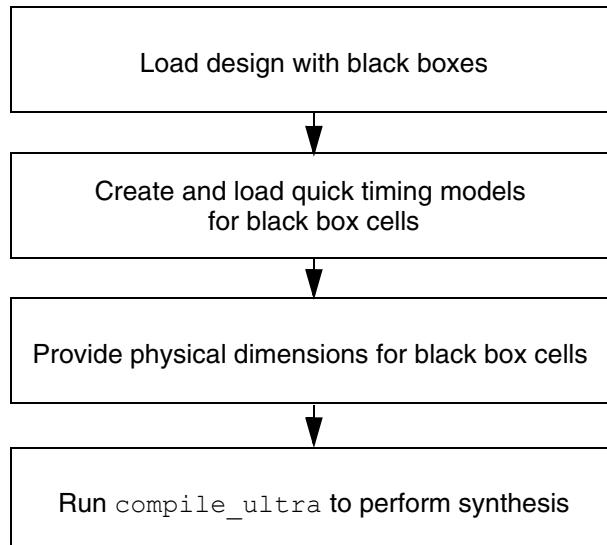
Topographical technology allows you to define the physical dimensions for physical black box cells, as described in [Defining Physical Dimensions](#).

If the physical representation of a cell is not available with the user-supplied physical libraries, Design Compiler in topographical mode defines it automatically. For more information, see [Automatic Creation of Physical Library Cells](#).

Black Box Flow

[Figure 4-3](#) shows the flow for defining the timing and physical dimensions for black box cells and synthesizing the design.

Figure 4-3 Supported Black Box Flow



You can visually examine black boxes in your floorplan by viewing them in the Design Vision layout window. You can control the visibility, selection, and display style properties of black-box cells in the active layout view by setting options on the View Settings panel. Black boxes are visible and enabled for selection by default. For more information about controlling the visibility of black boxes in the active layout view, see Design Vision Help.

Defining Timing in Quick Timing Model Format

Design Compiler allows you to define timing for logical black box cells. You can create logic library cells and provide a timing model for the new cells in the quick timing model format. A quick timing model is an approximate timing model that is useful early in the design cycle to describe the rough initial timing of a black box. You create a quick timing model for a black box by specifying the model ports, the setup and hold constraints on the inputs, the clock-to-output path delays, and the input-to-output path delays. You can also specify the loads on input ports and the drive strength of output ports. Design Compiler saves the timing models in .db format, which can then be used during synthesis.

To create a quick timing model for a simple black box in Design Compiler in wire load mode or in topographical mode, perform the following steps:

1. Create a new model using the `create_qtm_model` command:

```
dc_shell-topo> create_qtm_model BB
```

BB is the model name.

2. Specify the technology information, such as the name of the logic library, the maximum transition time, the maximum capacitance, and the wire load information.

```
dc_shell-topo> set_qtm_technology -library library_name
dc_shell-topo> set_qtm_technology -max_transition trans_value
dc_shell-topo> set_qtm_technology -max_capacitance cap_value
```

3. Specify global parameters, such as setup and hold characteristics.

```
dc_shell-topo> set_qtm_global_parameter -param setup -value
setup_value
dc_shell-topo> set_qtm_global_parameter -param hold -value hold_value
dc_shell-topo> set_qtm_global_parameter -param clk_to_output \
-value cto_value
```

4. Specify the ports using the `create_qtm_port` command:

```
dc_shell-topo> create_qtm_port -type input A
dc_shell-topo> create_qtm_port -type input B
dc_shell-topo> create_qtm_port -type output OP
```

5. Specify delay arcs using the `create_qtm_delay_arc` command:

```
dc_shell-topo> create_qtm_delay_arc -name A_OP_R -from A \
-from_edge rise -to OP -value 0.10 -to_edge rise
dc_shell-topo> create_qtm_delay_arc -name A_OP_F -from A \
-from_edge fall -to OP -value 0.11 -to_edge fall
dc_shell-topo> create_qtm_delay_arc -name B_OP_R -from B \
-from_edge rise -to OP -value 0.15 -to_edge rise
dc_shell-topo> create_qtm_delay_arc -name B_OP_F -from B \
-from_edge fall -to OP -value 0.16 -to_edge fall
```

6. (Optional) Generate a report that shows the defined parameters, the ports, and the timing arcs in the quick timing model.

```
dc_shell-topo> report_qtm_model
```

7. Save the quick timing model using the `save_qtm_model` command:

```
dc_shell-topo> save_qtm_model
```

8. Write out the .db file for the quick timing model using the `write_qtm_model` command:

```
dc_shell-topo> write_qtm_model -out_dir QTM
```

9. Add the .db file to the `link_library` variable to load the quick timing model:

```
dc_shell-topo> lappend link_library "QTM/BB.db"
```

Defining Physical Dimensions

Design Compiler allows you to set the size for physical black box cells based on an estimation of the objects that it will contain when replaced with real logic. You can also define the base unit area for gate equivalence calculations for estimating the size of black boxes. The following topics discuss these strategies for defining the physical dimensions of black box cells:

- [Estimating the Size of Black Boxes](#)
- [Determining the Gate Equivalent Area](#)

Estimating the Size of Black Boxes

To set the size and shape for physical black box cells based on an estimation of the objects that it will contain, use the `estimate_fp_black_boxes` command.

To set the width and height, use the `-sm_size` option:

```
dc_shell-topo> estimate_fp_black_boxes -sm_size size
```

To create a rectilinear black box, use the `-polygon` option:

```
dc_shell-topo> estimate_fp_black_boxes -polygon polygon_area
```

The following example estimates a black box named alu1 and specifies it as a soft macro with a size of 100x100 and a utilization of 0.7:

```
dc_shell-topo> estimate_fp_black_boxes \
    -sm_size {100 100} -sm_util 0.7 \
    [get_cells -filter "is_physical_black_box==true" alu1]
```

The following example estimates a black box named alu1 and specifies it as a rectilinear soft macro:

```
dc_shell-topo> estimate_fp_black_boxes \
    -polygon {{1723.645 1925.365} {1723.645 1722.415} \
    {1803.595 1722.415} {1803.595 1530.535} \
    {799.915 1530.535} {799.915 1925.365} \
    {1723.645 1925.365}} \
    [get_cells -filter "is_physical_black_box==true" alu1]
```

In the following example, the size of the black box named U1 is estimated from the size of hard macro ram16x128:

```
dc_shell-topo> estimate_fp_black_boxes -hard_macros ram16x128 U1
```

Determining the Gate Equivalent Area

Design Compiler also allows you to define the base unit area for gate equivalence calculations for estimating the size of black boxes. Use the `set_fp_base_gate` command to specify either a library leaf cell area or a user-specified cell area as the base unit area to be used for gate equivalence calculations.

To specify a gate from the library as the base unit area to be used for the calculations, use the `-cell` option:

```
dc_shell-topo> set_fp_base_gate -cell UNIT
```

In this example, UNIT specifies the reference name of the library leaf cell to be used as the base unit area for gate equivalence calculations.

To specify the cell area in square microns as the base unit area to be used for the calculations, use the `-area` option:

```
dc_shell-topo> set_fp_base_gate -area 10
```

In this example, the base gate area is set to 10 square microns.

Identifying Black Box Cells

Design Compiler sets the following attributes on black box cells:

- `is_black_box`

When you specify the `is_black_box` attribute with the `get_cells` command, Design Compiler identifies functionally unknown black boxes:

```
dc_shell-topo> get_cells -hier -filter "is_black_box==true"
```

- `is_logical_black_box`

When you specify the `is_logical_black_box` attribute with the `get_cells` command, Design Compiler identifies the logical black boxes in the design:

```
dc_shell-topo> get_cells -hier -filter "is_logical_black_box==true"
```

- `is_physical_black_box`

When you specify the `is_physical_black_box` attribute with the `get_cells` command, Design Compiler identifies the physical black boxes in the design:

```
dc_shell-topo> get_cells -hier -filter "is_physical_black_box==true"
```

Automatic Creation of Physical Library Cells

If the physical representation of a cell is not available with the user-supplied physical libraries, Design Compiler defines it automatically. The tool can create physical library cells for the following:

- Logic library cells (leaf cells and macros)
- Empty hierarchy cells or black-boxed modules
- Unlinked or unresolved cells
- Unmapped cells

The tool issues the following warning message when it creates physical library cells:

```
Warning: Created physical library cell for logical library
%s. (OPT-1413)
```

5

Using a Milkyway Database

The Milkyway database is the unifying design storage format for the Synopsys Galaxy™ Design Platform. The database provides persistent storage of physical design data that links Galaxy platform tools together. The database is periodically updated with new features to support advances in EDA technology.

Design Compiler writes a mapped, unqualified design into the Milkyway database, including the netlist, synthesis constraints, and any physical guidance information, when you use the `write_milkyway` command. You can use a single Milkyway library across the entire Galaxy flow.

Note:

Design Compiler does not support the `read_milkyway` command.

When you use a Milkyway database, you do not need to use an intermediate netlist file exchange format such as Verilog or VHDL to communicate with other Synopsys Galaxy platform tools. Before you can use a Milkyway database within Design Compiler, you must prepare a design library and a reference library and understand the following concepts and tasks:

- [About the Milkyway Database](#)
- [Guidelines for Using the Milkyway Database](#)
- [Preparing to Use the Milkyway Database](#)
- [Writing the Milkyway Database](#)

About the Milkyway Database

The Milkyway database stores design data in the Milkyway design library and physical library data in the Milkyway reference library.

- Milkyway design library

The Milkyway directory structure used to store design data—that is, the unqualified, mapped netlist and constraints—is referred to as the Milkyway design library. You specify the Milkyway design library for the current session by setting the `mw_design_library` variable to the root directory path.

- Milkyway reference library

The Milkyway directory structure used to store physical library data is referred to as the Milkyway reference library. Reference libraries contain standard cells, macro cells, and pad cells. For information about creating reference libraries, see the Milkyway documentation.

You specify the Milkyway reference library for the current session by setting the `mw_reference_library` variable to the root directory path. The order in the list implies priority for reference conflict resolution. If more than one reference library has a cell with the same name, the first reference library has precedence.

Before you use a Milkyway database, see

- [Required License and Files](#)
 - [Invoking the Milkyway Environment Tool](#)
-

Required License and Files

Before you use a Milkyway database, you need to have the following required license and files:

- The Milkyway-Interface license
Design Compiler provides this license, which is used to run the `write_milkyway` command.
- Source for logic libraries (.lib)
- Compiled databases
 - Logic libraries (.db), which contains standard cell timing, power, function, test, and so forth
 - Milkyway library (FRAM), which contains technology data

Invoking the Milkyway Environment Tool

To invoke the Milkyway Environment tool, enter

```
% Milkyway -galaxy
```

The command checks out the Milkyway-Interface license. The Milkyway Environment tool is a graphical user interface (GUI) that enables manipulation of the Milkyway libraries. You can use the tool to maintain your Milkyway design library, such as delete unused versions of your design.

See Also

- The Milkyway documentation

Guidelines for Using the Milkyway Database

When you use the `write_milkyway` command, observe these guidelines:

- Make sure all the cells present in the Milkyway reference library have corresponding cells in the timing library. The port direction of the cells in the Milkyway reference libraries are set from the port direction of cells in the timing library. If cells are present in the Milkyway reference library but are not in the timing library, the port direction of cells present in the Milkyway reference library is not set.
- Run the `uniquify` command before you run the `write_milkyway` command.
- You must make sure the units in the logic library and the Milkyway technology file are consistent.

The SDC file does not contain unit information. If the units in the logic library and Milkyway technology file are inconsistent, the `write_milkyway` command cannot convert them automatically. For example, if the logic library uses femtofarad as the capacitance unit and the Milkyway technology file uses picofarad as the capacitance unit, the output of the `write_sdc` command shows different net load values.

In the following example, the capacitance units in the logic library and the Milkyway technology file are not consistent. The following `set_load` information is shown for the net `gpdhi_word_d_21_` before the `write_milkyway` command is run:

```
set_load 1425.15 [get_nets {gpdhi_word_d_21_}]
```

After the `write_milkyway` command is run, the SDC file shows

```
set_load 8.36909 [get_nets {gpdhi_word_d_21_}]
```

- Design Compiler is case-sensitive. You can use the tool in case-insensitive mode by doing the following tasks before you run the `write_milkyway` command:
 - Prepare uppercase versions of the libraries used in the link library.
 - Use the `change_names` command to make sure the netlist is uppercased.

Preparing to Use the Milkyway Database

You use the Milkyway design library to specify physical libraries and save designs in the Milkyway format. The inputs required to create a Milkyway design library are the Milkyway reference library and the Milkyway technology file.

To create a Milkyway design library,

1. Create the Milkyway design library by using the `create_mw_lib` command.

For example,

```
dc_shell-topo> create_mw_lib -technology $mw_tech_file \
    -mw_reference_library $mw_reference_library $mw_design_library_name
```

2. Open the Milkyway library that you created by using the `open_mw_lib` command.

For example,

```
dc_shell-topo> open_mw_lib $mw_design_library_name
```

3. (Optional) Attach the TLUPlus files by using the `set_tlu_plus_files` command.

For example,

```
dc_shell-topo> set_tlu_plus_files -max_tluplus $max_tlu_file \
    -min_tluplus $min_tlu_file -tech2itf_map $prs_map_file
```

4. In subsequent sessions, use the `open_mw_lib` command to open the Milkyway library. If you are using the TLUPlus files for RC estimation, use the `set_tlu_plus_files` command to attach these files.

For example,

```
dc_shell-topo> open_mw_lib $mw_design_library_name
dc_shell-topo> set_tlu_plus_files -max_tluplus $max_tlu_file \
    -min_tluplus $min_tlu_file -tech2itf_map $prs_map_file
```

See Also

- [Using TLUPlus Files for RC Estimation](#)

Describes how to use TLUPlus files to provide more accurate capacitance and resistance data

Writing the Milkyway Database

To save the design data in a Milkyway design library, use the `write_milkyway` command. The `write_milkyway` command writes netlist and physical data from memory to the Milkyway design library format, re-creates the hierarchy preservation information, and saves the design data for the current design in a design file. The path for the design file is `design_dir/CEL/file_name:version`, where `design_dir` is the location you specified in the `mw_design_library` variable.

For example, you use the `-output` option to specify the file name.

```
dc_shell-topo> write_milkyway -output my_file -overwrite
```

To write design information from memory to a Milkyway library named `testmw` and name the design file `TOP`, enter

```
dc_shell-topo> set_app_var mw_design_library testmw
dc_shell-topo> write_milkyway -output TOP
```

[Example 5-1](#) shows a script to set up and write a Milkyway database.

Example 5-1 Script to Set Up and Write a Milkyway Database

```
set_app_var search_path "$search_path ./libraries"
set_app_var link_library "* max.lib.db"
set_app_var target_library "max.lib.db"

create_mw_lib -technology $mw_tech_file -mw_reference_library \
               $mw_reference_library $mw_lib_name
open_mw_lib $mw_lib_name
read_file -format ddc design.ddc
current_design TopDesign
link
write_milkyway -output myTop
```

Important Points About the `write_milkyway` Command

When you use the `write_milkyway` command, keep the following points in mind:

- You must run the `create_mw_lib` command before running the `write_milkyway` command.
- If a design file already exists (that is, you ran the `write_milkyway` command on the design with the same output directory), the `write_milkyway` command creates an additional design file and increments the version number. You must ensure that you open the correct version in Milkyway; by default, Milkyway opens the latest version. To avoid creating an additional version, specify the `-overwrite` option to overwrite the current version of the design file and save disk space.

- The command does not modify in-memory data.
- Attributes present in the design in memory that have equivalent attributes in Milkyway are translated (not all attributes present in the design database are translated).
- A hierarchical netlist translated by using the `write_milkyway` command retains its hierarchy in the Milkyway database.

Limitations When Writing Milkyway Format

The following limitations apply when you write your design in Milkyway format:

- The design must be mapped.

Because the Milkyway format describes physical information, it supports mapped designs only. You cannot use the Milkyway format to store design data for unmapped designs.

- The design must not contain multiple instances.

You must uniquify your design before saving it in Milkyway format. Use the `check_design -multiple_designs` command to report information related to multiply-instantiated designs.

- The `write_milkyway` command saves the entire hierarchical design in a single Milkyway design file. You cannot generate separate design files for each subdesign.
- When you save a design in Milkyway format, the `write_milkyway` command does not save the block abstraction instances in the Milkyway design library. You must explicitly save each block abstraction in .ddc format. For more information about block abstraction usage, see [Using Hierarchical Models](#).

6

Working With Designs in Memory

The Design Compiler tool reads designs into memory from design files. Many designs can be in memory at any time. After a design is read in, you can change it in numerous ways, such as grouping or ungrouping its subdesigns or changing subdesign references.

To learn how to work with designs in memory, see

- [Reading Designs](#)
- [Listing Designs in Memory](#)
- [Setting the Current Design](#)
- [Linking Designs](#)
- [Listing Design Objects](#)
- [Specifying Design Objects](#)
- [Creating Designs](#)
- [Copying Designs](#)
- [Renaming Designs](#)
- [Changing the Design Hierarchy](#)
- [Editing Designs](#)
- [Translating Designs From One Technology to Another](#)

- [Removing Designs From Memory](#)
- [Saving Designs](#)

Reading Designs

See the following topics for an overview of the design formats supported by Design Compiler and the commands you can use to read designs:

- [Supported Design Input Formats](#)
- [Reading HDL Files](#)
- [Reading .ddc Files](#)
- [Reading .db Files](#)

Supported Design Input Formats

Before you read a design, make sure that your design is in one of the following formats listed in [Table 6-1](#).

Table 6-1 Supported Input Formats

Format	Description
.ddc	Synopsys internal database format (recommended)
.db	Synopsys internal database format
Verilog	IEEE standard Verilog (see the HDL Compiler documentation)
VHDL	IEEE standard VHDL (see the HDL Compiler documentation)
SystemVerilog	IEEE standard SystemVerilog (see the HDL Compiler documentation)
equation	Synopsys equation format
pla	Berkeley (Espresso) PLA format

Reading HDL Files

You can open HDL design files by using one of the following methods:

- [Reading Designs With Dependencies Automatically](#)

You can read designs with dependencies automatically by using the `-autoread` option with the `analyze` or `read_file` command. The tool automatically analyzes and elaborates designs and any files dependent on them in the correct order.

- [Running the read_file Command](#)

The `read_file` command analyzes the design and translates it into a technology-independent (GTECH) design in a single step.

- [Running the analyze and elaborate Commands](#)

The `analyze` command checks the design and reports errors. The `elaborate` command translates the design into a technology-independent design (GTECH) from the intermediate files produced during analysis.

- [Running the read_verilog or read_vhdl Command](#)

The commands checks the code for correct syntax and build a generic technology (GTECH) netlist that Design Compiler uses to optimize the design. You can use the `read_verilog` or `read_vhdl` commands to do both functions automatically.

See Also

- The HDL Compiler documentation

Reading Designs With Dependencies Automatically

You can read designs with dependencies automatically by using the `-autoread` option with the `read_file` or `analyze` command:

```
prompt> read_file -autoread verilog my_design.v
```

The tool automatically analyzes and elaborates designs and any files dependent on them in the correct order.

The `read_file -autoread` command analyzes and elaborates the top-level design; the `analyze -autoread` command analyzes the design but does not perform elaboration.

When you use the `-autoread` option, the resulting GTECH representation is retained in memory. Dependencies are determined only from the files or directories in the `file_list`; if the `file_list` changes between consecutive `-autoread` option runs, the tool uses the latest set of files to determine the dependencies. You can use the `-autoread` option with any VHDL, Verilog, or SystemVerilog language version.

See Also

- [Running the read_file Command](#)
- [Running the analyze and elaborate Commands](#)

Running the `read_file` Command

The `read_file` command analyzes the design and translates it into a technology-independent (GTECH) design in a single step:

```
prompt> read_file -format verilog my_design.v
```

The following HDL formats are supported:

- Verilog format—the .v, .verilog, .v.gz, and .verilog.gz extensions
The command does not create intermediate files for Verilog. To create intermediate files, set the `hdlin_auto_save_templates` variable to `true`.
- SystemVerilog—the .sv, .sverilog, .sv.gz, and .sverilog.gz extensions
- VHDL—the .vhd, .vhdl, vhd.gz, and .vhdl.gz extensions
The command creates .mr and .st intermediate files for VHDL.

If you do not specify the design format, the `read_file` command infers the format based on the file extension. If no known extension is used, the tool uses .ddc format. Supported extensions for automatic inference are not case-sensitive.

The `read_file` command can read compressed files for all formats except the .ddc format, which is compressed internally as it is written. To enable the tool to automatically infer the file format for compressed files, use the following naming structure: `filename.format.gz`.

For designs in memory, Design Compiler uses the `path_name/design.ddc` naming convention. The `path_name` argument is the directory from which the original file was read, and the `design` argument is the name of the design. If you later read in a design that has the same file name, Design Compiler overwrites the original design. To prevent this, use the `-single_file` option with the `read_file` command.

The `read_file` command does not execute the `link` command automatically. For more information, see [Linking Designs](#).

See Also

- [Differences Between the `read_file` Command and the `analyze` and `elaborate` Commands](#)

Running the `analyze` and `elaborate` Commands

The `analyze` command performs the following tasks:

- Reads an HDL source file
- Checks for errors without building generic logic for the design
- Creates HDL library objects in an HDL-independent intermediate format

- Stores the intermediate files in a location you define

If the `analyze` command reports errors, fix them in the HDL source file and then run the `analyze` command again. After a design is analyzed, you must reanalyze it only when you change it.

The `elaborate` command performs the following tasks:

- Translates the design into a technology-independent design (GTECH) from the intermediate files produced during analysis
- Allows changing of parameter values defined in the source code
- Allows VHDL architecture selection
- Replaces the HDL arithmetic operators in the code with DesignWare components
- Automatically executes the `link` command, which resolves design references

To use this method, analyze the top-level design and all subdesigns in bottom-up order and then elaborate the top-level design and any subdesigns that require parameters to be assigned or overwritten:

```
prompt> analyze -format verilog -library -work RISCTYPES.v
prompt> analyze -format verilog -lib -work {ALU.v STACK_TOP.v \
                      STACK_MEM.v ...}
prompt> elaborate RISC_CORE -architecture STRUCT -library WORK -update
```

See Also

- [Differences Between the `read_file` Command and the `analyze` and `elaborate` Commands](#)

Differences Between the `read_file` Command and the `analyze` and `elaborate` Commands

[Table 6-2](#) summarizes the differences between using the `read_file` command and using the `analyze` and `elaborate` commands to read design files.

Table 6-2 The `read_file` Command Versus the `analyze` and `elaborate` Commands

Comparison	<code>read_file</code> command	<code>analyze</code> and <code>elaborate</code> commands
Input formats	All formats are supported.	VHDL and Verilog formats are supported.
When to use	Use to synthesize netlists, precompiled designs, and so on.	Use to synthesize VHDL or Verilog files.

Table 6-2 The read_file Command Versus the analyze and elaborate Commands (Continued)

Comparison	read_file command	analyze and elaborate commands
Parameters	Does not allow you to pass parameters. You must use directives in HDL.	You can set parameter values with the <code>elaborate</code> command. For parameterized designs, you can use the <code>analyze</code> and <code>elaborate</code> commands to build a new design with nondefault values.
Architecture	Does not allow you to specify the architecture to be elaborated.	You can specify the architecture to be elaborated.
Linking designs	You must use the <code>link</code> command to resolve references.	The <code>elaborate</code> command executes the <code>link</code> command automatically to resolve references.

Running the read_verilog or read_vhdl Command

The `read_verilog` and `read_vhdl` commands are derived from the `read_file` command. Both commands check the code for correct syntax and build a generic technology (GTECH) netlist that Design Compiler uses to optimize the design. To run either command, specify the file type:

```
prompt> read_vhdl my_design.vhdl
```

See Also

- [Running the read_file Command](#)

Reading .ddc Files

To read the design data from a `.ddc` file, use the `read_ddc` command or the `read_file -format ddc` command:

```
prompt> read_ddc design_file.ddc
```

The `.ddc` format is backward compatible but not forward compatible. You can read a `.ddc` file that was generated with an earlier software version, but you cannot read a `.ddc` file that was generated with a later software version.

See Also

- [Running the read_file Command](#)

Reading .db Files

Note:

Although you can read files in Synopsys database .db format, it is recommended that you use .ddc format.

To read in a .db file, use the `read_db` command or the `read_file -format db` command:

```
prompt> read_db my_design_file.db
```

Design Compiler supports the .db, .sldb, .sdb, .db.gz, .sldb.gz, and .sdb.gz file extensions. The version of a .db file is the version of Design Compiler that created the file. For a .db file to be read into Design Compiler, its file version must be the same as or earlier than the version of Design Compiler you are running. If you attempt to read in a .db file generated by a Design Compiler version that is later than the Design Compiler version you are using, an error message appears. The error message provides details about the version mismatch.

See Also

- [Running the `read_file` Command](#)
 - [Reading .ddc Files](#)
-

Listing Designs in Memory

To list the names of the designs loaded in memory, use the `list_designs` command.

```
prompt> list_designs
A (*)    B      C
1
```

The asterisk (*) next to design A shows that A is the current design.

To list the memory file name corresponding to each design name, use the `-show_file` option.

```
prompt> list_designs -show_file

/user1/designs/design_A/A.ddc
A (*)  

/home/designer/dc/B.ddc
B      C
1
```

The asterisk (*) next to design A shows that A is the current design. File B.ddc contains both designs B and C.

Setting the Current Design

You can set the current design (the design you are working on) in the following ways:

- With the `read_file` command

When the `read_file` command successfully finishes processing, it sets the current design to the design that was read in.

```
prompt> read_file -format ddc MY_DESIGN.ddc
Reading ddc file '/designs/ex/MY_DESIGN.ddc'
Current design is 'MY_DESIGN'
```

- With the `elaborate` command

- With the `current_design` command

Use this command to set any design in dc_shell memory as the current design.

```
prompt> current_design ANY_DESIGN
Current design is 'ANY_DESIGN'.
{ANY_DESIGN}
```

You should avoid writing scripts that use a large number of `current_design` commands, such as in a loop. Using a large number of `current_design` commands can increase runtime. For more information, see *Using Tcl With Synopsys Tools*.

To display the name of the current design, enter the following command:

```
prompt> printvar current_design
current_design = "test"
```

Linking Designs

For a design to be complete, it must connect to all the library components and designs it references. This process is called *linking the design* or *resolving references*.

To link designs, uses the `link` command. The `link` command uses the `link_library` and `search_path` system variables and the `local_link_library` attribute to resolve design references.

By default, the case sensitivity of the linking process depends on the source of the references. To explicitly define the case sensitivity of the linking process, set the `link_force_case` variable.

To learn how Design Compiler links designs and resolves references, see [How the Tool Resolves References](#).

In addition to linking designs and resolving references, you can perform the following tasks:

- [Locating Designs by Using a Search Path](#)
- [Changing Design References](#)
- [Querying Design References](#)
- [Mapping Physical Variant Cells in Netlists or DEF Files](#)

How the Tool Resolves References

The Design Compiler tool resolves references by carrying out the following steps:

1. It determines which library components and subdesigns are referenced in the current design and its hierarchy.
2. It searches the link libraries to locate these references.
 - a. Design Compiler first searches the libraries and design files defined in the current design's `local_link_library` attribute.
 - b. If an asterisk is specified in the value of the `link_library` variable, Design Compiler searches in memory for the reference.
 - c. Design Compiler then searches the libraries and design files defined in the `link_library` variable.
3. If it does not find the reference in the link libraries, it searches in the directories specified by the `search_path` variable.

For more information, see [Locating Designs by Using a Search Path](#).

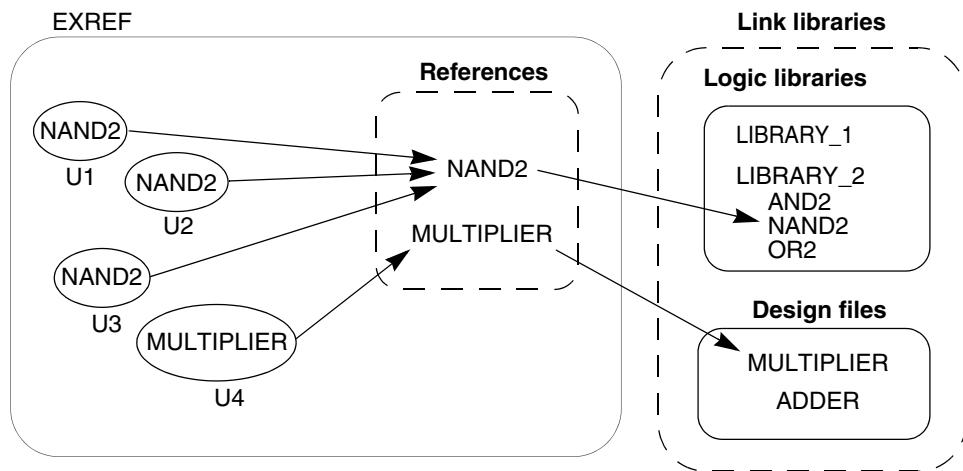
4. It links (connects) the located references to the design.

Note:

In a hierarchical design, Design Compiler considers only the top-level design's local link library. It ignores local link libraries associated with the subdesigns.

Design Compiler uses the first reference it locates. If it locates additional references with the same name, it generates a warning message identifying the ignored, duplicate references. If Design Compiler does not find the reference, a warning appears advising that the reference cannot be resolved.

The arrows in [Figure 6-1](#) show the connections that the linking process added between the instances, references, and link libraries. In this example, Design Compiler finds library component NAND2 in the LIBRARY_2 logic library; it finds subdesign MULTIPLIER in a design file.

Figure 6-1 Resolving References

Locating Designs by Using a Search Path

You can specify the design file location by using the complete path or only the file name. If you specify only the file name, the Design Compiler tool uses the search path defined in the `search_path` variable. The tool looks for the design files starting with the leftmost directory specified in the `search_path` variable and uses the first design file it finds. By default, the search path includes the current working directory and `$SYNOPSYS/libraries/syn`, where `$SYNOPSYS` is the path to the installation directory. To see where Design Compiler finds a file when using the search path, use the `which` command. For example, enter

```
prompt> which my_design.ddc
{/usr/designers/example/my_design.ddc}
```

To specify other directories in addition to the default search path, use the following command:

```
prompt> lappend search_path project
```

See Also

- [Specifying a Library Search Path](#)

Changing Design References

To change the component or design to which a cell or reference is linked, use the `change_link` command. You can use the command on a hierarchical design from any level in the design without using the `current_design` command to specify the current design. When you specify a cell instance as the object, the link for that cell is changed. When you specify a reference as the object, the links for all cells having that reference are changed. Design Compiler changes the link only when you specify a component or design that has the same number of ports with the same size and direction as the original reference.

Running the `change_link` command copies all link information from the old design to the new design. If the old design is a synthetic module, all attributes of the old synthetic module are moved to the new link. After running the `change_link` command, you must run the design with the `link` command on the design.

If any cell specified in the object list is an instance at a lower level in the hierarchy and its parent cell is not unique, use the `-all_instances` option. All similar cells under the same parent design are automatically linked to the new reference design. You do not have to change the current design to change the link for the instance cells in the lower design hierarchy.

The following command shows how cells U1 and U2 are linked from the current design to MY_ADDER:

```
prompt> copy_design ADDER MY_ADDER
prompt> change_link {U1 U2} MY_ADDER
```

The following command changes the link for cell U1, which is at a lower level in the hierarchy:

```
prompt> change_link top/sub_inst/U1 lsi_10k/AN3
```

This example shows how you can use the `-all_instances` option to change the link for inv1, when its parent design, bot, is instantiated multiple times. The design bot is instantiated twice: mid1/bot1 and mid1/bot2.

```
prompt> change_link -all_instances mid1/bot1/inv1 lsi_10k/AN3
```

```
Information: Changed link for all instances of cell 'inv1'
in subdesign 'bot'. (UID-193)
```

```
prompt> get_cells -hierarchical -filter "ref_name == AN3"
```

```
{mid1/bot1/inv1 mid1/bot2/inv1}
1
```

You can direct the tool to resolve a cell reference that is equivalent to the original cell reference but has different pin names. To map the pin names between the original and new

cell references, use the `-pin_map {{old_pin1 new_pin1} {old_pin2 new_pin2}...}` option with the `change_link` command.

For example, the following command links the U1 cell in the current design to the AN21 cell reference and maps the old pin names (A1, A2, and Z) to new pin names (A, B, and Y):

```
prompt> change_link [get_cells U1] AN21 \
    -pin_map {{A1 A} {A2 B} {Z Y}}
```

Querying Design References

You can query design references by using the following methods:

- To report information about all the references in the current instance or the current design, use the `report_reference` command. To display information across the hierarchy, use the `-hierarchy` option.
- To return a collection of instances that have a specific reference, use the `get_references` command.

For example, the following command returns a collection of instances in the current design that have the reference AN2:

```
prompt> get_references AN2
{U2 U3 U4}
```

- To report the reference names, use the `report_cell` command.

For example,

```
prompt> report_cell [get_references AN*]
```

Cell Attributes	Reference	Library	Area
U2	AN2	lsi_10k	2.000000
U3	AN2	lsi_10k	2.000000
U4	AN2	lsi_10k	2.000000
U8	AN3	lsi_10k	2.000000

Mapping Physical Variant Cells in Netlists or DEF Files

You can map physical variant cells in a netlist or a DEF file to the master cell in the library. Physical variant cells differ from the master library cell in mask color or small physical geometry. For advanced technology nodes, the timing characteristics of these cells do not vary, so you can use the same Liberty model for the master cell and its variants.

To enable this feature, set the `link_allow_physical_variant_cells` variable to `true`. The default is `false`.

To manage the library with efficiency, you need to define a list of physical variant cells of a master library cell by using the `physical_variant_cells` attribute in Liberty syntax. When reading a netlist or DEF file containing physical variant cells, the Design Compiler tool

- Resolves the physical variant cell references by searching the list specified by the `physical_variant_cells` attribute
- Maps the physical variant cells to their master cell and preserves the cells throughout the design flow

Listing Design Objects

The Design Compiler tool provides commands for accessing various design objects. These commands refer to design objects located in the current design. Each command in [Table 6-3](#) performs one of the following actions:

- List
Provides a listing with minimal information.
- Display
Provides a report that includes characteristics of the design object.
- Return
Returns a collection that can be used as input to another dc_shell command.

[Table 6-3](#) lists the commands and the actions they perform.

Table 6-3 Commands to Access Design Objects

Object	Command	Action
Instance	<code>list_instances</code>	Lists instances and their references.
	<code>report_cell</code>	Displays information about instances.
Reference	<code>report_reference</code>	Displays information about references.
Port	<code>report_port</code>	Displays information about ports.
	<code>report_bus</code>	Displays information about bused ports.
	<code>all_inputs</code>	Returns all input ports.
	<code>all_outputs</code>	Returns all output ports.
Net	<code>report_net</code>	Displays information about nets.
	<code>report_bus</code>	Displays information about bused nets.

Table 6-3 Commands to Access Design Objects (Continued)

Object	Command	Action
Clock	report_clock	Displays information about clocks.
	all_clocks	Returns all clocks.
Register	all_registers	Returns all registers.
Collections	get_*	Returns a collection of cells, designs, libraries and library cell pins, nets, pins, and ports.

You can also use the GUI to list design objects and attribute values, provide reports, and return collections. To list and display information about designs, cells, nets, ports, and pins, use the List menu commands. To generate design object reports, use the Design menu commands. For more information about using the GUI, see the *Design Vision User Guide* and the Design Vision Help.

Specifying Design Objects

You can specify design objects by using either a relative path or an absolute path:

- [Using a Relative Path](#)
- [Using an Absolute Path](#)
- [Using Attributes](#)

Using a Relative Path

If you use a relative path to specify a design object, the object must be in the current design. Specify the path relative to the current instance. The current instance is the frame of reference within the current design. By default, the current instance is the top level of the current design. Use the `current_instance` command to change the current instance.

For example, to place a `dont_touch` attribute on hierarchical cell U1/U15 in the Count_16 design, you can enter either

```
prompt> current_design Count_16
Current design is 'Count_16'.
{Count_16}
prompt> set_dont_touch U1/U15
```

or

```
prompt> current_design Count_16
Current design is 'Count_16'.
{Count_16}
prompt> current_instance U1
Current instance is '/Count_16/U1'.
/Count_16/U1
prompt> set_dont_touch U15
1
```

In the first command sequence, the frame of reference remains at the top level of design Count_16. In the second command sequence, the frame of reference changes to instance U1. Design Compiler interprets all future object specifications relative to instance U1.

To reset the current instance to the top level of the current design, enter the `current_instance` command without an argument.

```
prompt> current_instance
Current instance is the top-level of the design 'Count_16'
```

The `current_instance` variable points to the current instance. To display the current instance, enter the following command:

```
prompt> printvar current_instance
current_instance = "Count_16/U1"
```

Using an Absolute Path

When you use an absolute path to specify a design object, the object can be in any design in dc_shell memory. Use the following syntax to specify an object by using an absolute path:

[file:]design/object

file

The path name of a memory file followed by a colon (:). Use the file argument when multiple designs in memory have the same name.

design

The name of a design in dc_shell memory.

object

The name of the design object, including its hierarchical path. If several objects of different types have the same name and you do not specify the object type, Design Compiler looks for the object by using the types allowed by the command.

To specify an object type, use the `get_*` command. For more information about these commands, see *Using Tcl With Synopsys Tools*.

For example, to place a `dont_touch` attribute on hierarchical cell U1/U15 in the Count_16 design, enter

```
prompt> set_dont_touch /usr/designs/Count_16.ddc:Count_16/U1/U5  
1
```

Using Attributes

An attribute is a string or value associated with an object in the design that carries some information about that object. Attributes can describe logical, electrical, physical, and other properties. They are attached to the design object and saved with the design database.

Design Compiler uses attributes on the following types of objects:

- Entire designs
- Design objects, such as clocks, nets, pins, and ports
- Design references and cell instances within a design
- Library cells and cell pins

Each attribute has a name, a type, and a value. Attributes can have the following types: string, numeric, or logical (Boolean).

Some attributes are predefined and are recognized by Design Compiler; other attributes are user-defined.

Some attributes are read-only. Design Compiler sets these attribute values and you cannot change them. Other attributes are read/write. You can change these attribute values at any time.

Most attributes apply to one object type; for example, the `rise_drive` attribute applies only to input and inout ports. Some attributes apply to several object types; for example, the `dont_touch` attribute can apply to a net, cell, port, reference, or design. You can get detailed information about the predefined attributes that apply to each object type by using the commands listed in [Table 6-4](#).

Table 6-4 Commands to Get Attribute Descriptions

Object type	Command
All	<code>man attributes</code>
Designs	<code>man design_attributes</code>
Cells	<code>man cell_attributes</code>
Clocks	<code>man clock_attributes</code>

Table 6-4 Commands to Get Attribute Descriptions (Continued)

Object type	Command
Nets	<code>man net_attributes</code>
Pins	<code>man pin_attributes</code>
Ports	<code>man port_attributes</code>
Libraries	<code>man library_attributes</code>
Library cells	<code>man library_cell_attributes</code>
References	<code>man reference_attributes</code>

Setting Attribute Values

To set the value of an attribute, use one of the following:

- [Using an Attribute-Specific Command](#)
- [Using the `set_attribute` Command](#)

Using an Attribute-Specific Command

Use an attribute-specific command to set the value of the command's associated attribute.

For example,

```
prompt> set_dont_touch U1
```

Using the `set_attribute` Command

Use the `set_attribute` command to set the value of any attribute or to define a new attribute and set its value.

For example, to set the `dont_touch` attribute on the `lsi_10k/FJK3` library cell, enter

```
prompt> set_attribute lsi_10K/FJK3 dont_touch true
```

The `set_attribute` command enforces the predefined attribute type and generates an error if you try to set an attribute with a value of an incorrect type.

For example, the predefined type for the `max_fanout` attribute is float. Therefore, if you enter the following command, Design Compiler returns an error message:

```
prompt> set_attribute lib/lcell/lpin max_fanout 1 -type integer
```

To determine the predefined type for an attribute, use the `list_attributes -application` command. This command generates a list of all application attributes and their types. To generate a smaller report, you can use the `-class` attribute to limit the list to attributes that apply to one of the following classes: design, port, cell, clock, pin, net, lib, or reference.

If an attribute applies to more than one object type, Design Compiler searches the database for the named object. For information about the search order, see [The Object Search Order](#).

When you set an attribute on a reference (subdesign or library cell), the attribute applies to all cells in the design with that reference. When you set an attribute on an instance (cell, net, or pin), the attribute overrides any attribute inherited from the instance's reference.

Viewing Attribute Values

To see all attributes on an object, use the `report_attribute` command:

```
prompt> report_attribute object_list
```

For example,

```
dc_shell> report_attribute myAdd
*****
Report : Attribute
Design : test
...
*****
Design      Object        Type       Attribute Name      Value
-----      -----
test        myAdd        cell       verification_priority    high
test        myAdd        cell       operator_label     myAdd
```

To see the value of a specific attribute on an object, use the `get_attribute` command. For example, to get the value of the maximum fanout on port OUT7, enter

```
prompt> get_attribute OUT7 max_fanout
Performing get_attribute on port 'OUT7'.
{3.000000}
```

When you run Design Compiler NXT and Design Compiler Graphical, you can also query the bounding box information for pins and ports. For example, the following commands report the `bbox` attribute on the Q pin of register reg_1 and the `test_mode` port of the current design:

```
dc_shell-topo> get_attribute [get_pins reg_1/Q] bbox
dc_shell-topo> get_attribute [get_ports test_mode] bbox
```

Note:

The `bbox` attribute is not supported in wire load mode. Querying placed pins and ports in topographical mode requires additional licensing. For details about licensing, contact your Synopsys representative.

If an attribute applies to more than one object type, Design Compiler searches the database for the named object. For information about the search order, see [The Object Search Order](#).

You can use the Properties dialog box in the GUI to view attributes and other object properties for selected designs, design objects, or timing paths. You can also set, change, or remove the attribute values for certain properties.

For more information about using the GUI, see the *Design Vision User Guide* and the Design Vision Help.

Saving Attribute Values

Design Compiler does not automatically save attribute values when you exit dc_shell. Use the `write_script` command to generate a dc_shell script that re-creates the attribute values.

By default, the `write_script` command writes the dc_shell commands to standard output. Use the redirection operator (>) to redirect the output to a file.

```
prompt> write_script > attr.scr
```

The `write_script` command does not support user-defined attributes.

Defining Attributes

To define a new attribute, use the `define_user_attribute` command. To set the attribute to a specified value on a specified list of objects, use the `set_attribute` command.

The following example defines an integer cell attribute named X and then sets this attribute to 30 on all cells in this level of the hierarchy:

```
prompt> define_user_attribute -type int -classes cell X
cell
prompt> set_attribute [get_cells *] X 30
{U1}
```

If you want to change the value of an attribute, remove the attribute and then re-create it to store the desired type.

See Also

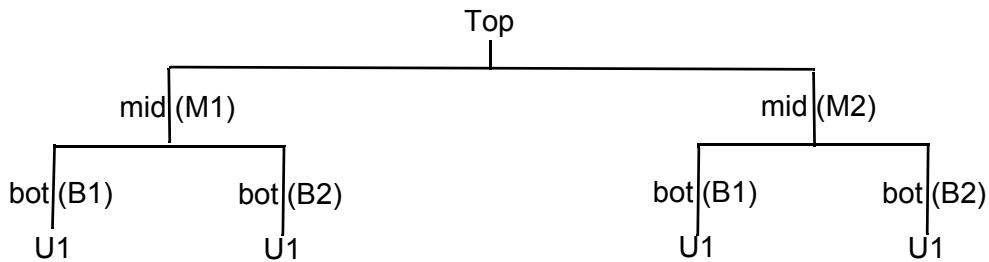
- [Setting Attribute Values](#)

Propagating Attributes

To propagate user-defined attributes set on cell instances of lower-level designs to cell instances in the higher-level hierarchy, use the `propagate_user_attributes` command. For example in [Figure 6-2](#),

- The current design is set to mid, and the user-defined attribute is defined as XYZ with a value of false on all cells in the mid design.
- The user-defined attribute is set on the B1 and B2 instances of the bot design with cell instances U1.

Figure 6-2 Cell Instances U1 in the bot Design



- When the current design (mid) is changed to Top as shown in [Example 6-1](#), use the `propagate_user_attributes` command to propagate all the user-defined attributes set on all the cell instances in the bot design to the Top design.

Example 6-1 Cell Instances U1 in the bot Design Propagate to the Top Design

```

prompt> current_design mid
Current design is 'mid'.
{mid}

prompt> define_user_attribute -type boolean -classes cell XYZ
XYZ

prompt> set_user_attribute [get_cells -hierarchical *] XYZ false
Information: Attribute 'XYZ' is set on 4 objects. (UID-186)
B1/U1 B2/U1 B1 B2

prompt> current_design Top
Current design is 'top'.
{top}

prompt> get_cells -hierarchical -filter "XYZ==false"
{M1/B1 M1/B2 M2/B1 M2/B2}

prompt> propagate_user_attributes {XYZ}
1

prompt> get_cells -hierarchical -filter "XYZ==false"
{M1/B1 M1/B1/U1 M1/B2 M1/B2/U1 M2/B1 M2/B1/U1 M2/B2 M2/B2/U1}
  
```

See Also

- [Setting Attribute Values](#)

Removing Attributes

To remove a specific attribute from an object, use the `remove_attribute` command.

You cannot use the `remove_attribute` command to remove inherited attributes. For example, if a `dont_touch` attribute is assigned to a reference, remove the attribute from the reference, not from the cells that inherited the attribute.

For example, to remove the `max_fanout` attribute from port OUT7, enter

```
prompt> remove_attribute OUT7 max_fanout
```

You can remove selected attributes by using the `remove_*` commands. Note that some attributes still require the `set_*` command with a `-default` option specified to remove the attribute previously set by the command. See the man page for a specific command to determine whether it has the `-default` option or uses a corresponding `remove` command.

To remove all user-defined attributes from the current design, except those defined with the `set_attribute` command, use the `reset_design` command:

```
prompt> reset_design
Resetting current design 'EXAMPLE'.
1
```

The `reset_design` command removes all design information, including clocks, input and output delays, path groups, operating conditions, timing ranges, and wire load models. The result of using `reset_design` is often equivalent to starting the design process from the beginning. To remove attributes defined with `set_attribute`, use the `remove_attribute` command.

The Object Search Order

When Design Compiler searches for an object, the search order is command dependent. Objects include designs, cells, nets, references, and library cells.

If you do not use a `get_*` command, Design Compiler uses an implicit find to locate the object. Commands that can set an attribute on more than one type of object use this search order to determine the object to which the attribute applies.

For example, the `set_dont_touch` command operates on cells, nets, references, and library cells. If you define an object, X, with the `set_dont_touch` command and two objects (such as the design and a cell) are named X, Design Compiler applies the attribute to the first object type found. In this case, the attribute is set on the design, not on the cell.

Design Compiler searches until it finds a matching object, or it displays an error message if it does not find a matching object.

You can override the default search order by using the `get_*` command to specify the object.

For example, assume that the current design contains both a cell and a net named `critical`. The following command sets the `dont_touch` attribute on the cell because of the default search order:

```
prompt> set_dont_touch critical  
1
```

To place the `dont_touch` attribute on the net instead, use the following command:

```
prompt> set_dont_touch [get_nets critical]  
1
```

Creating Designs

To create a new design, use the `create_design` command. The memory file name is `my_design.db`, and the path is the current working directory. For example,

```
prompt> create_design my_design  
1  
prompt> list_designs -show_file  
  
/work_dir/mapped/test.ddc  
test (*) test_DW01_inc_16_0 test_DW02_mult_16_16_1  
  
/work_dir/my_design.db  
my_design  
1
```

Designs created with `create_design` contain no design objects. Use the appropriate create commands, such as `create_clock`, `create_cell`, or `create_port` to add design objects to the new design.

See Also

- [Editing Designs](#)

Copying Designs

To copy a design in memory and rename the copy, use the `copy_design` command. The new design has the same path and memory file as the original design. For example,

```
prompt> copy_design test test_new
Information: Copying design /designs/test.ddc:to designs/
test.ddc:test_new
1
prompt> list_designs -show_file

/designs/test.ddc
test (*) test_new
```

You can use the `copy_design` command with the `change_link` command to manually create unique instances. For example, assume that a design has two identical cells, U1 and U2, both linked to COMP. Enter the following commands to create unique instances:

```
prompt> copy_design COMP COMP1
Information: Copying design /designs/COMP.ddc:COMP to
           designs/COMP.ddc:COMP1
1

prompt> change_link U1 COMP1
Performing change_link on cell 'U1'.
1

prompt> copy_design COMP COMP2
Information: Copying design /designs/COMP.ddc:COMP to
           designs/COMP.ddc:COMP2
1

prompt> change_link U2 COMP2
Performing change_link on cell 'U2'.
1
```

Renaming Designs

To rename a design in memory, use the `rename_design` command. You can assign a new name to a design or move a list of designs to a file. To save a renamed file, use the `write_file` command.

In the following example, the `list_designs` command is used to show the design before and after you use the `rename_design` command:

```
prompt> list_designs -show_file
/designs/test.ddc
test(*) test_new
1

prompt> rename_design test_new test_new_1
Information: Renaming design /designs/test.ddc:test_new to
           /designs/test.ddc:test_new_1
1
```

```
prompt> list_designs -show_file
/designs/test.ddc
test (*) test_new test_new_1
1
```

To rename designs and update cell links for the entire design hierarchy, use the `-prefix`, `-postfix`, and `-update_links` options. For example, the following script prefixes the string `NEW_` to the name of the design `D` and updates links for its instance cells:

```
prompt> get_cells -hierarchical -filter "ref_name == D"
{b_in_a/c_in_b/d1_in_c b_in_a/c_in_b/d2_in_c}

prompt> rename_design D -prefix NEW_ -update_links
Information: Renaming design /test_dir/D.ddc:D to
                  /test_dir/D.ddc:NEW_D. (UIMG-45)

prompt> get_cells -hierarchical -filter "ref_name == D"
# no such cells!

prompt> get_cells -hierarchical -filter "ref_name == NEW_D"
{b_in_a/c_in_b/d1_in_c b_in_a/c_in_b/d2_in_c}
```

Cells `b_in_a/c_in_b/d1_in_c` and `b_in_a/c_in_b/d2_in_c` instantiate design `D`. After you have run the `rename_design D -prefix NEW -update_links` command, the instances are relinked to the renamed reference design `NEW_D`.

Changing the Design Hierarchy

When possible, reflect the design partitioning in your HDL description. If your HDL code is already developed, Design Compiler allows you to change the hierarchy without modifying the HDL description.

To display the design hierarchy, use the `report_hierarchy` command. Use this command to understand the current hierarchy before making changes and to verify the hierarchy changes.

Design Compiler provides the following hierarchy manipulation capabilities:

- [Adding Levels of Hierarchy](#)
- [Removing Levels of Hierarchy](#)
- [Preserving Hierarchical Pin Timing Constraints During Ungrouping](#)
- [Merging Cells From Different Subdesigns](#)

Adding Levels of Hierarchy

Adding a level of hierarchy is called *grouping*. You can create a level of hierarchy by grouping cells or related components into subdesigns.

To group cells (instances) in the design into a new subdesign, creating a new level of hierarchy, use the `group` command. The grouped cells are replaced by a new instance that references the new subdesign.

The ports of the new subdesign are named after the nets to which they are connected in the design. The direction of each port of the new subdesign is determined from the pins of the corresponding net.

When the parent design is unique, the cell list that you specify can include cells from a lower level in the hierarchy; however, these cells should be at the same level of hierarchy in relation to one another.

If you do not specify an instance name, Design Compiler creates one for you. The created instance name has the format `Un`, where *n* is an unused cell number (for example, `U107`).

Note:

Grouping cells might not preserve all the attributes and constraints of the original cells.

To group two cells into a new design named `SAMPLE` with an instance name `U`, enter

```
prompt> group {u1 u2} -design_name SAMPLE -cell_name U
```

To group all cells that begin with `alu` into a new design `uP` with cell name `UCELL`, enter

```
prompt> group "alu*" -design_name uP -cell_name UCELL
```

In the following example, three cells—`bot1`, `cell1`, and `j`—are grouped into a new subdesign named `SAMPLE`, with an instance name `U1`. The cells are at a lower level in the hierarchy and at the same hierarchical level; the parent design is unique.

```
prompt> group {mid1/bot1 mid1/cell1 mid1/j} \
           -cell_name U1 -design_name SAMPLE
```

The preceding command is equivalent to issuing the following two commands:

```
prompt> current_design mid
prompt> group {bot1 cell1 j} -cell_name U1 -design_name SAMPLE
```

To group all cells in the HDL function bar in the `ftj` process into the `new_block` design, enter

```
prompt> group -hdl_block ftj/bar -design_name new_block
```

To group all bussed gates beneath the `ftj` process into separate levels of hierarchy, enter

```
prompt> group -hdl_block ftj -hdl_bussed
```

See Also

- [Keeping Related Combinational Logic Together](#)

Removing Levels of Hierarchy

Removing a level of hierarchy is called *ungrouping*. Ungrouping merges subdesigns of a specific level of the hierarchy into the parent cell or design. You can perform ungrouping manually before optimization, or you can perform ungrouping during optimization, either manually or automatically.

Note:

Designs, subdesigns, and cells that have the `dont_touch` attribute cannot be ungrouped (including automatic ungrouping) before or during optimization.

To learn how to remove levels of hierarchy, see

- [Ungrouping Hierarchies Before Optimization](#)
- [Ungrouping Hierarchies Explicitly During Optimization](#)
- [Ungrouping Hierarchies Automatically During Optimization](#)

Ungrouping Hierarchies Before Optimization

To ungroup one or more designs before optimization, use the `ungroup` command. You can use the `ungroup` command on a hierarchical design from any level in the design without using the `current_design` command to specify the current design.

When the parent design is unique, the cell list can include cells from a lower level in the hierarchy (that is, the `ungroup` command can accept instance objects).

The `ungroup` command can ungroup all cells in the current design or current instance, ungroup each cell recursively until all levels of hierarchy within the current design (instance) are removed, or ungroup cells recursively starting at any hierarchical level.

Note:

If you ungroup cells and then use the `change_names` command to modify the hierarchy separator (/), you might lose attribute and constraint information.

To ungroup a list of cells, enter

```
prompt> ungroup {high_decoder_cell low_decoder_cell}
```

To ungroup the cell U1 and specify the prefix to use when creating new cells, enter

```
prompt> ungroup U1 -prefix "U1:"
```

To completely collapse the hierarchy of the current design, enter

```
prompt> ungroup -all -flatten
```

To recursively ungroup cells belonging to CELL_X, which is three hierarchical levels below the current design, enter

```
prompt> ungroup -start_level 3 CELL_X
```

To recursively ungroup cells that are three hierarchical levels below the current design and belong to cells U1 and U2 (U1 and U2 are child cells of the current design), enter

```
prompt> ungroup -start_level 2 {U1 U2}
```

To recursively ungroup all cells that are three hierarchical levels below the current design, enter

```
prompt> ungroup -start_level 3 -all
```

This example illustrates how the `ungroup` command can accept instance objects (cells at a lower level of hierarchy) when the parent design is unique. In the example, MID1/BOT1 is a unique instantiation of design BOT. The command ungroups the cells MID1/BOT1/CELL1 and MID1/BOT1/CELL2.

```
prompt> ungroup {MID1/BOT1/CELL1 MID1/BOT1/CELL2}
```

The preceding command is equivalent to issuing the following two commands:

```
prompt> current_instance MID1/BOT1
prompt> ungroup {CELL1 CELL2}
```

Ungrouping Hierarchies Explicitly During Optimization

You can control which designs are ungrouped during optimization by using the `set_ungroup` command followed by the `compile_ultra` command or the `compile -ungroup_all` command.

- Use the `set_ungroup` command when you want to specify the cells or designs to be ungrouped. This command assigns the `ungroup` attribute to the specified cells or referenced designs. If you set the attribute on a design, all cells that reference the design are ungrouped.

For example, to ungroup cell U1 during optimization, enter the following commands:

```
prompt> set_ungroup U1
prompt> compile_ultra
```

To see whether an object has the `ungroup` attribute set, use the `get_attribute` command.

```
prompt> get_attribute object ungroup
```

To remove an `ungroup` attribute, use the `remove_attribute` command or set the `ungroup` attribute to `false`.

```
prompt> set_ungroup object false
```

- Use the `-ungroup_all` option with the `compile` command to remove all lower levels of the current design hierarchy (including DesignWare parts). For example, enter

```
prompt> compile -ungroup_all
```

Ungrouping Hierarchies Automatically During Optimization

The `compile_ultra` command automatically ungroups logical hierarchies. Ungrouping merges subdesigns of a given level of the hierarchy into the parent cell or design. It removes hierarchical boundaries and allows Design Compiler to improve timing by reducing the levels of logic and to improve area by sharing logic.

During optimization, Design Compiler performs the following types of automatic grouping:

- Area-based automatic ungrouping

Before initial mapping, the `compile_ultra` command performs area-based automatic ungrouping. The tool estimates the area for unmapped hierarchies and removes small subdesigns; the goal is to improve area and timing quality of results. Because the tool performs automatic ungrouping at an early stage, it has a better optimization context.

Additionally, datapath extraction is enabled across ungrouped hierarchies. These factors improve the timing and area quality of results.

- Delay-based automatic ungrouping

During delay optimization, the `compile_ultra` command performs delay-based automatic ungrouping. It ungroups hierarchies along the critical path and is used essentially for timing optimization.

- QoR-based automatic ungrouping

When you use the `-spg` option with the `compile_ultra` command, Design Compiler NXT and Design Compiler Graphical ungroup additional hierarchies to improve QoR.

See Also

- [Automatic Ungrouping](#)

Provides more information about the `compile_ultra` automatic ungrouping capabilities

Preserving Hierarchical Pin Timing Constraints During Ungrouping

Hierarchical pins are removed when a cell is ungrouped. Depending on whether you are ungrouping a hierarchy before optimization or after optimization, Design Compiler handles timing constraints placed on hierarchical pins in different ways.

When preserving timing constraints, Design Compiler reassigns the timing constraints to appropriate adjacent, persistent pins (that is, pins on the same net that remain after ungrouping). The constraints are moved forward or backward to other pins on the same net. Note that the constraints can be moved backward only if the pin driving the given hierarchical pin drives no other pin. Otherwise the constraints must be moved forward.

If the constraints are moved to a leaf cell, that cell is assigned a `size_only` attribute to preserve the constraints during a compile. Thus, the number of `size_only` cells can increase, which might limit the scope of the optimization process. To counter this effect, when both the forward and backward directions are possible, Design Compiler chooses the direction that helps limit the number of newly assigned `size_only` attributes to leaf cells.

You can disable this behavior by setting the `auto_ungroup_preserve_constraints` variable to `false`. The default is `true`.

When you apply ungrouping to an unmapped design, the constraints on a hierarchical pin are moved to a leaf cell and the `size_only` attribute is assigned. However, the constraints are preserved through the compile process only if there is a one-to-one match between the unmapped cell and a cell from the target library.

Only the timing constraints set with the following commands are preserved:

- `set_false_path`
- `set_multicycle_path`
- `set_min_delay`
- `set_max_delay`
- `set_input_delay`
- `set_output_delay`
- `set_disable_timing`
- `set_case_analysis`
- `create_clock`
- `create_generated_clock`
- `set_propagated_clock`
- `set_clock_latency`

Note:

The `set_rtl_load` constraint is not preserved. Also, only the timing constraints of the current design are preserved. Timing constraints in other designs might be lost as a result of ungrouping hierarchy in the current design.

Merging Cells From Different Subdesigns

To merge cells from different subdesigns into a new subdesign,

1. Group the cells into a new design.
2. Ungroup the new design.

For example, the following command sequence creates a new alu design that contains the cells that initially were in subdesigns `u_add` and `u_mult`.

```
prompt> group {u_add u_mult} -design alu
prompt> current_design alu
prompt> ungroup -all
prompt> current_design top_design
```

Editing Designs

Design Compiler provides commands for incrementally editing a design that is in memory. These commands allow you to change the netlist or edit designs by using `dc_shell` commands instead of an external format.

Table 6-5 Design Editing Tasks and Commands

Object	Task	Command
Cells	Create a cell	<code>create_cell</code>
	Delete a cell	<code>remove_cell</code>
Nets	Create a net	<code>create_net</code>
	Connect a net	<code>connect_net</code>
	Disconnect a net	<code>disconnect_net</code>
	Delete a net	<code>remove_net</code>
Ports	Create a port	<code>create_port</code>
	Delete a port	<code>remove_port</code>
		<code>remove_unconnected_ports</code>
Pins	Connect pins	<code>connect_pin</code>

Table 6-5 Design Editing Tasks and Commands (Continued)

Object	Task	Command
Buses	Create a bus	<code>create_bus</code>
	Delete a bus	<code>remove_bus</code>

For unique designs, these netlist editing commands accept instance objects—that is, cells at a lower level of hierarchy. You can operate on hierarchical designs from any level in the design without using the `current_design` command. For example, you can enter the following command to create a cell called cell1 in the design mid1:

```
prompt> create_cell mid1/cell1 my_lib/AND2
```

When connecting or disconnecting nets, use the `all_connected` command to see the objects that are connected to a net, port, or pin. For example, this sequence of commands replaces the reference for cell U8 with a high-power inverter.

```
prompt> get_pins U8/*
{"U8/A", "U8/Z"}
prompt> all_connected U8/A
{"n66"}
prompt> all_connected U8/Z
{"OUTBUS[10]"}
prompt> remove_cell U8
Removing cell 'U8' in design 'top'.
1
prompt> create_cell U8 IVP
Creating cell 'U8' in design 'top'.
1
prompt> connect_net n66 [get_pins U8/A]
Connecting net 'n66' to pin 'U8/A'.
1
prompt> connect_net OUTBUS[10] [get_pins U8/Z]
Connecting net 'OUTBUS[10]' to pin 'U8/Z'.
1
```

Note:

You can achieve the same result by using the `change_link` command instead of the series of commands listed previously. For example, the following command replaces the reference for cell U8 with a high-power inverter:

```
prompt> change_link U8 IVP
```

Tasks you can perform, include

- Resizing a cell

To return a collection of equivalent library cells for a specific cell or library cell, use the `get_alternative_lib_cells` command. You can then use the collection to replace or

resize the cell. The `size_cell` command allows you to change the drive strength of a leaf cell by linking it to a new library cell that has the required properties.

- Inserting buffers or inverter pairs

To add a buffer at pins or ports, use the `insert_buffer` command. The `-inverter_pair` option allows you specify that a pair of inverting library cells is to be inserted instead of a single non-inverting library cell. To retrieve a collection of all buffers and inverters from the library, you can use the `get_buffers` command.

- Inserting repeaters

To select a driver of a two-pin net and insert a chain of single-fanout buffers in the net driven by this driver, use the `insert_buffer` command with the `-no_of_cells` option.

- Removing buffers

To remove buffers, use the `remove_buffer` command.

Translating Designs From One Technology to Another

To translate a design from one technology to another, use the `translate` command. If you are using Design Compiler in topographical mode, use the `compile_ultra-incremental` command. Designs are translated cell by cell from the original logic library to a new logic library, preserving the gate structure of the original design. The translator uses the functional description of each existing cell (component) to determine the matching component in the new logic library (target library). If no exact replacement exists for a component, it is remapped with components from the target library.

You can influence the replacement-cell selection by preferring or disabling specific library cells (using the `set_prefer` and `set_dont_use` commands) and by specifying the types of registers (using the `set_register_type` command). The target libraries are specified in the `target_library` variable. The `local_link_library` attribute of the top-level design is set to the `target_library` value after the design is linked.

The `translate` command does not operate on cells or designs having the `dont_touch` attribute. After the translation process, Design Compiler reports cells that are not successfully translated.

For details, see

- [Translating Designs in Design Compiler Wire Load Mode](#)
- [Translating Designs in Design Compiler Topographical Mode](#)
- [Restrictions on Translating Between Technologies](#)

Translating Designs in Design Compiler Wire Load Mode

The following procedure works for most designs, but manual intervention might be necessary for some complex designs.

To translate a design,

1. Read in your mapped design.

```
dc_shell> read_file design.ddc
```

2. Set the target library to the new logic library.

```
dc_shell> set target_library target_lib.db
```

3. Invoke the `translate` command.

```
dc_shell> translate
```

After a design is translated, you can compile it to improve the implementation in the new logic library.

Translating Designs in Design Compiler Topographical Mode

To map a design to a new technology in Design Compiler topographical mode,

1. Add the original logic library to the link library:

```
dc_shell-topo> lappend link_library tech_orig.db
```

2. Set up your Design Compiler environment for the new target technology.

- a. Specify the logic libraries.
- b. Specify the physical libraries.

3. Read in your mapped design:

```
dc_shell-topo> read_verilog design.v
```

4. Run the `compile_ultra -incremental` command to translate the design to the new technology.

Restrictions on Translating Between Technologies

Keep the following restrictions in mind when you translate a design from one technology to another:

- The `translate` command translates functionality logically but does not preserve drive strength during translation. It always uses the lowest drive strength version of a cell, which might produce a netlist with violations.
- Buses driven by CMOS three-state components must be fully decoded (Design Compiler can assume that only one bus driver is ever active). If this is the case, bus drivers are translated into control logic. To enable this feature, set the `compile_assume_fully_decoded_three_state_buses` variable to `true` before translating.
- If a three-state bus within a design is connected to one or more output ports, translating the bus to a multiplexed signal changes the port functionality. Because `translate` does not change port functionality, this case is reported as a translation error.

Removing Designs From Memory

To remove designs from dc_shell memory, use the `remove_design` command. For example, after completing a compilation session and saving the optimized design, you can use `remove_design` to delete the design from memory before reading in another design.

By default, the `remove_design` command removes only the specified design. To remove its subdesigns, specify the `-hierarchy` option. To remove all designs and libraries from memory, specify the `-all` option.

If you defined variables that reference design objects, Design Compiler removes these references when you remove the design from memory. This prevents future commands from attempting to operate on nonexistent design objects. For example,

```
prompt> set PORTS [all_inputs]
{"A0", "A1", "A2", "A3"}
prompt> query_objects $PORTS
PORTS = {"A0", "A1", "A2", "A3"}
prompt> remove_design
Removing design 'top'
1
prompt> query_objects $PORTS
Error: No such collection '_sel2' (SEL-001)
```

Saving Designs

You can save (write to disk) the designs and subdesigns of the design hierarchy at any time, using different names or formats. After a design is modified, you should manually save it. Design Compiler does not automatically save designs before it exits.

For information about supported output formats, using the GUI, and ensuring naming consistency, see

- [Supported Design File Output Formats](#)
- [Writing a Design Netlist or Schematic](#)
- [Writing To a Milkyway Database](#)
- [Saving Designs Using GUI Commands](#)
- [Ensuring Name Consistency Between the Design Database and the Netlist](#)

Supported Design File Output Formats

Design Compiler supports the design file formats listed in [Table 6-6](#).

Table 6-6 Supported Design File Output Formats

Format	Description
.ddc	Synopsys internal database format
Verilog	IEEE Standard Verilog (see the HDL Compiler documentation)
svsim	SystemVerilog netlist wrapper Note: The <code>write_file -format svsim</code> command writes out only the netlist wrapper, not the gate-level DUT itself. To write out the gate-level DUT, you must use the existing <code>write_file -format verilog</code> command. For details, see the <i>HDL Compiler for SystemVerilog User Guide</i> .
VHDL	IEEE Standard VHDL (see the HDL Compiler documentation)
Milkyway	Format for writing a Milkyway database within Design Compiler

In topographical mode,

- The .ddc format contains back-annotated net delays and constraints. Subsequent topographical mode sessions restore virtual layout data. The .ddc format is recommended for subsequent topographical mode optimizations and verification.

- The Milkyway format contains back-annotated net delays and constraints. The Milkyway format cannot be read back into topographical mode for subsequent optimizations. This format is recommended if you intend to use Synopsys tools for the back-end flow.
- The ASCII format, either in a Verilog or a VHDL netlist, does not contain back-annotated delays or Synopsys design constraints (SDC constraints). Use the `write_sdc` command to write out the SDC constraints and the `write_parasitics` command to write out parasitics. This format is recommended only if you intend to use a third-party tool.

Writing a Design Netlist or Schematic

Design Compiler does not automatically save the designs loaded in memory. To write a design netlist or schematic to a file before exiting, use the `write_file` command:

```
prompt> write_file -format ddc -hierarchy -output my_design.ddc
```

By default, the `write_file` command saves just the top-level design. To save the entire design, specify the `-hierarchy` option.

If you do not use the `-output` option with the `write_file` command to specify the output file name, the tool creates a file called *top_design.ddc*, where *top_design* is the name of the current design.

Writing To a Milkyway Database

To write to a Milkyway database, use the `write_milkyway` command. The `write_milkyway` command creates a design file based on the netlist in memory and saves the design data for the current design in that file.

Note:

You cannot use the Milkyway format to store design data for unmapped designs or non-uniquified designs. Before you use the `write_milkyway` command, run the following command:

```
prompt> uniquify -force -dont_skip_empty_designs
```

See Also

- [Using a Milkyway Database](#)

Saving Designs Using GUI Commands

You can also use the GUI to save the current design and each of its subdesigns. To learn how to save designs from the GUI, see the *Design Vision User Guide*.

Ensuring Name Consistency Between the Design Database and the Netlist

Before writing a netlist from within dc_shell, make sure that all net and port names conform to the naming conventions for your layout tool. Also ensure that you are using a consistent bus naming style.

Some ASIC and EDA vendors have a program that creates a .synopsys_dc.setup file that includes the appropriate commands to convert names to their conventions. If you need to change any net or port names, use the `define_name_rules` and `change_names` commands. For more information, see [Naming Rules Section of the .synopsys_dc.setup File](#).

To define the name mapping and replacement rules and resolve naming problems, see

- [Specifying the Name Mapping and Replacement Rules](#)
- [Resolving Naming Problems in the Flow](#)
- [Avoiding Bit-Blasted Ports in SystemVerilog and VHDL Structures](#)

Specifying the Name Mapping and Replacement Rules

To define the name mapping and replacement rules to avoid an error in the format of the string, use the `-map` option with the `define_name_rules` command, as shown in [Example 6-2](#). If you do not follow this convention, an error appears.

Example 6-2 Using define_name_rules -map

```
define_name_rules naming_convention  
-map {{string1, string2}} } -type cell
```

For example, to remove trailing underscores from cell names, enter

```
prompt> define_name_rules naming_convention \  
          -map {{$_, ""}} } -type cell
```

Resolving Naming Problems in the Flow

You might encounter conflicts in naming conventions in design objects, input and output files, and tool sets. In the design database file, you can have many design objects (such as ports, nets, cells, logic modules, and logic module pins), all with their own naming conventions. Furthermore, you might be using several input and output file formats in your flow. Each file format is different and has its own syntax definitions. Using tool sets from several vendors can introduce additional naming problems.

Correct naming eliminates name escaping and mismatch errors in your design. To resolve naming issues, use the `change_names` command to make sure that all the file names match,

and make the name changes in the design database file before you write any files. Follow this flow:

1. Read in your design RTL, and apply constraints.
No changes to your method need to be made here.
2. Compile the design to produce a gate-level description.
Compile or reoptimize your design as you normally would, using your standard set of scripts.
3. Apply name changes and resolve naming issues. Use the `change_names` command and its Verilog or VHDL option before you write the design.

Important:

Always use the `change_names -rules [verilog|vhdl] -hierarchy` command whenever you want to write out a Verilog or VHDL design because naming in the design database file is not Verilog or VHDL compliant. For example, enter

```
prompt> change_names -rules verilog -hierarchy
```

4. Write the files to disk. Use the `write_file -format verilog` command.
Look for reported name changes, which indicate you need to repeat step 3 and refine your name rules.
5. If all the appropriate name changes have been made, your output file should match the design database file. Enter the following commands and compare the output.

```
prompt> write_file -format verilog -hierarchy -output "consistent.v"
prompt> write_file -format ddc -hierarchy -output "consistent.ddc"
```

6. Write the files for third-party tools.

If you need more specific naming control, use the `define_name_rules` command.

See Also

- [Specifying the Name Mapping and Replacement Rules](#)

Avoiding Bit-Blasted Ports in SystemVerilog and VHDL Structures

You can use the `change_names` command to avoid bit-blasted ports in SystemVerilog structs and VHDL records (packed or unpacked). To enable this capability, specify the `-preserve_struct_ports` option with the `define_name_rules` command. Add this option to the existing Verilog naming rules as follows:

```
prompt> define_name_rules verilog -preserve_struct_ports
prompt> change_names -hierarchy -rules verilog
```

Note that using either of the following options of the `define_name_rules` command overrides the `-preserve_struct_ports` option:

- `-remove_port_bus`
- `-dont_change_bus_members`

As shown in the following examples, the Verilog netlist 2 preserves the struct ports using the naming rules specified by the `define_name_rules` and `change_names` commands:

- Original RTL

```
typedef struct {logic blue; logic red;} color;
module top (
    input color i_color,
    output color o_color
);
assign o_color = i_color;
endmodule
```

- Netlist 1 with bit-blasted struct ports

```
module top ( i_color_blue_, i_color_red_, o_color_blue_, o_color_red_ );
    input i_color_blue_, i_color_red_;
    output o_color_blue_, o_color_red_;
    wire o_color_blue_, o_color_red_;
    assign o_color_blue_ = i_color_blue_;
    assign o_color_red_ = i_color_red_;
endmodule
```

- Netlist 2 with preserved struct ports

```
module top ( i_color, o_color );
    input [1:0] i_color;
    output [1:0] o_color;
    assign o_color[1] = i_color[1];
    assign o_color[0] = i_color[0];
endmodule
```

Summary of Commands for Changing Names

[Table 6-7](#) summarizes commands for changing names.

Table 6-7 Summary of Commands for Changing Names

To do this	Use this
Change the names of ports, cells, and nets in a design to be Verilog or VHDL compliant.	<code>change_names</code>
Show effects of <code>change_names</code> without making the changes.	<code>report_names</code>

Table 6-7 Summary of Commands for Changing Names (Continued)

To do this	Use this
Define a set of rules for naming design objects. Name rules are used by <code>change_names</code> and <code>report_names</code> .	<code>define_name_rules</code>
List available name rules.	<code>report_name_rules</code>

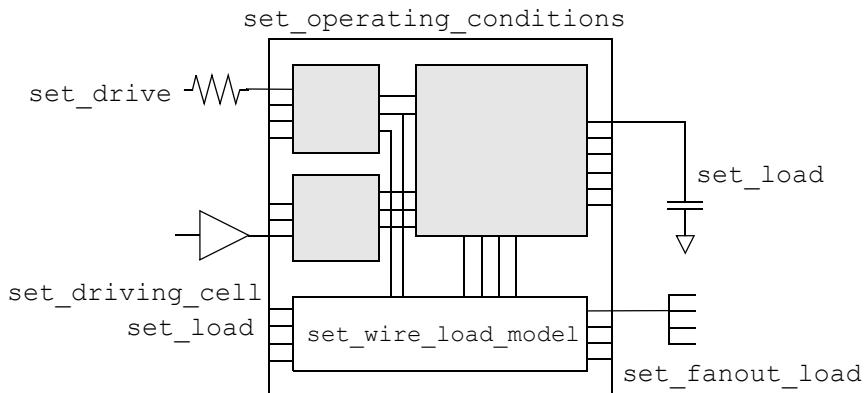
7

Defining the Design Environment

Before a design can be optimized, you must define the environment in which the design is expected to operate. You define the environment by specifying operating conditions, system interface characteristics, and wire load models (used only when Design Compiler is not operating in topographical mode). Operating conditions include temperature, voltage, and process variations. System interface characteristics include input drivers, input and output loads, and fanout loads. The environment model directly affects design synthesis results.

In Design Compiler, the model is defined by a set of attributes and constraints that you assign to the design using specific dc_shell commands. [Figure 7-1](#) illustrates commands used to define the design environment.

Figure 7-1 Commands Used to Define the Design Environment



To learn how to define the design environment, see

- [Operating Conditions](#)
- [Defining Operating Conditions](#)
- [Reporting Operating Conditions](#)
- [Modeling the System Interface](#)
- [Setting Logic Constraints on Ports](#)
- [Wire Load Models](#)
- [Specifying Wire Load Models and Modes](#)
- [Defining the Environment Using Topographical Mode](#)
- [Leakage Power and Dynamic Power Optimization](#)
- [Comparing Design Compiler Topographical and IC Compiler Environments](#)
- [Comparing Design Settings Between Design Compiler and IC Compiler II](#)

Operating Conditions

In most technologies, variations in operating temperature, supply voltage, and manufacturing process can strongly affect circuit performance (speed). These factors, called operating conditions, have the following general characteristics:

- Operating temperature variation

Temperature variation is unavoidable in the everyday operation of a design. Effects on performance caused by temperature fluctuations are most often handled as linear scaling effects, but some submicron silicon processes require nonlinear calculations.

- Supply voltage variation

The design's supply voltage can vary from the established ideal value during day-to-day operation. Often a complex calculation (using a shift in threshold voltages) is employed, but a simple linear scaling factor is also used for logic-level performance calculations.

- Process variation

This variation accounts for deviations in the semiconductor fabrication process. Usually process variation is treated as a percentage variation in the performance calculation.

When performing timing analysis, Design Compiler must consider the worst-case and best-case scenarios for the expected variations in process, temperature, and voltage.

See Also

- [Defining Operating Conditions](#)
- [Reporting Operating Conditions](#)

Defining Operating Conditions

Most logic libraries have predefined sets of operating conditions. Before you set the operating conditions, you can

- List the operating conditions defined in a logic library by using the `report_lib` command.

The library must be loaded in memory before you run the `report_lib` command. To see which libraries are loaded in memory, use the `list_libs` command.

The following example generates a report for the `my_lib` library that is stored in `my_lib.db`:

```
prompt> read_file my_lib.db
prompt> report_lib my_lib
```

- List the operating conditions defined for the current design by using the `current_design` command.

If the logic library contains operating condition specifications, you can use them as the default conditions. To specify explicit operating conditions that supersede the default library conditions, use the `set_operating_conditions` command.

The following example sets the operating conditions for the current design to worst-case commercial:

```
prompt> set_operating_conditions WCCOM -library my_lib
```

Reporting Operating Conditions

To see the operating conditions that are defined for the current design, use the `report_design` command.

[Example 7-1](#) shows an operating conditions report.

Example 7-1 Operating Conditions Report

```
*****
Report : library
Library: my_lib
...
*****
...
Operating Conditions:

Name      Library      Process      Temp      Volt      Interconnect Model
-----
WCCOM     my_lib       1.50        70.00     4.75      worst_case_tree
WCIND     my_lib       1.50        85.00     4.75      worst_case_tree
WCMIL     my_lib       1.50       125.00     4.50      worst_case_tree
...
...
```

Modeling the System Interface

To model the design's interaction with the external system, perform the following tasks:

- [Defining Drive Characteristics for Input Ports](#)
- [Defining Loads on Input and Output Ports](#)
- [Defining Fanout Loads on Output Ports](#)

Defining Drive Characteristics for Input Ports

To determine the delay and transition time characteristics of incoming signals, Design Compiler needs information about the external drive strength and the loading at each input port. Drive strength is the reciprocal of the output drive resistance, and the transition delay at an input port is the product of the drive resistance and the capacitance load of the input port. Design Compiler uses drive strength information to buffer nets appropriately in the case of a weak driver.

By default, Design Compiler assumes zero drive resistance on input ports, meaning infinite drive strength. To set a realistic drive strength, use one of the following commands:

- `set_driving_cell`

Use the `set_driving_cell` command to specify drive characteristics on ports that are driven by cells in the logic library. This command is compatible with all the delay models, including the nonlinear delay model and piecewise linear delay model. The `set_driving_cell` command associates a library pin with an input port so that delay calculators can accurately model the drive capability of an external driver.

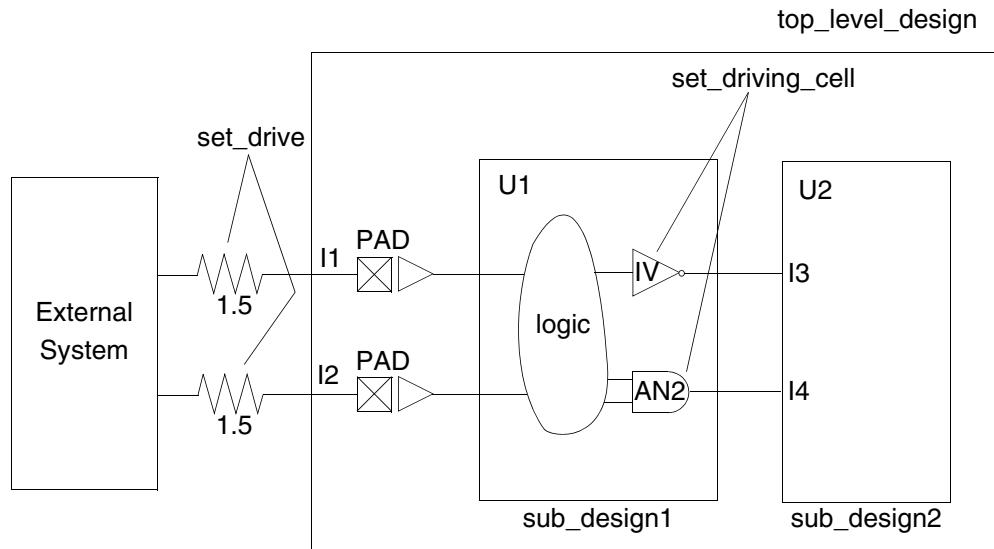
- `set_drive`
- `set_input_transition`

Both the `set_driving_cell` and `set_input_transition` commands affect the port transition delay, but they do not place design rule requirements, such as `max_fanout` and `max_transition`, on input ports. However, the `set_driving_cell` command does place design rules on input ports if the driving cell has design rule constraints.

Both the `set_drive` and the `set_driving_cell` commands affect the port transition delay. The `set_driving_cell` command can place design rule requirements, such as the `max_fanout` or `max_transition` attributes, on input ports if the specified cell has input ports. The most recently used command takes precedence. For example, setting a drive resistance on a port with the `set_drive` command overrides previously run `set_driving_cell` commands.

Use the `set_drive` or `set_input_transition` command to set the drive resistance on the top-level ports of the design when the input port drive capability cannot be characterized with a cell in the logic library.

[Figure 7-2](#) shows a hierarchical design. The top-level design has two subdesigns, U1 and U2. The I1 and I2 ports of the top-level design are driven by the external system and have a drive resistance of 1.5.

Figure 7-2 Drive Characteristics

To set the drive characteristics for this example, follow these steps:

1. Because ports I1 and I2 are not driven by library cells, use the `set_drive` command to define the drive resistance:

```
prompt> current_design top_level_design
prompt> set_drive 1.5 {I1 I2}
```

2. To describe the drive capability for the ports on design sub_design2, change the current design to sub_design2:

```
prompt> current_design sub_design2
```

3. An IV cell drives port I3. Use the `set_driving_cell` command to define the drive resistance. Because IV has only one output and one input, define the drive capability as follows:

```
prompt> set_driving_cell -lib_cell IV {I3}
```

4. An AN2 cell drives port I4. Because the different arcs of this cell have different transition times, select the worst-case arc to define the drive. For checking setup violations, the worst-case arc is the slowest arc. For checking hold violations, the worst-case arc is the fastest arc.

For this example, assume that you want to check for setup violations. The slowest arc on the AN2 cell is the B-to-Z arc, so define the drive as follows:

```
prompt> set_driving_cell -lib_cell AN2 -pin Z -from_pin B {I4}
```

Note:

For heavily loaded driving ports, such as clock lines, keep the drive strength setting at 0 so that Design Compiler does not buffer the net. Each semiconductor vendor has a different way of distributing these signals within the silicon.

To remove driving cell attributes on ports, use the `remove_driving_cell` or `reset_design` command.

Defining Loads on Input and Output Ports

By default, Design Compiler assumes zero capacitive load on input and output ports. To set a capacitive load value on input and output ports of the design, use the `set_load` command. This information helps Design Compiler select the appropriate cell drive strength of an output pad and helps model the transition delay on input pads.

The following example sets a load of 30 on output pin out1:

```
prompt> set_load 30 {out1}
```

You should specify the load value units consistent with the target logic library. For example, if the library represents the load value in picofarads, the value you set with the `set_load` command must be in picofarads.

Use the `report_lib` command to list the library units, as shown in [Example 7-2](#).

Example 7-2 Library Units Report

```
*****
Report : library
Library: my_lib
...
*****
Library Type      : Technology
Tool Created      : 1999.05
Date Created      : February 7, 1992
Library Version   : 1.800000
Time Unit         : 1ns
Capacitive Load Unit : 0.100000ff
Pulling Resistance Unit : 1kilo-ohm
Voltage Unit      : 1V
Current Unit       : 1uA
...
```

Defining Fanout Loads on Output Ports

You can model the external fanout effects by specifying the expected fanout load values on output ports with the `set_fanout_load` command:

```
prompt> set_fanout_load 4 {out1}
```

Design Compiler tries to ensure that the sum of the fanout load on the output port plus the fanout load of cells connected to the output port driver is less than the maximum fanout limit of the library, library cell, and design.

Fanout load is not the same as load. Fanout load is a unitless value that represents a numerical contribution to the total fanout. Load is a capacitance value. Design Compiler uses fanout load primarily to measure the fanout presented by each input pin. An input pin normally has a fanout load of 1, but it can have a higher value.

See Also

- [Defining Maximum Fanout](#)

Setting Logic Constraints on Ports

To improve optimization results, you can set logic constraints on ports to eliminate redundant ports or inverters by using the following methods:

- Setting logic equivalence

Some input ports are driven by logically related signals. For example, the signals driving a pair of input ports might always be the same (logically equal) or might always be different (logically opposite). To specify that two input ports are logically equivalent or logically opposite, use the `set_equal` or `set_opposite` command, respectively.

The following example specifies that the IN_X and IN_Y ports are logically equal:

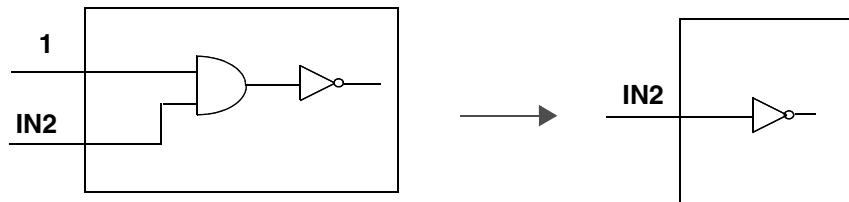
```
prompt> set_equal IN_X IN_Y
```

- Assigning constant values to input ports

Setting a constant value to an input allows Design Compiler to simplify the surrounding logic function during optimization and create a smaller design. To assign a don't care, logic 1, or logic 0 value to inputs in the current design, use the `set_logic_dc`, `set_logic_one`, or `set_logic_zero` command, respectively.

[Figure 7-3](#) shows a simplified input port logic.

Figure 7-3 Simplified Input Port Logic



The following example sets the A and B inputs to don't care and the IN input to logic 1:

```
prompt> set_logic_dc {A B}
prompt> set_logic_one IN
```

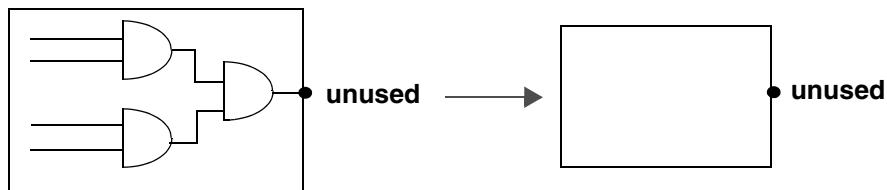
To reset the values set by these commands, use the `remove_attribute` command.

For more information, see [Allowing Assignment of Any Signal to an Input](#) and [Specifying Input Ports as Always One or Zero](#).

- Specifying unconnected output ports

If an output port is not used, that is, if it is unconnected, the logic driving the port can be minimized or eliminated during optimization, as shown in [Figure 7-4](#).

Figure 7-4 Minimizing Logic Driving an Unconnected Output Port



To specify output ports to be unconnected to outside logic, use the `set_unconnected` command:

```
prompt> set_unconnected OUT
```

To undo this command, use the `remove_attribute` command.

Allowing Assignment of Any Signal to an Input

The `set_logic_dc` command specifies an input port driven by a `dont_care` to create smaller designs during compile. After optimization, a port connected to a `dont_care` usually does not drive anything inside the optimized design.

Use `set_logic_dc` to allow assignment of any signal to that input, including but not limited to 0 and 1 during compilation. The outputs of the design are significant only when the inputs that are not `dont_care` completely determine all the outputs, independent of the `dont_care` inputs.

Use the `set_logic_dc` command on input ports:

```
prompt> set_logic_dc { A B }
```

For a 2:1 multiplexer design with inputs S, A, B, and output Z, the function is computed as

$$Z = S * A + S' * B$$

The command `set_logic_dc B` implies that the value of Z is significant when S = 1 and is a don't care when S = 0. The resulting simplification, done during compilation, gives the reduced logic as a wire and the function is

```
Z = A
```

To undo this command, use the `remove_attribute` command.

To specify output ports as unused, use the `set_unconnected` command.

Specifying Input Ports as Always One or Zero

If an input port is always logic-high or -low, Design Compiler might be able to simplify the surrounding logic function during optimization and create a smaller design.

You can specify that input ports are connected to logic 1 or logic 0:

- Tying input ports to logic 1

The `set_logic_one` command lists the input ports tied to logic 1. After optimization, a port connected to logic 1 usually does not drive anything inside the optimized design.

Use the `set_logic_one` command on input ports. To specify output ports as unused, use the `set_unconnected` command.

To undo this command, use the `remove_attribute` command.

- Tying input ports to logic 0

The `set_logic_zero` command lists input ports tied to logic 0. After optimization, a port connected to logic 0 usually does not drive anything inside the optimized design.

Use the `set_logic_zero` command on input ports:

```
prompt> set_logic_one IN
```

To specify output ports as unused, use the `set_unconnected` command.

Wire Load Models

Wire load models are used only when Design Compiler is not operating in topographical mode. Wire load models estimate the effect of wire length and fanout on the resistance, capacitance, and area of nets. Design Compiler uses these physical values to calculate wire delays and circuit speeds.

Semiconductor vendors develop wire load models, based on statistical information specific to the vendors' process. The models include coefficients for area, capacitance, and resistance per unit length, and a fanout-to-length table for estimating net lengths (the number of fanouts determines a nominal length).

In the absence of back-annotated wire delays, Design Compiler uses the wire load models to estimate net wire lengths and delays. Design Compiler determines which wire load model to use for a design based on the following factors, listed in order of precedence:

1. Explicit user specification
2. Automatic selection based on design area
3. Default specification in the logic library

If none of this information exists, Design Compiler does not use a wire load model. Without a wire load model, Design Compiler does not have complete information about the behavior of your target technology and cannot compute loading or propagation times for your nets; therefore, your timing information will be optimistic.

In hierarchical designs, Design Compiler must also determine which wire load model to use for nets that cross hierarchical boundaries. The tool determines the wire load model for cross-hierarchy nets based on one of the following factors, listed in order of precedence:

1. Explicit user specification
2. Default specification in the logic library
3. Default mode in Design Compiler

For information about selecting wire load models for nets and designs, see

- [Hierarchical Wire Load Models](#)
- [Determining Available Wire Load Models](#)
- [Specifying Wire Load Models and Modes](#)

Hierarchical Wire Load Models

Design Compiler supports three modes for determining which wire load model to use for nets that cross hierarchical boundaries:

- Top

Design Compiler models nets as if the design has no hierarchy and uses the wire load model specified for the top level of the design hierarchy for all nets in a design and its subdesigns. The tool ignores any wire load models set on subdesigns with the `set_wire_load_model` command.

Use top mode if you plan to flatten the design at a higher level of hierarchy before layout.

- Enclosed

Design Compiler uses the wire load model of the smallest design that fully encloses the net. If the design enclosing the net has no wire load model, the tool traverses the design hierarchy upward until it finds a wire load model. Enclosed mode is more accurate than top mode when cells in the same design are placed in a contiguous region during layout.

Use enclosed mode if the design has similar logical and physical hierarchies.

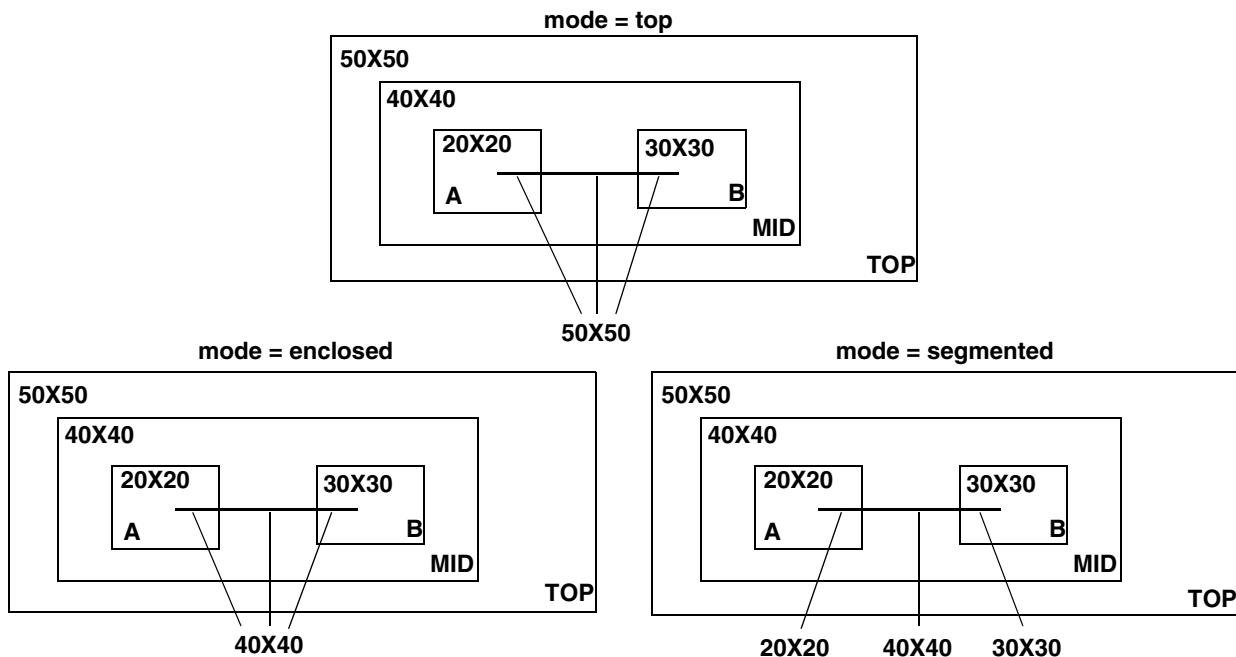
- Segmented

Design Compiler determines the wire load model of each segment of a net by the design encompassing the segment. Nets crossing hierarchical boundaries are divided into segments. For each net segment, Design Compiler uses the wire load model of the design containing the segment. If the design contains a segment that has no wire load model, the tool traverses the design hierarchy upward until it finds a wire load model.

Use segmented mode if the wire load models in your technology have been characterized with net segments.

[Figure 7-5](#) shows a sample design with a cross-hierarchy net, `cross_net`. The top level of the hierarchy (design TOP) has a wire load model of 50x50. The next level of hierarchy (design MID) has a wire load model of 40x40. The leaf-level designs, A and B, have wire load models of 20x20 and 30x30, respectively.

Figure 7-5 Comparison of Wire Load Mode



In top mode, Design Compiler estimates the wire length of net cross_net, using the 50x50 wire load model. Design Compiler ignores the wire load models on designs MID, A, and B.

In enclosed mode, Design Compiler estimates the wire length of net cross_net, using the 40x40 wire load model (the net cross_net is completely enclosed by design MID).

In segmented mode, Design Compiler uses the 20x20 wire load model for the net segment enclosed in design A, the 30x30 wire load model for the net segment enclosed in design B, and the 40x40 wire load model for the segment enclosed in design MID.

Determining Available Wire Load Models

Most logic libraries have predefined wire load models. To list the wire load models defined in a logic library, use the `report_lib` command. The library must be loaded in memory before you run the `report_lib` command. To see a list of libraries loaded in memory, use the `list_libs` command.

The wire load report contains the following sections:

- Wire Loading Model section
 - This section lists the available wire load models.
- Wire Loading Model Mode section
 - This section identifies the default wire load mode. If a library default does not exist, Design Compiler uses top mode.
- Wire Loading Model Selection Group section
 - The presence of this section indicates that the library supports automatic area-based wire load model selection.

To generate a wire load report for the `my_lib` library, enter

```
dc_shell> read_file my_lib.db
dc_shell> report_lib my_lib
```

Example 7-3 shows the resulting wire load models report. The library `my_lib` contains three wire load models: 05x05, 10x10, and 20x20. The library does not specify a default wire load mode (so Design Compiler uses top as the default wire load mode), and it supports automatic area-based wire load model selection.

Example 7-3 Wire Load Models Report

```
*****
Report : library
Library: my_lib
...
*****
```

Wire Loading Model:

```

Name      : 05x05
Location  : my_lib
Resistance : 0
Capacitance : 1
Area      : 0
Slope     : 0.186
Fanout    Length   Points Average Cap Std Deviation
-----
1        0.39

Name      : 10x10
Location  : my_lib
Resistance : 0
Capacitance : 1
Area      : 0
Slope     : 0.311
Fanout    Length   Points Average Cap Std Deviation
-----
1        0.53

Name      : 20x20
Location  : my_lib
Resistance : 0
Capacitance : 1
Area      : 0
Slope     : 0.547
Fanout    Length   Points Average Cap Std Deviation
-----
1        0.86

```

Wire Loading Model Selection Group:

```

Name      : my_lib

      Selection           Wire load name
min area   max area
-----
0.00      1000.00       05x05
1000.00    2000.00       10x10
2000.00    3000.00       20x20
...

```

Specifying Wire Load Models and Modes

The logic library can define a default wire load model that is used for all designs implemented in that technology. The `default_wire_load` library attribute identifies the default wire load model for a logic library.

Some libraries support automatic area-based wire load selection. Design Compiler uses the library function `wire_load_selection` to choose a wire load model based on the total cell area. The wire load model selected the first time you compile is used in subsequent compiles.

For large designs with many levels of hierarchy, automatic wire load selection can increase runtime. To manage runtime, set the wire load manually.

You can turn off automatic selection of the wire load model by setting the `auto_wire_load_selection` variable to `false`:

```
dc_shell> set auto_wire_load_selection false
```

The logic library can also define a default wire load mode. The `default_wire_load_mode` library attribute identifies the default mode. If the current library does not define a default mode, Design Compiler looks for the attribute in the libraries specified in the `link_library` variable. In the absence of a library default and an explicit specification, Design Compiler uses the top mode, which sets the wire load model to the top-level design.

To change the wire load model or mode specified in a logic library, use the `set_wire_load_model` and `set_wire_load_mode` commands. The wire load model and mode you define override all defaults. Explicitly selecting a wire load model also disables area-based wire load model selection for that design.

The following example selects the 10x10 wire load model:

```
dc_shell> set_wire_load_model "10x10"
```

The following example selects the 10x10 wire load model and specifies enclosed mode:

```
dc_shell> set_wire_load_mode enclosed
```

The wire load model you choose for a design depends on how that design is implemented in the chip. Consult your semiconductor vendor to determine the best wire load model for your design.

Use the `report_design` command or the `report_timing` command to see the wire load model and mode defined for the current design.

To remove the wire load model, use the `remove_wire_load_model` command with no model name.

Defining the Environment Using Topographical Mode

In topographical mode, Design Compiler derives the virtual layout of the design so that the tool can accurately predict and use real net capacitances instead of statistical net approximations based on wire load models. In ultra deep submicron designs, interconnect parasitics have a major effect on path delays; accurate estimates of resistance and capacitance are necessary to calculate path delays.

Topographical mode supports

- [General Gate-Level Power Optimization](#)
- [Power Correlation](#)
- [Multivoltage Designs](#)
- [Low Power Intent](#)
- [Multicorner-Multimode Designs](#)

Note:

Do not specify wire load models. If you do, Design Compiler ignores them.

Sometimes Design Compiler in topographical mode and the IC Compiler or IC Compiler II tool have different environment settings. These differences can lead to correlation problems. For more information, see

- [Comparing Design Compiler Topographical and IC Compiler Environments](#)
 - [Comparing Design Settings Between Design Compiler and IC Compiler II](#)
-

General Gate-Level Power Optimization

To perform power optimization, Design Compiler reduces power consumption on paths with positive timing slack. The more paths in the design with positive slack, the more opportunity for Design Compiler to reduce power consumption by using low-power cells. Designs with excessively restrictive timing constraints have little or no positive slack to trade for power reductions.

Designs that have black box cells, such as RAM and ROM, and customized subdesigns that have the `dont_touch` attribute, benefit from power optimization.

In Design Compiler topographical mode, to set the positive timing slack limit, use the `physopt_power_critical_range` variable. In the following example, Design Compiler optimizes only the timing paths with positive slack of 0.2 or more:

```
dc_shell-topo> set_app_var physopt_power_critical_range 0.2
```

Power Correlation

Design Compiler in topographical mode can perform power correlation by estimating the clock tree power. In addition, you can use the `set_power_prediction` command with the `-ct_references` option to specify clock tree references to improve correlation. The `set_power_prediction` command enables the tool to correlate post-synthesis power numbers with those after clock tree synthesis.

To perform power correlation in topographical mode, run a script similar to the following after you set up your design environment and apply synthesis constraints:

```
read_verilog  
set_power_prediction true  
compile_ultra -gate_clock -scan  
report_power  
write_file -format ddc -output synthesized.ddc
```

The `report_power` command splits the power numbers into two categories: netlist power (based on cells already in the design) and estimated power (an estimate of the clock tree power).

See Also

- The *Power Compiler User Guide*
-

Multivoltage Designs

Design Compiler in topographical mode supports power optimization and power correlation for single voltage and multivoltage designs. For multivoltage designs, the subdesign instances (blocks) operate at different voltages. To reduce power consumption, multivoltage designs typically make use of power domains. The blocks of a power domain can be powered up and down, independent of the power state of other power domains (except where a relative always-on relationship exists between two power domains). In particular, power domains can be defined and level shifter and isolation cells can be used as needed to adjust voltage differences between power domains and to isolate shut-down power domains.

A power domain is defined as a grouping of one or more hierarchical blocks in a design that share the following:

- Primary voltage states or voltage range (that is, the same operating voltage)
- Power net hookup requirements
- Power-down control and acknowledge signals (if any)
- Power switching style

- Same process, voltage, and temperature (PVT) operating condition values (all cells of the power domain except level shifters)
- Same set or subset of nonlinear delay model (NLDM) target libraries

Using IEEE™ 1801 Unified Power Format (UPF) Standard commands, you can specify the power domains of the design and the desired multivoltage implementation and verification strategies for each domain, including the domain's isolation cells, retention cells, and level shifters. During a compile operation, Design Compiler automatically inserts these power management cells according to the specified strategies. The same set of UPF commands can be used throughout the design, analysis, verification, and implementation flow.

Power domains are not voltage areas. A power domain is a grouping of logic hierarchies, whereas the corresponding voltage area is a physical placement area into which the cells of the power domain's hierarchies are placed. This correspondence is not automatic. You are responsible for correctly aligning the hierarchies to the voltage areas. You use the `create_voltage_area` command to set voltage areas.

Note:

Because multivoltage designs use power domains, these designs usually require that certain library cells are marked as always-on cells and certain library cell pins are marked as always-on library cell pins. These always-on attributes are necessary to establish any always-on relationships between power domains. For more information, see the *Power Compiler User Guide*.

See Also

- The *Power Compiler User Guide*
Provides descriptions of the principal power domain commands

Low Power Intent

The IEEE 1801, also known as Unified Power Format (UPF), standard establishes a set of commands used to specify the low-power design intent for electronic systems. Using UPF commands, you can specify the power domains of the design and the desired multivoltage implementation and verification strategies for each domain, including the domain's isolation cells, retention cells, and level shifters. During a compile operation, Design Compiler automatically inserts these power management cells according to the specified strategies. The same set of UPF commands can be used throughout the design, analysis, verification, and implementation flow. A Power Compiler license is required for using UPF commands in Design Compiler.

See Also

- [Power Optimization in Multicorner-Multimode Designs](#)

- The *Power Compiler User Guide*

Provides information about using UPF commands and details about the UPF flow for multicorner-multimode design optimization.

Multicorner-Multimode Designs

Designs are often required to operate under multiple modes, such as test or standby mode, and multiple operating conditions, sometimes referred to as corners. Such designs are referred to as multicorner-multimode designs. Design Compiler NXT and Design Compiler Graphical can analyze and optimize designs across multiple modes and corners concurrently. The multicorner-multimode feature in Design Compiler NXT and Design Compiler Graphical provides compatibility between flows in Design Compiler and IC Compiler.

You define modes and corners by using the `create_scenario` command. A scenario definition includes commands that specify the TLUPlus libraries, operating conditions, and constraints. You can also include other commands. For example, you can set leakage power optimization constraints on specific scenarios by using the `set_scenario_options -leakage_power true` command and set dynamic power constraints on specific scenarios by using the `set_scenario_options -dynamic_power true` command. For multiple dynamic power scenarios, optimization uses the average switching activity calculated from data in the SAIF files.

See Also

- [Optimizing Multicorner-Multimode Designs](#)

Leakage Power and Dynamic Power Optimization

Design Compiler automatically performs leakage power optimization (except in DC Expert, in which case you must specifically enable the optimization). During leakage power optimization, Design Compiler tries to reduce the overall leakage power in your design without affecting the performance. To do this, the optimization is performed on paths that are not timing-critical. When the target libraries are characterized for leakage power and contain cells characterized for multiple threshold voltages, Design Compiler uses the library cells with appropriate threshold voltages to reduce the leakage power of the design.

When using DC Expert, use the following command to enable leakage power optimization:

```
prompt> set_leakage_optimization true
```

You can also perform dynamic power optimization to reduce dynamic power consumption without affecting the performance. Dynamic power optimization requires switching activity information. The tool optimizes the dynamic power by placing nets with high switching

activity close to each other. Because the dynamic power saving is based on the switching activity of the nets, you need to annotate the switching activity by using the `read_saif` command. You can also optimize dynamic power incrementally to provide better QoR and reduce the runtime.

To enable dynamic optimization in all Design Compiler tools, use the following command:

```
prompt> set_dynamic_optimization true
```

For more information about dynamic power optimization, see the *Power Compiler User Guide*.

Note:

When both leakage and dynamic power optimization are enabled in DC Expert, the tool performs only leakage power optimization.

To learn more about leakage power optimization, see the following topics:

- [Leakage Power Optimization Based on Threshold Voltage](#)
- [Leakage Optimization for Multicorner-Multimode Designs](#)

Leakage Power Optimization Based on Threshold Voltage

Leakage power optimization can use single threshold voltage or multithreshold voltage libraries. However, multithreshold voltage libraries can save more leakage power.

Leakage power is very sensitive to threshold voltage. The leakage power can vary up to 40 times or more based on the threshold voltage used. The higher the threshold voltage, the lower the leakage power. For the same library, timing varies by a much smaller amount, up to 30 percent. The lower the threshold voltage, the faster the timing. For the single-voltage library, the variance of threshold voltage and timing is of a similar magnitude.

For designs that have strict timing constraints, optimize for leakage power only on the non timing-critical paths, using the higher threshold-voltage cells from the multithreshold voltage libraries. When your design has a relatively easy-to-meet timing constraint, you might have a large number of low threshold-voltage cells in your design, resulting in higher leakage power consumption. For optimum results you should start with 1 to 5 percent of the number of cells in the design for the low threshold-voltage cells and gradually increase the percentage until the timing constraint is met. With this technique, your design meets the timing constraint with minimal leakage power consumption.

Multithreshold Voltage Library Attributes

To define threshold voltage groups in the libraries, use the `set_attribute` command and add the following attributes:

- Library-level attribute:

```
default_threshold_voltage_group :string;
```

- Library-cell-level attribute:

```
threshold_voltage_group :string;
```

With these attributes, the threshold voltages are differentiated by the string you specify. When the library has at least two threshold voltage groups or if you have defined threshold voltage groups for your library cells using the `set_attribute` command, the library cells are grouped by the threshold voltage. For accurate multiple threshold voltage optimization, define the threshold voltage group attributes.

For information about comparing the leakage power with respect to the timing characteristics of the target library cells belonging to each threshold voltage group, see [Analyzing Multithreshold Voltage Library Cells](#).

Setting Multithreshold Voltage Constraints

To set the multithreshold voltage constraint, use the `set_multi_vth_constraint` command. The command specifies the maximum percentage of cells, by cell count or area, that the design can contain from the specified low-threshold-voltage groups.

You can also specify whether the `set_multi_vth_constraint` constraint should have a higher or lower priority than the timing constraint by using the `-type` option. When you specify `-type hard`, Design Compiler tries to meet the `set_multi_vth_constraint` constraint, even if it results in timing degradation. When you specify `-type soft`, Design Compiler tries to meet the `set_multi_vth_constraint` constraint without degrading the timing.

Note:

The `-type soft` option is supported only in Design Compiler topographical mode.

While calculating the percentage of low-threshold-voltage cells in the design, the tool does not consider the black box cells. To consider the black box cells in the percentage calculation, specify the `-include_blackboxes` option.

After synthesis, use the `report_threshold_voltage_group` command to see the percentage of the total design, by cell count and by area, that is occupied by the low threshold-voltage cells.

In the following example, the maximum percentage of low-threshold-voltage cells in the design is set to 15 percent. Design Compiler tries to meet this constraint without compromising the timing constraint.

```
dc_shell-topo> set_multi_vth_constraint \
    -lvth_groups {lvt svt} -lvth_percentage percentage \
    -type soft -include_blackboxes
```

Note:

If you use the `set_multi_vth_constraint` command, it takes precedence over leakage optimization.

Leakage Optimization for Multicorner-Multimode Designs

For multicorner-multimode designs, you can specify the leakage power optimization for specific scenarios by using the `set_scenario_options` command.

For more information about leakage power optimization for multicorner-multimode designs, see [Power Optimization in Multicorner-Multimode Designs](#).

Comparing Design Compiler Topographical and IC Compiler Environments

Sometimes Design Compiler in topographical mode and IC Compiler have different environment settings. These differences can lead to correlation problems. To help fix these correlation issues, use the `consistency_checker` command in your Linux shell to compare the respective environments.

Before you can compare the two environments, you need to determine the environments of each tool. To do this, use the `write_environment` command as shown in [Example 7-4](#) and [Example 7-5](#). Use the `-output` option to specify the name of your output report. In the Design Compiler tool, run the `write_environment` command before `compile_ultra`; in the IC Compiler tool, run the `write_environment` command before `place_opt`.

Example 7-4 Using write_environment in Design Compiler

```
dc_shell-topo> write_environment -consistency -output DCT.env
```

Example 7-5 Using write_environment in IC Compiler

```
icc_shell> write_environment -consistency -output ICC.env
```

To write compressed environment files, specify the `-compress` option when you run the `write_environment -consistency` command. You must decompress the output file by using the `gunzip` Linux command before using the `consistency_checker` command.

After you have the environments for each tool, compare the environments using the `consistency_checker` command, as shown in [Example 7-6](#).

Example 7-6 Comparing Design Compiler and IC Compiler Environments

```
% consistency_checker -file1 DCT.env -file2 ICC.env \
                           -folder temp \
                           -html html_report | tee cc.log
```

The resulting HTML output report lists mismatched commands and variables. In [Example 7-6](#), the HTML output report is written to the `./html_report` directory. To access this report, open the `./html_report/index.html` file in any Web browser. To reduce correlation problems, fix these mismatches before proceeding to optimization.

In addition to the HTML output report, the tool prints a summary report to the shell while the `consistency_checker` command is running. You can use the Linux `tee` command, as shown in [Example 7-6](#), to save this report to a file. In [Example 7-6](#), the report is saved to the `./cc.log` file.

Note:

The `./temp` directory stores intermediate files the tool uses when executing the `consistency_checker` command. After the command completes, you can remove the `./temp` directory.

See Also

- [SolvNet article 026366, “Using the Consistency Checker to Automatically Compare Environment Settings Between Design Compiler and IC Compiler”](#)
- [Comparing Design Settings Between Design Compiler and IC Compiler II](#)

Comparing Design Settings Between Design Compiler and IC Compiler II

Sometimes the Design Compiler and IC Compiler II tools have different design settings, which might lead to correlation and QoR problems. Checking the design settings manually can be time consuming. To resolve the correlation issues and other problems, use the Design Compiler to IC Compiler II consistency checker to extract and automatically compare the settings.

The Design Compiler to IC Compiler II consistency checker is supported in the Design Compiler tool in wire load mode, topographical mode, Design Compiler NXT, and Design Compiler Graphical.

Before you use the consistency checker, you must download the `consistency_checker.tar.gz` tar file. Extracting the files produces a directory containing the following scripts:

- `snps_settings.tcl` – Tcl script to write the tool settings into a file that works in both tools.
- `snps_consistency_checker.pl` – Perl script to compare the setting files generated by the Design Compiler and IC Compiler II tools.

To use the consistency checker, perform the following steps after loading the design and applying the constraints and before running the `compile_ultra` command in Design Compiler and the `place_opt` command in IC Compiler II:

1. Source the `snps_settings.tcl` file in both the Design Compiler and IC Compiler II shells:

```
prompt> source consistency_checker/snps_settings.tcl
```

2. Run the `report_settings` procedure and redirect the output to a file. For example, in the Design Compiler tool, enter

```
dc_shell> report_settings > dc_settings
```

In the IC Compiler II tool, enter

```
icc2_shell> report_settings > icc2_settings
```

3. In Linux, run the `snps_consistency_checker.pl` script and specify the Design Compiler and IC Compiler II setting files:

```
% consistency_checker/snps_consistency_checker.pl \
  dc_settings icc2_settings
```

Comparing timer settings:

```
-----
Warning: Variable mismatch: DC: timing_enable_through_paths = false
          ICC2: timing_enable_through_paths = true
Warning: Variable mismatch: DC: disable_auto_time_borrow = false
          ICC2: disable_auto_time_borrow = true
```

Comparing placer settings:

```
-----
Warning: Variable mismatch: DC: placer_enable_enhanced_router equiv_value = medium
          ICC2: place.coarse.congestion_analysis_effort = high
Warning: Variable mismatch: DC: placer_enable_enhanced_soft_blockages = true
          ICC2: place.coarse.enable_enhanced_soft_blockages = false
```

You can now analyze the mismatches and debug the inconsistencies.

To download the Design Compiler to IC Compiler II consistency checker, see [SolvNet article 2112936, “Automatically Finding Inconsistent Settings Between Design Compiler or IC Compiler and IC Compiler II.”](#)

See Also

- [Comparing Design Compiler Topographical and IC Compiler Environments](#)

8

Defining Design Constraints

Constraints are declarations that define your design's goals in measurable circuit characteristics, such as timing, area, and capacitance. Design Compiler needs these constraints to effectively optimize the design.

To learn the concepts and tasks necessary for defining design constraints, see

- [Constraint Types](#)
- [Design Rule Constraints](#)
- [Optimization Constraints](#)
- [Managing Constraint Priorities](#)
- [Disabling the Cost Function](#)
- [Reporting Constraints](#)
- [Reporting Runtime Issues Related to SDC Constraints](#)
- [Propagating Constraints in Hierarchical Designs](#)

Constraint Types

When Design Compiler optimizes your design, it uses two types of constraints:

- [Design Rule Constraints](#)

The logic library defines these implicit constraints. These constraints are required for a design to function correctly. They apply to any design that uses the library. By default, design rule constraints have a higher priority than optimization constraints.

- [Optimization Constraints](#)

You define these explicit constraints. Optimization constraints apply to the design on which you are working for the duration of the dc_shell session and represent the design's goals. During optimization, Design Compiler attempts to meet these goals, but no design rules are violated by the process. To optimize a design correctly, you must set realistic constraints.

You specify constraints interactively on the command line or in a constraints file. Design Compiler tries to meet both design rule constraints and optimization constraints, but design rule constraints take precedence.

[Figure 8-1](#) and [Figure 8-2](#) show the major Design Compiler design rule constraints and optimization constraints and the dc_shell interface commands to set the constraints.

Figure 8-1 Major Design Rule Constraints

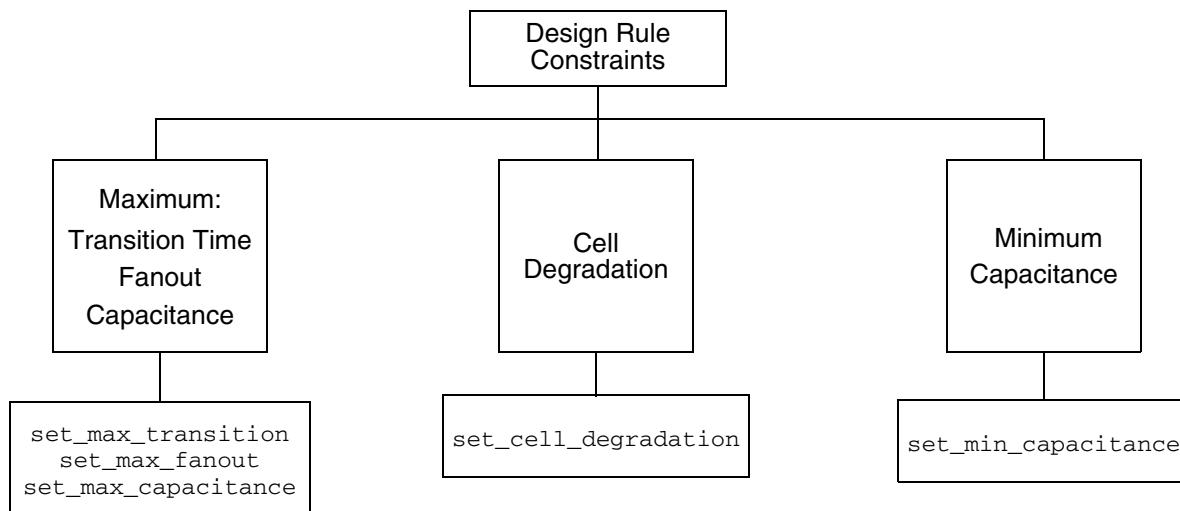
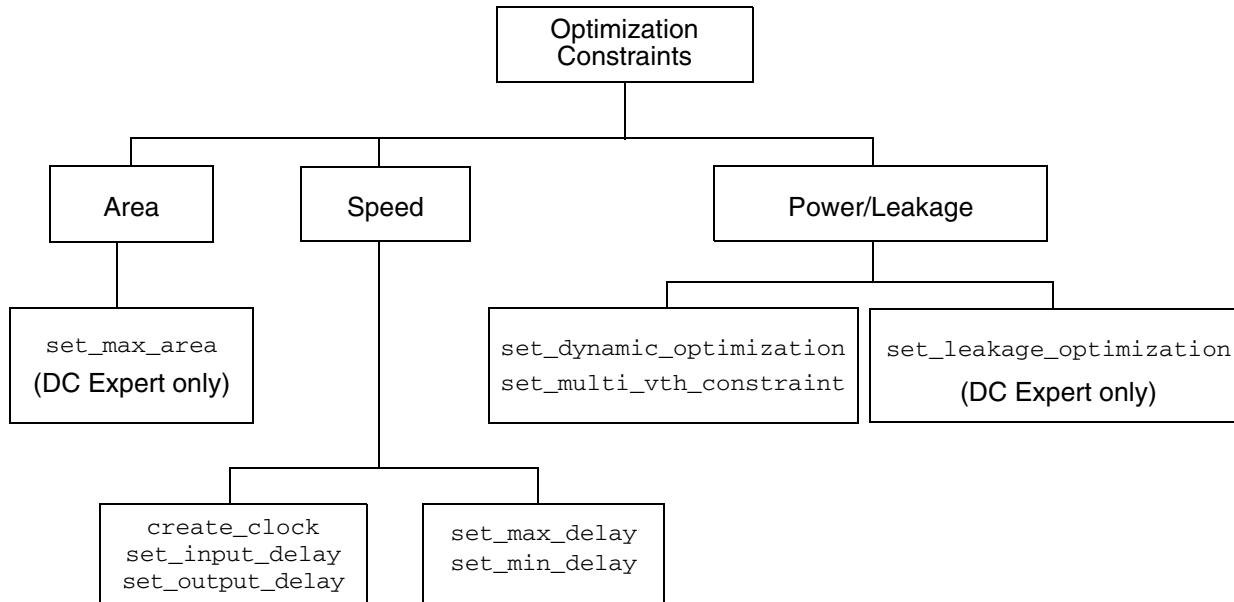


Figure 8-2 Major Design Optimization Constraints



See Also

- [Managing Constraint Priorities](#)
- [The Power Compiler User Guide](#)

Design Rule Constraints

Design rule constraints reflect technology-specific restrictions that your design must meet to function as intended. Design rules constrain the nets of a design but are associated with the pins of cells from a logic library. Most logic libraries specify default design rules.

Typical design rules constrain transition times, fanout loads, and capacitances. You can also specify additional design rules. Design Compiler cannot violate design rule constraints, even if it means violating optimization constraints (delay, power, and area goals). You can apply more restrictive design rules, but you cannot apply less restrictive ones.

These are the design rule constraint types:

- [Maximum Transition Time](#)
- [Maximum Fanout](#)
- [Maximum Capacitance](#)

- [Minimum Capacitance](#)
- [Cell Degradation](#)
- [Connection Class](#)

You cannot remove the `max_transition`, `max_fanout`, `max_capacitance`, and `min_capacitance` attributes set in a logic library, because they are required, but you can set stricter values.

For information about fixing design rule violations, see

- [Fixing Design Rule Violations](#)
- [Disabling DRC Violation Fixing on Special Nets](#)

See Also

- [Summary of Design Rule Commands and Objects](#)
- [Design Rule Constraint Precedence](#)
- [Design Rule Scenarios](#)

Maximum Transition Time

Maximum transition time is a design rule constraint. The maximum transition time for a net is the longest time required for its driving pin to change logic values. Many logic libraries contain restrictions on the maximum transition time for a pin, creating an implicit transition time limit for designs using that library. Transition times on nets are computed by use of timing data from the logic library.

To change or add to the implicit transition time values from a logic library, use the `set_max_transition` command. The command sets the `max_transition` attribute to the specified value on clock groups, ports, or designs. During compile, Design Compiler attempts to ensure that the transition time for each net is less than the specified value, for example, by buffering the output of the driving gate. The `max_transition` constraint value can vary with the operating frequency of a cell.

If both a library `max_transition` and a design `max_transition` attribute are defined, Design Compiler tries to meet the smaller (more restrictive) value.

If your design uses multiple logic libraries and each has a different default `max_transition` value, Design Compiler uses the smallest `max_transition` value globally across the design.

If the transition time for a given driver is greater than the `max_transition` value, Design Compiler reports a design rule violation and works to correct the violation.

The following example sets a maximum transition time of 3.2 for the adder design:

```
prompt> set_max_transition 3.2 [get_designs adder]
```

To undo a `set_max_transition` command, use the `remove_attribute` command:

```
prompt> remove_attribute [get_designs adder] max_transition
```

Specifying Clock-Based Maximum Transition

The `max_transition` value can vary with the operating frequency of a cell. The operating frequency of a cell is defined as the highest clock frequency on the registers driving a cone of logic. Clock frequencies are defined with the `create_clock` command.

For designs with multiple clock domains, you can use the `set_max_transition` command to set the `max_transition` attribute on pins in a specific clock group.

Design Compiler follows these rules in determining the `max_transition` value:

- When the `max_transition` attribute is set on a design or port and a clock group, the most restrictive constraint is used.
- If multiple clocks launch the same paths, the most restrictive constraint is used.
- If `max_transition` attributes are already specified in a logic library, the tool automatically attempts to meet these constraints during compile.

For example, the following command sets a `max_transition` value of 5 on all pins belonging to the Clk clock group:

```
prompt> set_max_transition 5 [get_clocks Clk]
```

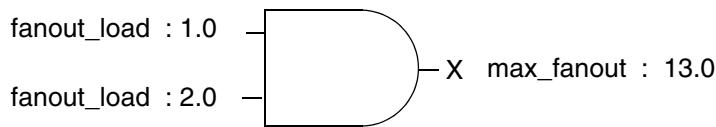
Maximum Fanout

Maximum fanout is a design rule constraint. Most logic libraries place fanout restrictions on driving pins, creating an implicit fanout constraint for every driving pin in designs using that library.

You can set a more conservative fanout constraint on an entire library or define fanout constraints for specific pins in the library description of an individual cell.

If a library fanout constraint exists and you specify a `max_fanout` attribute, Design Compiler tries to meet the smaller (more restrictive) value.

Design Compiler models fanout restrictions by associating a `fanout_load` attribute with each input pin and a `max_fanout` attribute with each output (driving) pin on a cell, as shown in [Figure 8-3](#).

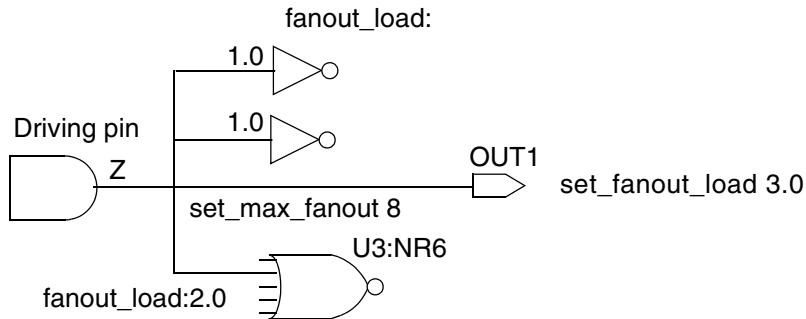
Figure 8-3 fanout_load and max_fanout Attributes

To evaluate the fanout for a driving pin (such as X in [Figure 8-3](#)), Design Compiler calculates the sum of all the `fanout_load` attributes for inputs driven by pin X and compares that number with the number of `max_fanout` attributes stored at the driving pin X.

- If the sum of the fanout loads is not more than the `max_fanout` value, the net driven by X is valid.
- If the net driven by X is not valid, Design Compiler tries to make that net valid, perhaps by choosing a higher-drive component.

Maximum Fanout Calculation Example

[Figure 8-4](#) shows how maximum fanout is calculated.

Figure 8-4 Calculation of Maximum Fanout

You can set a maximum fanout constraint on every driving pin and input port as follows:

```
prompt> set_max_fanout 8 [get_designs ADDER]
```

To check whether the maximum fanout constraint is met for driving pin Z, Design Compiler compares the specified `max_fanout` attribute against the fanout load. If the calculated fanout load is greater than the `max_fanout` value, Design Compiler reports a design rule violation and attempts to correct the violation.

In this case, the design meets the constraints.

$$\begin{array}{c} \text{Total Fanout Load} \\ 8 \geq 1.0 + 1.0 + 3.0 + 2.0 \\ \underbrace{}_{7} \end{array}$$

The fanout load imposed by a driven cell (U3) is not necessarily 1.0. Library developers can assign higher fanout loads (for example, 2.0) to model internal cell fanout effects.

You can also set a fanout load on an output port (OUT1) to model external fanout effects.

Fanout load is a dimensionless number, not a capacitance. It represents a numerical contribution to the total effective fanout.

Defining Maximum Fanout

The `set_max_fanout` command sets the maximum allowable fanout load for the listed input ports. The `set_max_fanout` command sets a `max_fanout` attribute on the listed objects.

To undo a maximum fanout value set on an input port or design, use the `remove_attribute` command. For example,

```
prompt> remove_attribute [get_ports port_name] max_fanout
prompt> remove_attribute [get_designs design_name] max_fanout
```

Defining Expected Fanout for Output Ports

The `set_fanout_load` command sets the expected fanout load value for listed output ports.

Design Compiler adds the fanout value to all other loads on the pin driving each port in `port_list` and tries to make the total load less than the maximum fanout load of the pin.

To undo a `set_fanout_load` command set on an output port, use the `remove_attribute` command:

```
prompt> remove_attribute port_name fanout_load
```

To determine the fanout load, use the `get_attribute` command. For example, to find the fanout load on the input pin of library cell AND2 in library libA, enter

```
prompt> get_attribute "libA/AND2/i" fanout_load
```

To find the default fanout load set on logic library libA, enter

```
prompt> get_attribute libA default_fanout_load
```

Maximum Capacitance

Maximum capacitance is a design rule constraint. It is a pin-level attribute that defines the maximum total capacitive load that an output pin can drive. That is, the pin cannot connect to a net that has a total capacitance (load pin capacitance and interconnect capacitance) greater than or equal to the maximum capacitance defined at the pin.

Design Compiler calculates the capacitance on a net by adding the wire capacitance of the net to the capacitance of the pins attached to the net. You can control capacitance directly and set a maximum capacitance for the nets attached to specified ports or to all the nets in a design by using the `set_max_capacitance` command. The command places a `max_capacitance` attribute on the listed objects. The value should be less than or equal to the `max_capacitance` value of the pin driving the net.

To determine whether a net meets the capacitance constraint, Design Compiler compares the calculated capacitance value with the `max_capacitance` value of the pin driving the net. If the calculated capacitance is greater than the `max_capacitance` value, Design Compiler reports a design rule violation and attempts to correct the violation.

The `max_capacitance` constraint operates similarly to `max_transition`, but the cost is based on the total capacitance of the net, rather than the transition time. The `max_capacitance` attribute functions independently, so you can use it with `max_fanout` and `max_transition`.

The `max_capacitance` value can vary with the operating frequency of a cell. You can have Design Compiler annotate each driver pin with a frequency-based `max_capacitance` value by setting the `compile_enable_dyn_max_cap` variable to `true`. For more information, see [Specifying Frequency-Based Maximum Capacitance](#).

The `max_capacitance` constraint has priority over the cell degradation constraint.

If both a library `max_capacitance` attribute and a design `max_capacitance` attribute exist, Design Compiler tries to meet the smaller (more restrictive) value.

To set a maximum capacitance of 3 for the design adder, enter

```
prompt> set_max_capacitance 3 [get_designs adder]
```

To undo a `set_max_capacitance` command, use the `remove_attribute` command:

```
prompt> remove_attribute [get_designs adder] max_capacitance
```

Specifying Frequency-Based Maximum Capacitance

The `max_capacitance` value can vary with the operating frequency of a cell. You can have Design Compiler annotate each driver pin with a frequency-based `max_capacitance` value by setting the `compile_enable_dyn_max_cap` variable to `true`.

Before you use this feature, your logic library should be characterized for multiple frequencies. This characterization consists of associating a `max_capacitance` value with each driver pin for each frequency and capturing this information in a one-dimensional lookup table. For information about creating the `max_capacitance` lookup table, see the Library Compiler documentation.

Design Compiler follows these steps in determining the `max_capacitance` value:

1. It identifies the operating frequency of each cell.

The operating frequency of a cell is defined as the highest clock frequency on the registers driving a cone of logic. Clock frequencies are defined with the `create_clock` command.

2. It looks up the corresponding `max_capacitance` value for the identified frequency from the lookup table in the logic library.
3. It annotates each driver pin with the appropriate `max_capacitance` value and uses these values in synthesis.

Generating Maximum Capacitance Reports

To report maximum capacitance on violating nets only, use the `report_constraint` command as follows:

```
prompt> report_constraint -all_violators -max_capacitance
...
      Required          Actual
Net        Capacitance    Capacitance   Slack
-----
n_6           0.0014226    0.0128121  -0.0113895 (VIOLATED)
n_16          0.0014226    0.0115580  -0.0101354 (VIOLATED)
n_121         0.0014226    0.0114569  -0.0100343 (VIOLATED)
n_21           0.0014226    0.0113311  -0.0099085 (VIOLATED)
div_18_17/n_2  0.0014226    0.0044171  -0.0029945 (VIOLATED)
div_18_17/n_43 0.0014226    0.0042607  -0.0028381 (VIOLATED)
```

To report maximum capacitance on all nets, use the `report_net` command as follows:

```
prompt> report_net -max_capacitance
...
Net:      Required Capacitance  Actual Capacitance  Slack
-----
CMD[0]       1.25              0.25            1.0 (Met)
CMD[1]       2.50              4.50           -2.0 (Violated)
```

These commands report the following details about nets:

- Required capacitance defined by users or derived using the libraries
- Actual capacitance calculated by the tool

- Slack (the required capacitance minus the actual capacitance)

Minimum Capacitance

Minimum capacitance is a design rule constraint. Some logic libraries specify minimum capacitance. The `min_capacitance` design rule specifies the minimum load a cell can drive. It specifies the lower bound of the range of loads with which a cell has been characterized to operate. During optimization, Design Compiler ensures that the load driven by a cell meets the minimum capacitance requirement for that cell. When a violation occurs, Design Compiler fixes the violation by sizing the driver.

You can use the `min_capacitance` design rule with the existing design rules. The `min_capacitance` design rule has higher priority than the maximum transition time, maximum fanout, and maximum capacitance constraints.

You can set a minimum capacitance for nets attached to input ports to determine whether violations occur for driving cells at the input boundary. If violations are reported after compilation, you can fix the problem by recompiling the module driving the ports. Use the `set_min_capacitance` command to set minimum capacitance on input or inout ports; you cannot set minimum capacitance on a design.

If library `min_capacitance` and design `min_capacitance` attributes both exist, Design Compiler tries to meet the larger (more restrictive) value.

Defining Minimum Capacitance

The `set_min_capacitance` command sets a defined minimum capacitance value on listed input or bidirectional ports. To set a minimum capacitance for nets attached to input or bidirectional ports, use the `set_min_capacitance` command.

To set a minimum capacitance value of 12.0 units on the port named `high_drive`, enter

```
prompt> set_min_capacitance 12.0 high_drive
```

To report only minimum capacitance constraint information, use the `-min_capacitance` option with the `report_constraint` command.

To get information about the current port settings, use the `report_port` command.

To undo a `set_min_capacitance` command, use the `remove_attribute` command.

Cell Degradation

Cell degradation is a design rule constraint. Some logic libraries contain cell degradation tables. The tables list the maximum capacitance that can be driven by a cell as a function of the transition times at the inputs of the cell.

The `cell_degradation` design rule specifies that the capacitance value for a net is less than the cell degradation value. The `cell_degradation` design rule can be used with other design rules, but the `max_capacitance` design rule has a higher priority than the `cell_degradation` design rule. If the `max_capacitance` rule is not violated, applying the `cell_degradation` design rule does not cause it to be violated.

The `set_cell_degradation` command sets the `cell_degradation` attribute to a specified value on specified input ports.

During compilation, if `cell_degradation` tables are specified in a logic library, Design Compiler tries to ensure that the capacitance value for a net is less than the specified value. The `cell_degradation` tables give the maximum capacitance that a cell can drive, as a function of the transition times at the inputs of the cell.

If `cell_degradation` tables are not specified in a logic library, you can set `cell_degradation` explicitly on the input ports.

By default, a port has no `cell_degradation` constraint.

Note:

Use of the `set_cell_degradation` command requires a DC Ultra license.

This command sets a maximum capacitance value of 2.0 units on the port named `late_riser`:

```
prompt> set_cell_degradation 2.0 late_riser
```

To get information about optimization and design rule constraints, use the `report_constraint` command.

To remove the `cell_degradation` attribute, use the `remove_attribute` command.

Connection Class

The connection class constraint on a port describes the connection requirements for a given technology. Only loads and drivers with the same connection class label can be legally connected. You can specify this constraint, or it can be specified in the library. To set a connection class constraint on a port, use the `set_connection_class` command.

Summary of Design Rule Commands and Objects

[Table 8-1](#) summarizes the design rule commands and the objects on which to set them.

Table 8-1 Design Rule Command and Object Summary

Command	Object
set_max_fanout	Input ports or designs
set_fanout_load	Output ports
set_load	Ports or nets
set_max_transition	Ports or designs
set_cell_degradation	Input ports
set_min_capacitance	Input ports

Fixing Design Rule Violations

Design Compiler helps you fix design rule violations. If there are multiple violations, Design Compiler tries to fix the violation with the highest priority first. When possible, it also evaluates and selects alternatives that reduce violations of other design rules. Design Compiler uses the same approach with delay violations.

[Figure 8-5](#) shows the design rule cost function equation.

Figure 8-5 Design Rule Cost Equation

$$\sum_{i=1}^m \max(d_i, 0) \times w_i$$

i = Index
 d = Delta Constraint
 m = Total Number of Constraints
 w = Constraint Weight

In a compilation report, the DESIGN RULE COST column reports the cost function for the design rule constraints on the design.

For more information about fixing design rule violations, see [Design Rule Fixing](#).

See Also

- [Disabling DRC Violation Fixing on Special Nets](#)
- [Managing Constraint Priorities](#)

Disabling DRC Violation Fixing on Special Nets

To enable or disable DRC violation fixing on clock, constant, or scan nets, use the `set_auto_disable_drc_nets` command. The command acts on all the nets of a given type in the current design; that is, depending on the options you specify, it acts on all the clock nets, all the constant nets, all the scan nets, or on any combination of these three net types. Nets that have DRC violation fixing disabled are marked with the `auto_disable_drc_nets` attribute.

By default, the clock and constant nets of a design have DRC violation fixing disabled. This is the same as using the `-default` option of the command; the scan nets do not. You can use the `-all` option to disable DRC violation fixing on all three net types.

Alternatively, you can independently disable or enable DRC violation fixing for the clock nets, constant nets, or scan nets by assigning a `true` or `false` value to the `-on_clock_network`, `-constant`, or `-scan` options, respectively. You can use the `-none` option to enable DRC violation fixing on all three types of nets.

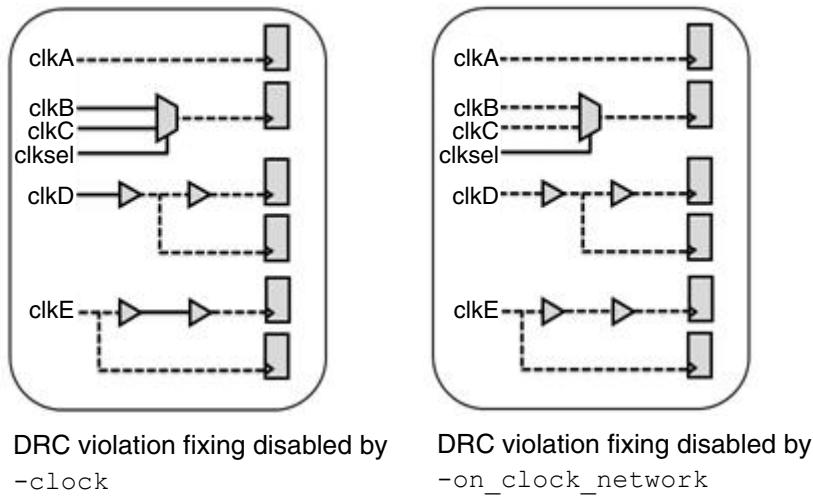
It is recommended that you use the `-on_clock_network` option instead of the `-clock` option to enable or disable DRC violation fixing for the clock network. The `-clock` option is supported for backward compatibility. However, it is set to `true` by default, meaning that it automatically disables DRC violation fixing on clock networks. As shown in [Figure 8-6](#), the `-clock` option starts at register clock pins and propagates back, but not through, the logic on the clock tree. Optimization might buffer non-DRC disabled clock nets upstream of existing clock logic.

To correctly disable DRC violation fixing on the entire clock tree, use the `-on_clock_network` option set to `true`, also shown in [Figure 8-6](#). The option is set to `false` by default. The default setting has not been changed to `true` to avoid affecting existing flows.

If both the `-on_clock_network` option and the `-clock` option are set to `true`, the effect is the same as using the `-on_clock_network` option alone; the `-on_clock_network` option overrides the `-clock` option in this case.

[Figure 8-6](#) shows DRC violation fixing disabled for clock nets by the `-clock` option (on the left) and by the `-on_clock_network` option (on the right). The DRC-disabled clock nets are represented by dashed lines.

Figure 8-6 DRC Violation Fixing Disabled for Clock Nets



Clock nets are ideal nets by default. Using the `set_auto_disable_drc_nets` command to enable design rule fixing does not affect the ideal timing properties of clock nets. You must use the `set_propagated_clock` command to affect the ideal timing of clock nets.

You cannot use the `set_auto_disable_drc_nets` command to override disabled design rule fixing on ideal networks marked with ideal network attributes. This command never overrides the settings specified by the `set_ideal_net` or `set_ideal_network` command.

See Also

- [Fixing Design Rule Violations](#)

Design Rule Constraint Precedence

Design Compiler follows this descending order of precedence when it tries to resolve conflicts among design rule constraints:

1. Minimum capacitance
2. Maximum transition
3. Maximum fanout
4. Maximum capacitance
5. Cell degradation

The following rules determine the precedence of design rule constraints:

- Maximum transition has precedence over maximum fanout. If a maximum fanout constraint is not met, investigate the possibility of a conflicting maximum transition constraint. Design Compiler does not make a transition time worse to fix a maximum fanout violation.
- Maximum fanout has precedence over maximum capacitance.
- Design Compiler calculates transition time for a net in two ways, depending on the library.
 - For libraries using the CMOS delay model, Design Compiler calculates the transition time by using the drive resistance of the driving cell and the capacitive load on the net.
 - For libraries using a nonlinear delay model, Design Compiler calculates the transition time by using table lookup and interpolation. This is a function of capacitance at the output pin and of the input transition time.
- The `set_driving_cell` and `set_drive` commands behave differently, depending on your logic library.
 - For libraries using the CMOS delay model, drive resistance is a constant. In this case, the `set_drive` command and the `set_driving_cell` command give the same result.
 - For libraries using a nonlinear delay model, the `set_driving_cell` command calculates the transition time dynamically, based on the load from the tables. The `set_drive` command returns one value when the command is issued and uses a value from the middle of the range in the tables for load.

Both the `set_driving_cell` and the `set_drive` commands affect the port transition delay. The `set_driving_cell` command places the design rule constraints, annotated with the driving cell, on the affected port.

The `set_load` command places a load on a port or a net. The units of this load must be consistent with your logic library. This value is used for timing optimizations, not for maximum fanout optimizations.

See Also

- [Managing Constraint Priorities](#)

Design Rule Scenarios

Typical design rule scenarios are

- `set_max_fanout` and `set_max_transition` commands
- `set_max_fanout` and `set_max_capacitance` commands

Typically, a logic library specifies a default `max_transition` or `max_capacitance`, but not both. To achieve the best result, do not mix `max_transition` and `max_capacitance`.

Optimization Constraints

Optimization constraints represent speed, area, and power design goals and restrictions that you want but that might not be crucial to the operation of a design. By default, optimization constraints are secondary to design rule constraints. However, that priority can be changed; see [Managing Constraint Priorities](#).

The optimization constraints consist of

- Input and output delays (timing constraints)
- Minimum and maximum delay (timing constraints)
- Maximum area
- Power optimization

To define optimization constraints, see

- [Defining Timing Constraints](#)
- [Defining Area Constraints \(DC Expert Only\)](#)
- [Defining Power Constraints](#)

See Also

- [Compile Cost Function](#)

Defining Timing Constraints

When defining timing constraints you should consider that your design has synchronous paths and asynchronous paths. Synchronous paths are constrained by specifying clocks in the design. Use the `create_clock` command to specify a clock. After specifying the clocks, it is recommended you also specify the input and output port timing specifications. Use the `set_input_delay` and `set_output_delay` commands.

Asynchronous paths are constrained by specifying minimum and maximum delay values. Use the `set_max_delay` and `set_min_delay` commands to specify these point-to-point delays.

For information about maximum and minimum delay calculation, see

- [Maximum Delay](#)
- [Minimum Delay](#)

See Also

- The *Synopsys Timing Constraints and Optimization User Guide*

Maximum Delay

Maximum delay is an optimization constraint. Design Compiler contains a built-in static timing analyzer for evaluating timing constraints. A static timing analyzer calculates path delays from local gate and interconnect delays but does not simulate the design.

The Design Compiler timing analyzer performs critical path tracing to check minimum and maximum delays for every timing path in the design. The most critical path is not necessarily the longest combinational path in a sequential design because paths can be relative to different clocks at path startpoints and endpoints. The timing analyzer calculates minimum and maximum signal rise and fall path values based on the timing values and environmental information in the logic library.

Maximum delay is usually the most important portion of the optimization cost function. The maximum delay optimization cost guides Design Compiler to produce a design that functions at the speed you want. [Figure 8-7](#) shows the maximum delay cost equation.

Figure 8-7 Cost Calculation for Maximum Delay

$$\sum_{i=1}^m v_i \times w_i$$

i = Index
v = worst violation
m = number of path groups
w = weight

You can specify a maximum delay target for paths in the current design by using the `set_max_delay` command. Maximum delay target values for each timing path in the design are automatically determined after considering clock waveforms and skew, library setup times, external delays, multicycle or false path specifications, and `set_max_delay` commands. Load, drive, operating conditions, wire load model, and other factors are also taken into account.

The maximum delay cost is affected by how paths are grouped by the `group_path` and `create_clock` commands.

- If only one path group exists, the maximum delay cost is the cost for that group: the amount of the worst violation multiplied by the group weight.
- If multiple path groups exist, the costs for all the groups are added together to determine the maximum delay cost of the design. The group cost is always zero or greater.

```
Delta = max(delta(pin1), delta(pin2), ... delta(pinN))
```

Design Compiler supports two methods for calculating the maximum delay cost:

- [Worst Negative Slack Method](#) (default behavior)
- [Critical Range Negative Slack Method](#)

Worst Negative Slack Method

By default, Design Compiler uses the worst negative slack method to calculate the maximum delay cost. With the worst negative slack method, only the worst violator in each path group is considered.

A path group is a collection of paths that to Design Compiler represent a group in maximum delay cost calculations. Each time you create a clock with the `create_clock` command, Design Compiler creates a path group that contains all the paths associated with the clock. You can also create path groups by using the `group_path` command. Design Compiler places in the default group any paths that are not associated with any particular group or clock. To see the path groups defined for your design, run the `report_path_group` command.

Because the worst negative slack method does not optimize near-critical paths, this method requires fewer CPU resources than the critical negative slack method. Because of the shorter runtimes, the worst negative slack method is ideal for the exploration phase of the design. Always use the worst negative slack method during default compile runs.

With the worst negative slack method, the equation for the maximum delay cost is

$$\sum_{i=1}^m v_i \times w_i$$

m is the number of path groups.

v_i is the worst violator in the i th path group.

w_i is the weight assigned to the i th path group (the default is 1.0).

Design Compiler calculates the maximum delay violation for each path group as

```
max (0, (actual_path_delay - max_delay))
```

Because only the worst violator in each path group contributes to the maximum delay violation, how you group paths affects the maximum delay cost calculation.

- If only one path group exists, the maximum delay cost is the amount of the worst violation multiplied by the group weight.
- When multiple path groups exist, the costs for all the groups are added to determine the maximum delay cost of the design.

During optimization, the Design Compiler focus is on reducing the delay of the most critical path. This path changes during optimization. If Design Compiler minimizes the initial path's delay so that it is no longer the worst violator, the tool shifts its focus to the path that is now the most critical path in the group.

Critical Range Negative Slack Method

If your design has complex clocking, complex timing requirements, or complex constraints, you can create path groups to focus Design Compiler on specific critical paths in your design.

Use the `group_path` command to create path groups. The `group_path` command allows you to

- Control the optimization of your design
- Optimize near-critical paths
- Optimize all paths

When you add a critical range to a path group, you change the maximum delay cost function from worst negative slack to critical negative slack. The critical range negative slack method considers all violators in each path group that are within a specified delay margin (referred to as the critical range) of the worst violator. Design Compiler optimizes all paths within the critical range.

For example, if the critical range is 2.0 ns and the worst violator has a delay of 10.0 ns, Design Compiler optimizes all paths that have a delay between 8.0 and 10.0 ns.

The critical range negative slack is the sum of all negative slack values within the critical range for each path group. When the critical range is large enough to include all violators, the critical negative slack is equal to the total negative slack.

Using the critical negative slack method, the equation for the maximum delay cost is

$$\sum_{i=1}^m \left(\left\langle \sum_{j=1}^n v_{ij} \right\rangle \times w_i \right)$$

m is the number of path groups.

n is the number of paths in the critical range in the path group.

v_{ij} is a violator within the critical range of the i th path group.

w_i is the weight assigned to the i th path group.

Design Compiler calculates the maximum delay violation for each path within the critical range as

```
max (0, (actual_path_delay - max_delay))
```

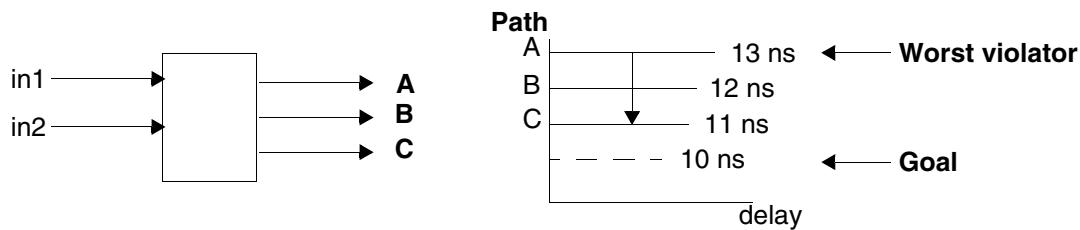
Specifying a critical range can increase runtime. To limit the runtime increase, use critical range only during the final implementation phase of the design, and use a reasonable critical range value. A guideline for the maximum critical range value is 10 percent of the clock period.

Use one of the following methods to specify the critical range:

- Use the `-critical_range` option of the `group_path` command.
- Use the `set_critical_range` command.

For example, [Figure 8-8](#) shows a design with three outputs, A, B, and C.

Figure 8-8 Critical Range Example



Assume that the clock period is 20 ns, the maximum delay on each of these outputs is 10 ns, and the path delays are as shown. By default, Design Compiler optimizes only the worst

violator (the path to output A). To optimize all paths, set the critical delay to 3.0 ns. For example,

```
create_clock -period 20 clk
set_critical_range 3.0 $current_design
set_max_delay 10 {A B C}
group_path -name group1 -to {A B C}
```

See Also

- [Creating Path Groups](#)
- [Optimizing Near-Critical Paths](#)

Minimum Delay

Minimum delay is an optimization constraint, but Design Compiler fixes the minimum delay constraint when it fixes design rule violations. Minimum delay constraints are set explicitly when you use the `set_min_delay` command or implicitly based on hold time requirements. The minimum delay to a pin or port must be greater than the target delay.

The minimum delay cost for a design is different from the maximum delay cost. The minimum delay cost is not affected by path groups, and all violations contribute to the cost. [Figure 8-9](#) shows the minimum delay cost equation.

Figure 8-9 Cost Calculation for Minimum Delay

$$\sum_{i=1}^m v_i = \begin{aligned} & i = \text{index} \\ & m = \text{number of paths affected by} \\ & \quad \text{set_min_delay or set_fix_hold} \\ & v = \text{minimum delay violation} \\ & \quad \max(0, \text{required_path_delay} - \text{actual_path_delay}) \end{aligned}$$

The minimum delay cost function has the same delta for single pins or ports and multiple pins or ports:

```
Delta = min_delay - minimum delay(pin or port)
```

Design Compiler considers the minimum delay cost only if you apply the `set_fix_hold` command to related clocks to fix hold violations at registers during compilation. If `fix_hold` or `min_delay` attributes are set, the minimum delay cost is a secondary optimization cost.

You can override the default path delay for paths affected by the `set_fix_hold` command by using the `set_false_path` or `set_multicycle_path` commands.

See Also

- [Delay Optimization](#)

Defining Area Constraints (DC Expert Only)

By default, DC Ultra in wire load mode, DC Ultra in topographical mode, Design Compiler NXT, and Design Compiler Graphical optimize the design to reduce area whenever possible, unless optimization adversely affects timing and power performance.

In DC Expert, you can specify the maximum allowable area for the current design by using the `set_max_area` command. Defining a maximum area directs Design Compiler to optimize the design for area after timing optimization is complete.

The `set_max_area` command specifies an area target and places a `max_area` attribute on the current design:

```
dc_shell> set_max_area 0.0
dc_shell> set_max_area 14.0
```

The maximum area represents the number of gates in the design, not the physical area the design occupies. Usually the area requirements for the design are stated as the smallest design that meets the performance goal. Design Compiler computes the area of a design by adding the areas of each component on the lowest level of the design hierarchy and the area of the nets. The cell and net areas are technology dependent. Design Compiler obtains this information from the logic library.

Design Compiler calculates the maximum area cost as

```
max (0, actual_area - max_area)
```

Design Compiler ignores the following components when it calculates circuit area:

- Unknown components
- Components with unknown areas
- Technology-independent generic cells

Determining the smallest design can be helpful. The following script guides Design Compiler to optimize for area only. It constrains a design only for minimum area, when you do not care about timing. For the timing to make sense, you must apply clocking and input and output delay.

```
# Example script for smallest design
remove_constraint -all
remove_clock -all
set_max_area 0
```

To prioritize area over total negative slack during area optimization, use the `-ignore_tns` option with the `set_max_area` command. When you use this option, the `ignore_tns` attribute is set on the design and compile might increase delay violations at an endpoint to improve area (if the new delay violation is smaller than the violation on the endpoint of the most critical path in the same path group).

See Also

- [Area Recovery](#)

Defining Power Constraints

Design Compiler automatically performs leakage power optimization (except in DC Expert, in which case you must specifically enable the optimization). During leakage power optimization, Design Compiler tries to reduce the overall leakage power in your design without affecting the performance. To do this, the optimization is performed on paths that are not timing-critical. When the target libraries are characterized for leakage power and contain cells characterized for multiple threshold voltages, Design Compiler uses the library cells with appropriate threshold voltages to reduce the leakage power of the design.

When using DC Expert, use the following command to enable leakage power optimization:

```
prompt> set_leakage_optimization true
```

You can also perform dynamic power optimization to reduce dynamic power consumption without affecting the performance. Dynamic power optimization requires switching activity information. The tool optimizes the dynamic power by placing nets with high switching activity close to each other. Because the dynamic power saving is based on the switching activity of the nets, you need to annotate the switching activity by using the `read_saif` command. You can also optimize dynamic power incrementally to provide better QoR and reduce the runtime.

To enable dynamic optimization in all Design Compiler tools, use the following command:

```
prompt> set_dynamic_optimization true
```

Calculating Maximum Power Cost

Design Compiler computes the maximum power cost only if your logic library is characterized for power.

The maximum power cost has two components:

- Maximum dynamic power

Design Compiler calculates the maximum dynamic power cost as

```
max (0, actual_power - max_dynamic_power)
```

- Maximum leakage power

Design Compiler calculates the maximum leakage power cost as

```
max (0, actual_power - max_leakage_power)
```

For more information about the maximum power cost, see the *Power Compiler User Guide*.

See Also

- [Leakage Power and Dynamic Power Optimization](#)

Managing Constraint Priorities

During optimization, Design Compiler uses a cost vector to resolve any conflicts among competing constraint priorities. [Table 8-2](#) shows the default order of priorities. The table shows that, by default, design rule constraints have priority over optimization constraints. However, you can reorder the priorities of the constraints listed in **bold** type by using the `set_cost_priority` command.

Table 8-2 Constraints Default Cost Vector

Priority (descending order)	Notes
connection classes	Design rule cost
multiple_port_net_cost	Design rule cost
min_capacitance	Design rule constraint
max_transition	Design rule constraint
max_fanout	Design rule constraint
max_capacitance	Design rule constraint
cell_degradation	Design rule constraint
max_delay	Optimization constraint
min_delay	Optimization constraint
power	Optimization constraint
area	Optimization constraint
cell count	

For example, in the following circumstances, you might want to move the `max_delay` optimization constraint ahead of the maximum design rule constraints:

- In many logic libraries, the only significant design rule violations that cannot be fixed without hurting delay are overconstrained nets, such as input ports with large external

loads or around logic marked `dont_touch`. Placing `max_delay` ahead of the design rule constraints in priority allows these design rule constraint violations to be fixed in a way that does not hurt delay. Design Compiler might, for example, resize the drivers in another module.

- In compilation of a small block of logic, such as an extracted critical region of a larger design, the possibility of overconstraints at the block boundaries is high. In this case, design rule fixing might better be postponed until the small block has been regrouped into the larger design.

To prioritize the `max_delay` constraint ahead of the maximum design rule constraints, enter
`prompt> set_cost_priority -delay`

To assign top priority to `max_capacitance`, `max_delay`, and `max_fanout`—in that order—enter

```
prompt> set_cost_priority {max_capacitance max_delay max_fanout}
```

Design Compiler assigns any costs you do not list to a lower priority than the costs you do list. This example does not list `max_transition`, `cell_degradation`, or `min_delay`, so Design Compiler assigns them priority following `max_fanout`.

If you specify `set_cost_priority` more than one time on a design, Design Compiler uses the most recent setting.

Disabling the Cost Function

You can avoid design rule fixing or compile with only design rule fixing. Use the following options with the `compile_ultra` command or `compile` command to control design rule fixing:

- To exit the compile process before fixing design rule violations, use the `-no_design_rule` option. This allows you to check the results in a constraint report.
- To perform only design rule fixing during compile, use the `-only_design_rule` option. Mapping optimizations are not performed. If you are using the `compile_ultra` command, you must use the `-only_design_rule` option with the `-incremental` option.
- To fix only the hold time violations during compile, use the `-only_hold_time` option with the `compile` command. You must specify the `set_fix_hold` command for hold time fixing to be performed. The `compile_ultra` command does not support this option.

Note:

Another way to limit design rule fixing is by using the `set_cost_priority` command with the `-delay` option.

Reporting Constraints

To report the constraint values in the current design to check design rules and optimization goals, use the `report_constraint` command.

The constraint report lists the following for each constraint in the current design:

- Whether the constraint was met or violated and by how much
- The design object that is the worst violator
- The maximum delay information, showing cost by path group. This includes violations of setup time on registers or ports with output delay as well as violations of the `set_max_delay` command.
- The minimum delay cost, which includes violations of hold time on registers or ports with output delay as well as violations of the `set_min_delay` command.

The constraint report summarizes the constraints in the order in which you set the priorities. Constraints not present in your design are not included in the report.

To include more information in a constraint report, use the `-verbose` option with the `report_constraint` command. To list all constraint violators, use the `report_constraint` command with the `-all_violators` option.

Reporting Runtime Issues Related to SDC Constraints

Long tool runtime might be due to factors such as constraint complexity or a high number of timing paths, clocks, or fanout nets. Identifying the runtime issues helps in fixing the runtime issues. You can analyze the SDC constraints that cause long runtime issues.

- When you use the `compile_ultra` command
- Or
- By analyzing the mapped netlist

The following table shows how to enable this feature:

During Compile	Analyzing a Mapped Netlist
<ol style="list-style-type: none"> 1. Read the design. 2. Load the constraints. 3. Set the <code>sdc_runtime_analysis_enable</code> variable to <code>true</code>. 4. Use the <code>compile_ultra</code> command, as shown in the following example. The <code>sdc_runtime.log</code> file is generated with the list of SDC constraints that cause long runtime issues. <pre>read_verilog rtl.v ... set_app_var sdc_runtime_analysis_enable true compile_ultra</pre>	<ol style="list-style-type: none"> 1. Load the <code>.ddc</code> file. 2. Use the <code>link</code> command. 3. Use the <code>check_timing -sdc_runtime</code> command, as shown in the following example. The <code>sdc_runtime.log</code> file is generated with the list of SDC constraints that cause long runtime issues. <pre>read_ddc compiled.ddc ... check_timing -sdc_runtime</pre>

The Design Compiler tool writes information messages related to SDC constraints in the `sdc_runtime.log` file. To specify a different log file name or a standard output file, use the `sdc_runtime_analysis_log_file` variable:

```
dc_shell-topo> set_app_var sdc_runtime_analysis_log_file \
my_log_file.log
```

[Table 8-3](#) lists the variables that affect the contents of the `sdc_runtime.log` file.

Table 8-3 SDC Variables to Report Runtime Issues

Variable Name	Default
<code>sdc_runtime_tightly_constrained_path_group_slack_percentage</code>	75
<code>sdc_runtime_tightly_constrained_same_clock_path_groups</code>	50

Table 8-3 SDC Variables to Report Runtime Issues(Continued)

Variable Name	Default
sdc_runtime_nets_missing_exceptions_fanout_threshold	500
sdc_runtime_top_fanout_nets_missing_exceptions	100
sdc_runtime_port_clock_constraint_threshold	20
sdc_runtime_paths_missing_inter_clock_constraints	5

[Table 8-4](#) describes information messages related to SDC constraints.

Table 8-4 TIM Messages Controlled by the SDC Variables

Information Message	Description
TIM-601	The tool issues this message when there are unused clocks in a design.
TIM-604	The tool issues this message when there are path groups with infeasible paths at different stages in the flow.
TIM-605	Multiple clocks reaching a pin might cause long runtime. This can be due to missing false paths, multicycle paths, or timing exception constraints on a path. In this case, the tool issues TIM-605 messages to identify paths with missing constraints between clock pins.
TIM-610	The tool issues this message when constraints are applied on paths that do not exist.
TIM-615	Missing inter-clock constraints in clock-domain-crossing paths might cause long runtime. In this case, the tool issues one of these messages based on the number of inter-clock constraints specified in the design.
TIM-616	
TIM-617	
TIM-602	The tool reports tightly constrained paths with a critical slack more than 75 percent of the clock period. Use the <code>sdc_runtime_tightly_constrained_path_group_slack_percentage</code> variable to set the threshold for reporting the paths. The variable does not impact optimization. For example, <ul style="list-style-type: none"> • By default, if the clock period is 2.0 ns, the paths with a critical slack more than 1.5 ns are reported. • If you set the variable to 50, the paths with a critical slack more than 1 ns are reported. Note that the QoR remains the same in both the instances.

Table 8-4 TIM Messages Controlled by the SDC Variables (Continued)

Information Message	Description
TIM-603	The tool reports tightly constrained paths when more than 50 path groups are created from a single clock. Use the <code>sdc_runtime_tightly_constrained_same_clock_path_groups</code> variable to set the threshold to identify whether there are too many path groups using the same clock.
TIM-606	The tool reports the top 100 tightly constrained paths with a fanout more than 500 when high-fanout nets are not set with the <code>set_ideal_network</code> or <code>set_dont_touch</code> command. Use the <code>sdc_runtime_nets_missing_exceptions_fanout_threshold</code> and <code>sdc_runtime_top_fanout_nets_missing_exceptions</code> variables to set the fanout threshold and the number of nets to display. The defaults are 500 and 100, respectively.
TIM-607	Multiple clocks associated with input and output ports might cause long runtime. In this case, the tool reports the port and the associated multiple clocks when the <code>set_input_delay</code> or <code>set_output_delay</code> command setting applied on a port is more than 20. Use the <code>sdc_runtime_port_clock_constraint_threshold</code> variable to set the threshold for reporting the paths.
TIM-608	Missing multicycle or false path constraints might cause high worst negative slack during the generation of the first QoR. In this case, the tool reports the top 5 tightly constrained paths per path group. Use the <code>sdc_runtime_paths_missing_inter_clock_constraints</code> variable to set the threshold for reporting the paths. Note that this variable also impacts how the critical paths are identified to report this message.

Propagating Constraints in Hierarchical Designs

Hierarchical designs are composed of subdesigns. You can propagate constraints up or down the hierarchy in the following ways:

- **Characterizing Subdesigns**

The characterizing method captures information about the environment of specific cell instances and assigns the information as attributes on the design to which the cells are linked.

- **Modeling**

The modeling method creates a characterized design as a library cell.

- [Propagating Constraints up the Hierarchy](#)

This method propagates clocks, timing exceptions, and disabled timing arcs from lower level subdesigns to the current design.

Characterizing Subdesigns

When you compile subdesigns separately, boundary conditions such as the input drive strengths, input signal delays (arrival times), and output loads can be derived from the parent design and set on each subdesign. You can do this in the following ways:

- Manually

Use the `set_drive`, `set_driving_cell`, `set_input_delay`, `set_output_delay`, and `set_load` commands.

- Automatically

Use the `characterize` command.

Using the `characterize` Command

The `characterize` command places on a design the information and attributes that characterize its environment in the context of a specified instantiation in the top-level design.

The primary purpose of `characterize` is to capture the timing environment of the subdesign. This occurs when you use `characterize` with no arguments or when you use its `-constraints`, `-connections`, or `-power` options.

The `characterize` command derives and asserts the following information and attributes on the design to which the instance is linked:

- Unless the `-no_timing` option is specified, the `characterize` command places on the subdesigns any timing characteristics previously set by the following commands:

<code>create_clock</code>	<code>set_load</code>
<code>group_path</code>	<code>set_max_delay</code>
<code>read_timing</code>	<code>set_max_time_borrow</code>
<code>set_annotated_check</code>	<code>set_min_delay</code>
<code>set_annotated_delay</code>	<code>set_multicycle_path</code>
<code>set_auto_disable_drc_nets</code>	<code>set_operating_conditions</code>
<code>set_drive</code>	<code>set_output_delay</code>

<code>set_driving_cell</code>	<code>set_resistance</code>
<code>set_false_path</code>	<code>set_timing_ranges</code>
<code>set_ideal_net</code>	<code>set_wire_load_model</code>
<code>set_ideal_network</code>	<code>set_wire_load_mode</code>
<code>set_input_delay</code>	<code>set_wire_load_selection_group</code>
<code>set_wire_load_min_block_size</code>	

- If you specify the `-constraint` option, the `characterize` command places on the subdesigns any area, power, connection class, and design rule constraints previously set by the following commands:

<code>set_cell_degradation</code>	<code>set_max_fanout</code>
<code>set_connection_class</code>	<code>set_max_power</code>
<code>set_dont_touch_network</code>	<code>set_max_transition</code>
<code>set_fanout_load</code>	<code>set_min_capacitance</code>
<code>set_max_area</code>	<code>set_max_capacitance</code>

- If you specify the `-connection` option, the `characterize` command places on the subdesigns the connection attributes set by the following commands. Connection class information is applied only when you use the `-constraint` option.

<code>set_equal</code>	<code>set_logic_zero</code>
<code>set_logic_dc</code>	<code>set_opposite</code>
<code>set_logic_one</code>	<code>set_unconnected</code>

- If you specify the `-power` option, the `characterize` command places on the subdesigns the switching activity information, toggle rates, and static probability previously set, calculated, or saved by the following commands:

<code>report_power</code>	<code>set_switching_activity</code>
---------------------------	-------------------------------------

Removing Previous Annotations

In most cases, characterizing a design removes the effects of a previous characterization and replaces the relevant information. However, in the case of back-annotation (`set_load`, `set_resistance`, `read_timing`, `set_annotated_delay`, `set_annotated_check`), the `characterize` step removes the annotations and cannot overwrite existing annotations made on the subdesign. In this case, you must explicitly remove annotations from the subdesign (using `reset_design`) before you run the `characterize` command again.

Optimizing Bottom Up Versus Optimizing Top Down

During optimization, you can use `characterize` with `set_dont_touch` to maintain hierarchy. This is known as bottom-up optimization, which you can apply by using a golden instance or a uniquify approach (either manually with the `uniquify` command or automatically as part of the `compile` command). An alternative to bottom-up optimization is top-down optimization, also called hierarchical compile. During top-down optimization, the tool automatically performs characterization and optimization for subdesigns.

Deriving the Boundary Conditions

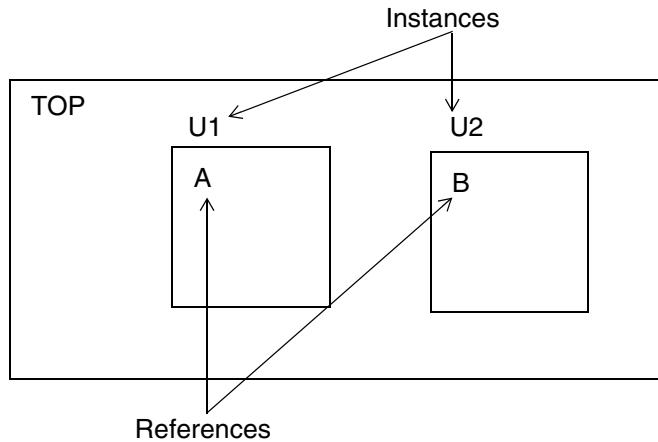
The `characterize` command automatically derives the boundary conditions of a subdesign based on its context in a parent design. It examines an instance's surroundings to obtain actual drive, load, and timing parameters and computes the following types of boundary conditions:

- Timing conditions
 - Expected signal delays at input ports.
- Constraints
 - Inherited requirements from the parent design, such as maximum delay.
- Connection relations
 - Logical relationships between ports or between ports and power or ground, such as always logic 0, logical opposite of another port, or unconnected.

The `characterize` command summarizes the boundary conditions for one instance of a subdesign in one invocation. The result is applied to the reference.

Figure 8-10 shows instances and references.

Figure 8-10 Instances and References



If a subdesign is used in more than one place, you must either characterize it manually or create a copy of the design for each instantiation and characterize each. For more information, see [Characterizing Multiple Instances](#).

Saving Attributes and Constraints

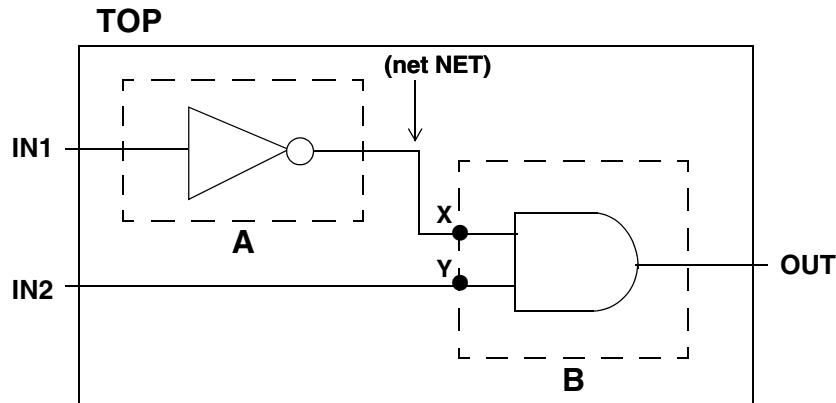
The `characterize` command captures the timing environment of the subdesign. Use the `write_script` command with `characterize` to save the attributes and constraints for the current design. For more information about saving and re-creating attribute values, see [Saving Attribute Values](#).

The `characterize` Command Calculations

The `characterize` command derives specific load and timing values for ports. The command uses values that allow the timing numbers on the output ports of a characterized design to be the same as if the design were flattened and then timed.

The [Load Calculations](#) and [Input Delay Calculations](#) sections use the hierarchical design example in [Figure 8-11](#) to describe the load and input delay calculations used by the `characterize` command.

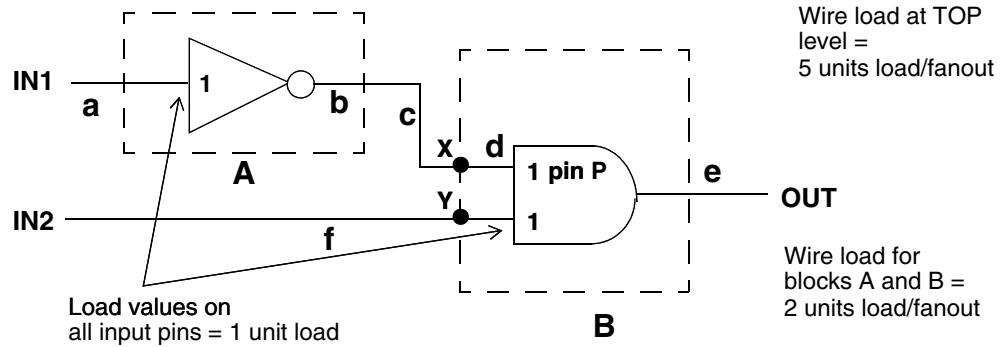
Figure 8-11 Hierarchical Design Example



Load Calculations

[Figure 8-12](#) provides values for wire loads and input pin capacitances for the example shown in [Figure 8-11](#).

Figure 8-12 Hierarchical Design With Annotated Loads



In [Figure 8-12](#), a through f are wire segments used in the calculations. This example assumes a segmented wire loading model, which takes the interconnection net loads on the blocks into account and uses a linear function for the wire loads.

- The calculation for the outside load of pin P of hierarchical block B is

outside load = sum of the loads of all pins on the net
loading P that are not in B
plus the sum of the loads of all segments of net
driving or loading P that are not in B

- The calculation for each segment's load is

segment load = number of fanouts * wire load

- The calculation for the outside load on input IN1 to block A uses 0 driving pins, a fanout count of 1 for segment a, and the TOP wire load of 5 loads per fanout.

The calculation is

$$\begin{aligned} \text{load pins on driving net + load of segment a} \\ = 0 + (1 * 5) \\ = 5 \end{aligned}$$

- The calculation for the outside load on the output of block A is

$$\begin{aligned} \text{load pins on net} \\ + \text{load of segment c} \\ + \text{load of segment d} \\ = 1 \text{ (for load pin P)} \\ + (1 * 5) \\ + (1 * 2) \\ = 8 \end{aligned}$$

For each segment in the calculation, the local wire load model is used to calculate the load. That is, the calculation for block A's output pin uses TOP's wire load of 5 loads per fanout for segment c and block B's wire load of 2 loads per fanout for segment d.

- The calculation for the outside load on the output of block B is

$$\begin{aligned} \text{load pins on net + load of segment e} \\ = 0 + (1 * 5) \\ = 5 \end{aligned}$$

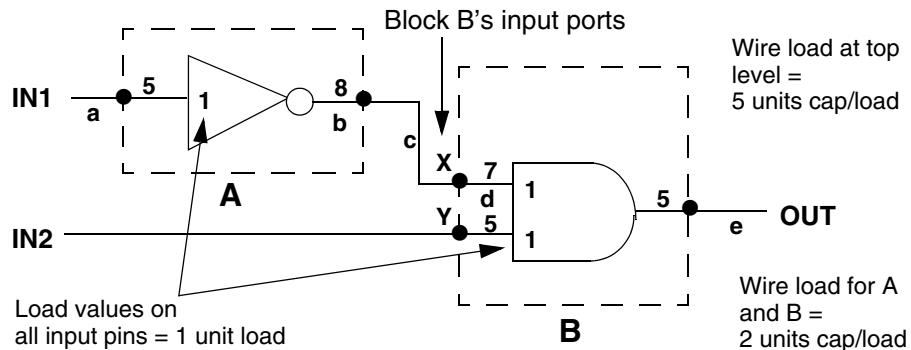
- The calculation for the outside load on the input pin X of block B is

$$\begin{aligned} \text{load pins on net} \\ + \text{load of segment b} \\ + \text{load of segment c} \\ = 0 + (1 * 2) + (1 * 5) \\ = 7 \end{aligned}$$

- The calculation for the outside load on the input pin Y is

$$\begin{aligned} \text{pin loads on driving net + load of segment f} \\ = 0 + 1 * 5 \\ = 5 \end{aligned}$$

[Figure 8-13](#) shows the modified version of [Hierarchical Design Example](#) with the outside loads annotated.

Figure 8-13 Loads After Characterization

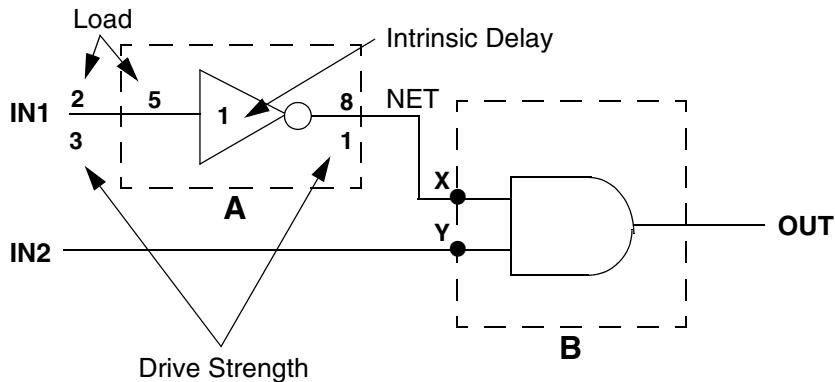
Input Delay Calculations

Because characterizing provides accurate details of outside loads, the path delays of input signals reflect only the delay through the intrinsic delay of the last gate driving the port. The path delays of input signals do not include the gate's load delay or the connect delay on the net.

For example, the characterized input delay on the input pins of block B is calculated from the delay to the pin that drives the port being characterized, without the gate's load delay or the connect delay on the net.

The timing calculations for characterizing block B follow [Figure 8-14](#).

[Figure 8-14](#) shows the default drive strengths and intrinsic delays of block A and signal IN1.

Figure 8-14 Design With Annotations for Timing Calculations

The delay calculation for input pin X is

```
drive strength at IN1 * (wire load + pin load)
    + intrinsic delay of A's
cell
    = 3 * (5 + 2 + 1)
    +
    = 25
```

Characterizing Subdesign Port Signal Interfaces

The `characterize` command with no options replaces a subdesign's port signal information (clocks, port drive, input and output delays, maximum and minimum path delays, and port load) with information derived from the parent design. The subdesign also inherits operating conditions and timing ranges from the top-level design.

You can manually set port signal information by using the following commands:

```
create_clock
set_clock_latency
set_clock_uncertainty
set_drive
set_driving_cell
set_input_delay
set_load
set_max_delay
set_min_delay
set_output_delay
set_propagated_clock
```

The `characterize` command sets the wire loading model selection group and model for subdesigns. The subdesigns inherit the top-level wire loading mode. The wire loading model for subdesigns is determined as follows:

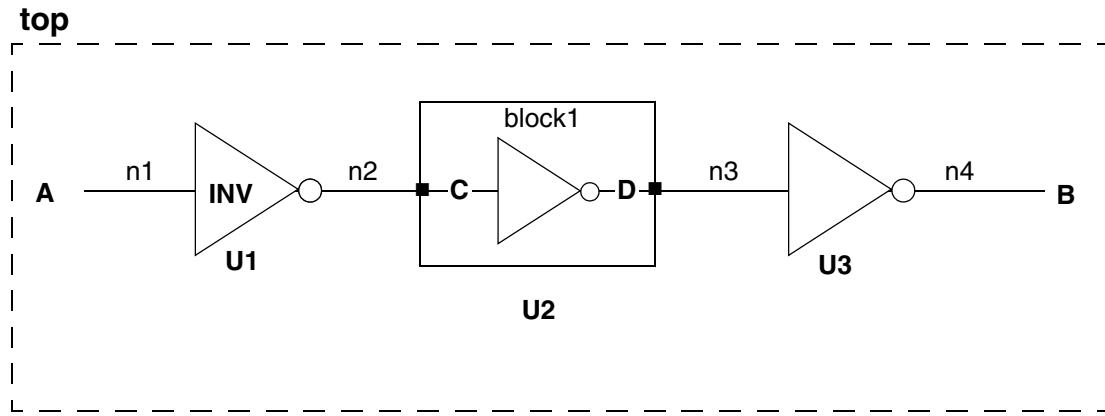
- If the top-level mode is top, the subdesigns inherit the top-level wire loading model. Unless an instance specific model or selection group has been specified. Instance specific model and selection group settings take precedence over design level settings.
- If the top-level mode is enclosed or segmented, the wire loading model is based on the following:
 - If no wire loading model is defined for the lower block and the wire load cannot be determined by the area, the wire loading model of the top-level design is used.
 - If no wire loading model is defined for the lower block but the wire load can be determined by the area, the wire loading model is reselected at compile time, based on the cell area.

Combinational Design Example

[Figure 8-15](#) shows subdesign block in the combinational design top. Set the port interface attributes either manually or automatically.

- Script 1 uses characterize to set the attributes automatically.
- Script 2 sets the attributes manually.

Figure 8-15 Characterizing Drive, Timing, and Load Values—Combinational Design



```
current_design top
set_input_delay 0 A
set_max_delay 10 -to B
```

Script 1

```
current_design top
characterize U2
```

Script 2

```
current_design block1
set_driving_cell -lib_cell INV {C}
set_input_delay 3.3 C
set_load 1.3 D
set_max_delay 9.2 -to D
current_design top
```

In Script 2,

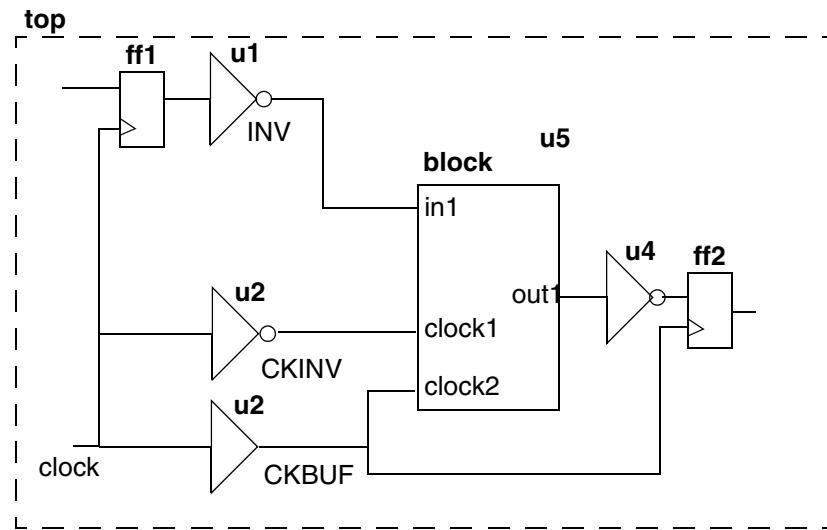
- Line 2 captures driving cell information.
- Line 3 sets the arrival time of net n2 as 3.3.
- Line 4 sets the load of U3 as 1.3.
- Line 5 sets the inherited `set_max_delay`, which is $10 - .8 (.4$ for each inverter).

Sequential Design Example

[Figure 8-16](#) shows subdesign block in the sequential design top. Set the port interface information either manually or automatically.

- Script 1 sets the information manually.
 - The input delay (from ff1/CP to u5/in1) is 1.8
 - The output delay (through u4 plus the setup time of ff2) is 1.2
 - The outside load on the out1 net is 0.85
- Script 2 uses the `characterize` command to set the information automatically.

Figure 8-16 Characterizing Sequential Design Drive, Timing, and Load Values



Script 1

```
current_design top
create_clock -period 10 -waveform {0 5} clock
current_design block
create_clock -name clock -period 10 -waveform {0 5} clock1
create_clock -name clock_bar -period 10 \
    -waveform {5 10} clock2
set_input_delay -clock clock 1.8 in1
set_output_delay -clock clock 1.2 out1
set_driving_cell -lib_cell INV -input_transition_rise 1 in1
set_driving_cell -lib_cell CKINV clock1
set_driving_cell -lib_cell CKBUF clock2
set_load 0.85 out1
current_design top
```

Script 2

```
current_design top
create_clock -period 10 -waveform {0 5} clock
characterize u5
```

Characterizing Subdesign Constraints

The `characterize -constraints` command uses values derived from the parent design to replace the `max_area`, `max_fanout`, `fanout_load`, `max_capacitance`, and `max_transition` attributes of a subdesign.

Characterizing Subdesign Logical Port Connections

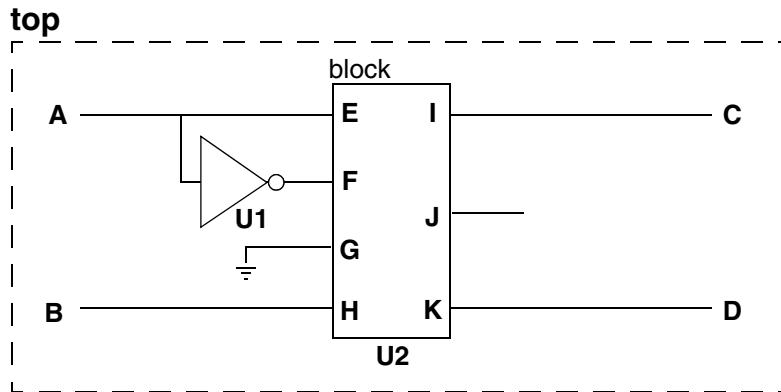
The `characterize -connections` command uses connection attributes derived from the parent design to replace the port connection attributes of a subdesign.

The connection attributes are those set by the commands `set_equal`, `set_opposite`, `set_logic_one`, `set_logic_zero`, and `set_unconnected`.

Figure 8-17 shows subdesign block in design top. Set logical port connections either manually or automatically.

- Script 1 sets the attributes manually.
- Script 2 uses `characterize -no_timing -connections` to set the attributes automatically. The `-no_timing` option inhibits computation of port timing information.

Figure 8-17 Characterizing Port Connection Attributes

**Script 1**

```
current_design block
set_opposite    { E F }
set_logic_zero  { G }
set_unconnected { J }
```

Script 2

```
current_design top
characterize U2 -no_timing -connections
```

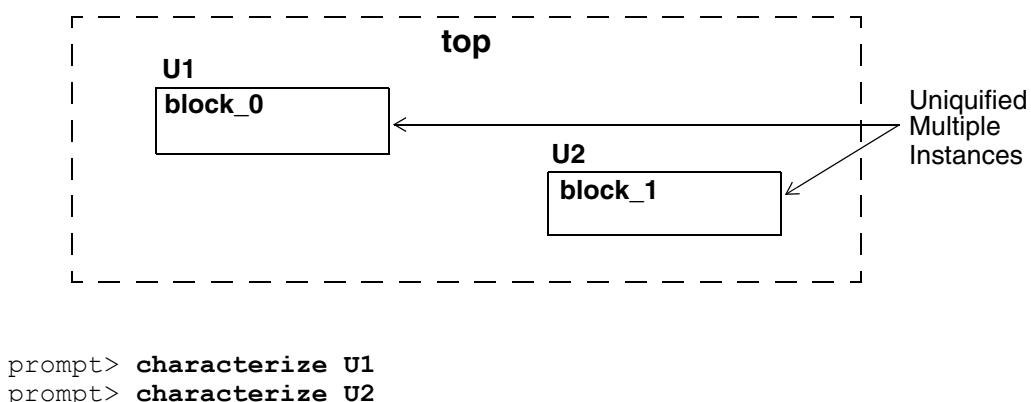
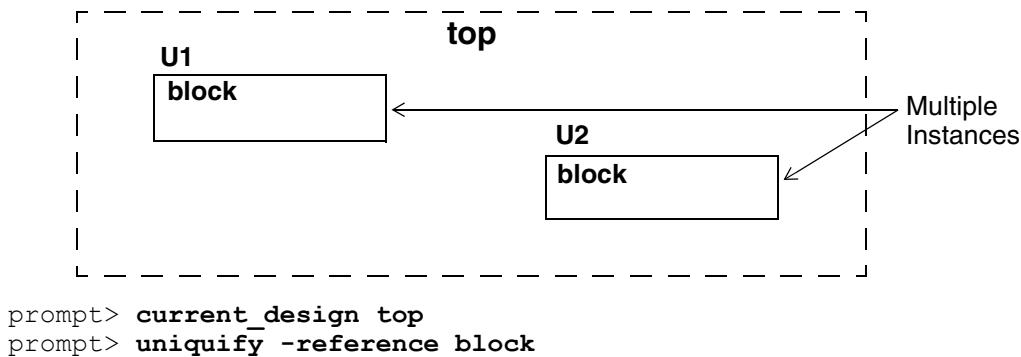
Characterizing Multiple Instances

The `characterize` command summarizes the boundary conditions for one instance of a subdesign in each invocation. If a subdesign is used more than one time, use the `uniquify` command to make each instance distinctive before using `characterize` for each instance.

The `uniquify` command creates copies of subdesigns that are referenced more than one time. It then renames the copies and updates the corresponding cell references.

[Figure 8-18](#) shows how to use `uniquify` and `characterize` for the subdesign block, which is referenced in cells U1 and U2.

Figure 8-18 Characterizing a Subdesign Referenced Multiple Times



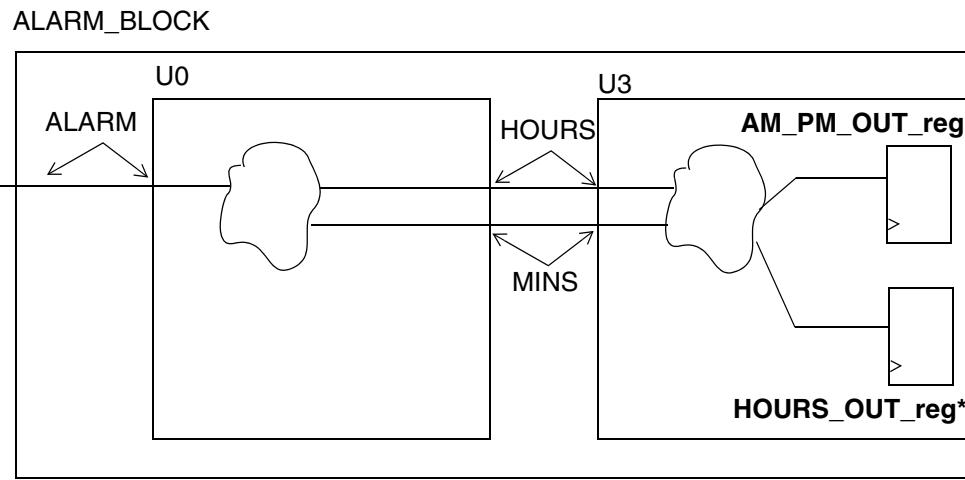
Characterizing Designs With Timing Exceptions

When paths crossing design hierarchies contain different timing exceptions, `characterize` creates timing constraints with virtual clocks to capture this information.

Possible timing exceptions include `set_multicycle_path`, `set_false_path`, `set_max_delay`, and `set_min_delay`. The virtual clock scheme can also handle multiple clocks.

[Figure 8-19](#) shows a sample design in which the path from the ALARM input port to `HOURS_OUT_reg*` in U3 is constrained as a two-cycle path. [Example 8-1](#) shows the a portion of the uncompressed `characterize -verbose` result of U0 block.

Figure 8-19 characterize -verbose Result of U0 Block



Example 8-1 characterize -verbose Output

```

create_clock -period 10 -waveform {0 5} [get_ports {CLK}]
create_clock -name "CLK_virtual1" -period 10 -waveform {0 5}
create_clock -name "CLK_virtual2" -period 10 -waveform {0 5}
create_clock -name "CLK_virtual3" -period 10 -waveform {0 5}
create_clock -name "CLK_virtual4" -period 10 -waveform {0 5}
create_clock -name "CLK_virtual5" -period 10 -waveform {0 5}
set_input_delay 2 -clock "CLK" [get_ports {MINUTES_BUTTON}]
set_input_delay 2 -clock "CLK" [get_ports {HOURS_BUTTON}]
set_input_delay 2 -clock "CLK_virtual1" [get_ports {ALARM_BUTTON}]
set_output_delay 7.62796 -max -rise -clock "CLK" [get_ports {MINS}]
set_output_delay 6.93955 -max -fall -clock "CLK" [get_ports {MINS}]
set_output_delay 2.20324 -min -rise -clock "CLK" [get_ports {MINS}]
set_output_delay 2.40013 -min -fall -clock "CLK" [get_ports {MINS}]
set_output_delay 8.05575 -add_delay -max -rise -clock "CLK_virtual2" \
    [get_ports {MINS}]
set_output_delay 6.83933 -add_delay -max -fall -clock "CLK_virtual2" \
    [get_ports {MINS}]
set_output_delay 2.89679 -add_delay -min -rise -clock "CLK_virtual2" \
    [get_ports {MINS}]
set_output_delay 2.80452 -add_delay -min -fall -clock "CLK_virtual2" \
    [get_ports {MINS}]
...

```

```

set_output_delay 10.232 -add_delay -max -rise -clock "CLK_virtual5" \
[get_ports {MINS}]
set_output_delay 9.4791 -add_delay -max -fall -clock "CLK_virtual5" \
[get_ports {MINS}]
set_output_delay 3.90222 -add_delay -min -rise -clock "CLK_virtual5" \
[get_ports {MINS}]
set_output_delay 3.57818 -add_delay -min -fall -clock "CLK_virtual5" \
[get_ports {MINS}]
...
set_multicycle_path 2 -from [get_clocks {CLK_virtual1}] -to \
[get_clocks {CLK_virtual2}]
set_multicycle_path 2 -from [get_clocks {CLK_virtual1}] -to \
[get_clocks {CLK_virtual3}]
set_multicycle_path 2 -from [get_clocks {CLK_virtual1}] -to \
[get_clocks {CLK_virtual4}]
set_multicycle_path 2 -from [get_clocks {CLK_virtual1}] -to \
[get_clocks {CLK_virtual5}]

```

Limitations of the characterize Command

The `characterize` command provides many useful features, but do not always rely on this command to derive constraints for the subdesigns in a design hierarchy. Before you characterize a design, keep in mind the following limitations:

The `characterize` command

- Does not derive timing budgets. (It reflects the current state of the design.)
- Ignores `clock_skew` and `max_time_borrow` attributes placed on a hierarchical boundary (generally not an issue, because these attributes are usually placed on clocks and cells).

With no options, the `characterize` command replaces a subdesign's port signal information (clocks, port drive, input and output delays, maximum and minimum path delays, and port load) with information derived from the parent design.

The `characterize` command recognizes when the top-level design has back-annotated information (load, resistance, or delay) and to move this data down to the subdesign in preparation for subsequent optimization.

Propagating Constraints up the Hierarchy

If you have hierarchical designs and compile the subdesigns, then move up to the higher-level blocks (bottom-up compilation), you can propagate clocks, timing exceptions, and disabled timing arcs from lower-level .ddc files to the current design, using the `propagate_constraints` command. If you do not use this command, you can propagate constraints from a higher-level design to the `current_instance`, but you cannot propagate constraints set on a lower-level block to the higher-level blocks in which it is instantiated.

Note:

Using the `propagate_constraints` command might cause memory usage to increase.

Example 8-2 shows a methodology in which constraints are propagated upward. Assume that A is the top-level and B is the lower-level design.

Example 8-2 Propagating Constraints Upward

```
current_design B
source constraints.tcl
compile
current_design A
propagate_constraints -design B
report_timing_requirements
compile
report_timing
```

To generate a report of all the constraints that were propagated up, use the `-verbose` and `-dont_apply` options and redirect the output to a file:

```
prompt> propagate_constraints -design name \
           -verbose -dont_apply -output report.cons
```

Use the `write_file -format ddc` command to save the .ddc file with the propagated constraints so there is no need to go through the propagation again when restarting a new dc_shell session.

Handling Conflicts Between Designs

Conflicts between the lower-level and top-level designs causes the following:

- Clock name conflict
 - The lower-level clock has the same name as the clock of the current design (or another block).
The clock is not propagated. A warning is issued.
- Clock source conflict
 - A clock source of a lower-level block is already defined as a clock source of a higher-level block.
The lower-level clock is not propagated. A warning is issued.
- Exceptions from or to an unpropagated clock
 - This can be either a virtual clock, or a clock that was not propagated from that block due to a conflict.

- Exceptions

A lower-level exception overrides a higher-level exception that is defined on the same path.

9

Using Floorplan Physical Constraints

Design Compiler in topographical mode supports high-level physical constraints such as die area, core area and shape, port locations, cell locations and orientations, keepout margins, placement blockages, preroutes, bounds, vias, tracks, voltage areas, and wiring keepouts.

Using floorplan physical constraints in topographical mode improves timing correlation with post-place-and-route tools, such as IC Compiler and IC Compiler II, by considering floorplanning information during optimization.

You provide floorplan physical constraints to Design Compiler topographical mode using one of the following methods:

- Export the floorplan information in DEF files from IC Compiler or IC Compiler II or a Tcl script from IC Compiler and import this information into Design Compiler
- Create the constraints manually

You can examine most of these objects visually in your floorplan by using the Design Vision layout window. For more information about using the GUI to view physical constraints, see the “Viewing the Floorplan” topic in Design Vision Help.

To learn how to create, import, reset, save, and report floorplan physical constraints, see

- [Importing Floorplan Information](#)
- [Manually Defined Physical Constraints](#)
- [Including Physical-Only Cells](#)

- Specifying Relative Placement
- Magnet Placement
- Resetting Physical Constraints
- Saving Physical Constraints Using the write_floorplan Command
- Saving Physical Constraints Using the write_def Command
- Reporting Physical Constraints

Importing Floorplan Information

The main reason to use floorplan constraints in topographical mode is to accurately represent the placement area and improve timing correlation with the post-place-and-route design. You can provide high-level physical constraints that determine core area and shape, port location, macro location and orientation, voltage areas, placement blockages, and placement bounds. These physical constraints can be derived from the IC Compiler or IC Compiler II floorplan data, extracted from an existing Design Exchange Format (DEF) file, or created manually.

You can import floorplan information into the Design Compiler tool by using one of the following methods:

- Using DEF Files

To provide floorplan physical constraints in DEF files to the Design Compiler tool, you export the floorplan information from the IC Compiler or IC Compiler II tool by using the `write_def` command and import this information into the Design Compiler tool by using the `extract_physical_constraints` command. For more information, see the following topics:

- [Using the write_def Command in IC Compiler or IC Compiler II](#)
- [Reading DEF Information in Design Compiler](#)

- Using a Tcl Script

Export the floorplan information from IC Compiler by using the `write_floorplan` command and import this information into the Design Compiler tool by using the `extract_physical_constraints` command. For more information, see the following topics:

- [Using the write_floorplan Command in IC Compiler](#)
- [Reading the Floorplan Script in Design Compiler](#)

See Also

- [Physical Constraints Imported in the DEF File](#)
- [Physical Constraints Imported in the Floorplan File](#)
- [Manually Defined Physical Constraints](#)

Using the write_def Command in IC Compiler or IC Compiler II

To improve timing and power correlation between Design Compiler and IC Compiler or IC Compiler II,

- Read your mapped Design Compiler netlist into IC Compiler or IC Compiler II.
- Create a basic floorplan in IC Compiler or IC Compiler II.
- Export this floorplan from IC Compiler or IC Compiler II.
- Read the floorplan back into Design Compiler.

To export floorplan information from IC Compiler or IC Compiler II in a Design Exchange Format (DEF) file for use in Design Compiler topographical mode, use the `write_def` command in IC Compiler or IC Compiler II. [Example 9-1](#) uses the `write_def` command to write physical design data to the `my_physical_data.def` file.

Example 9-1 Using the write_def Command to Extract Physical Data From IC Compiler or IC Compiler II

```
icc_shell> write_def -version 5.7 -rows_tracks_gcells -macro -pins \
    -blockages -specialnets -vias -regions_groups -verbose \
    -output my_physical_data.def
```

See Also

- [Reading DEF Information in Design Compiler](#)

Reading DEF Information in Design Compiler

To import floorplan information from a DEF file, use the `extract_physical_constraints` command. This command imports physical information from the specified DEF file and applies these constraints to the design. The applied floorplan information is saved in the .ddc file and does not need to be reapplied when you read in the .ddc file in subsequent topographical mode sessions.

The following command shows how you can import physical constraints from multiple DEF files:

```
dc_shell-topo> extract_physical_constraints \
    {des_1.def des2.def ... des_N.def}
```

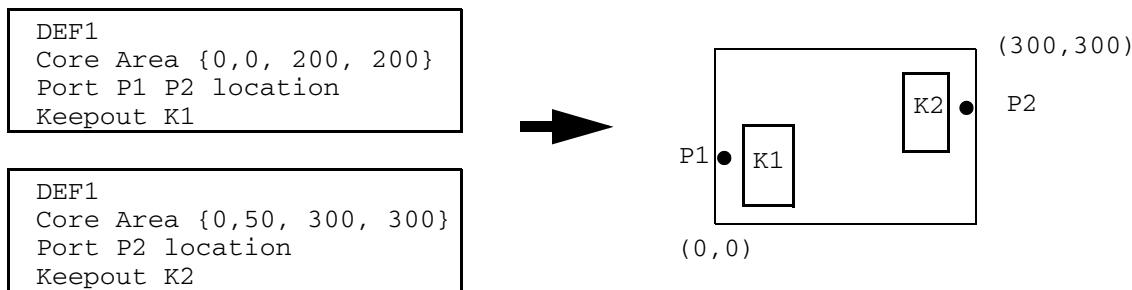
By default, the `extract_physical_constraints` command runs in incremental mode. That is, if you use the command to process multiple DEF files, the command preserves existing physical annotations on the design. In incremental mode, the placement area is imported based on the current core area and site rows in the DEF file. When incremental

mode is disabled, the placement area is imported based on the site rows in the DEF files. Conflicts are resolved as follows:

- Physical constraints that can have only one value are overwritten by the value from the latest DEF file. That is, port location and macro location are overwritten.
- Physical constraints that can have accumulated values are recomputed. That is, core area can be recomputed based on the existing value and the site row definitions in the latest DEF file. Placement keepouts from different DEF files are accumulated and the final keepout geometry is computed internally during synthesis.

[Figure 9-1](#) shows an example of incremental extraction performed by the `extract_physical_constraints` command.

Figure 9-1 Incremental Extraction With the extract_physical_constraints Command



`dc_shell-topo> extract_physical_constraints DEF1.def DEF2.def`

To disable running the command in the incremental mode, specify the `-no_incremental` option with the `extract_physical_constraints` command. In the non-incremental mode, the placement area is extracted based on the site rows in the DEF file. If there are no site rows in the DEF file, other constraints are not extracted and the following physical constraints are reset:

- Core area
- Route guide
- Layer preferred routing direction
- Voltage area
- [Die Area](#)
- [Placement Area](#)
- [Macro Location and Orientation](#)
- [Hard, Soft, and Partial Placement Blockages](#)
- [Wiring Keepouts](#)

- Placement Bounds
- Port Location
- Preroutes
- Site Array Information
- Vias
- Routing Tracks
- Keepout Margins

To learn about the imported DEF physical constraints and port and macro name matching considerations, see

- [Physical Constraints Imported in the DEF File](#)
- [Extracting Physical-Only Cells From a DEF File](#)
- [Macro and Port Name Matching With the extract_physical_constraints Command](#)
- [Site Name Matching](#)

Physical Constraints Imported in the DEF File

The following high-level floorplan physical constraints can be imported into Design Compiler in topographical mode from a DEF file with the `extract_physical_constraints` command:

- Die Area
- Placement Area
- Macro Location and Orientation
- Hard, Soft, and Partial Placement Blockages
- Wiring Keepouts
- Placement Bounds
- Port Location
- Preroutes
- Site Array Information
- Vias
- Routing Tracks

- [Keepout Margins](#)

You can visually inspect your extracted physical constraints by using the layout view in the Design Vision layout window. All physical constraints extracted from the DEF file are automatically added to the layout view.

Note:

Voltage areas are not defined in a DEF file. Therefore, for multivoltage designs, you have to use the `create_voltage_area` command to define voltage areas for the tool. If you are using IC Compiler as your floorplanning tool, you can use the `write_floorplan` and `read_floorplan` commands to obtain floorplan information that automatically includes voltage areas. You do not need to define voltage areas manually when using these commands.

Die Area

The `extract_physical_constraints` command automatically extracts the die area from a DEF file. DEF files are generated by floorplanning tools and used by Design Compiler to import physical constraints. The tool can extract both rectangular and rectilinear die areas that are defined in the DEF file. The die area is also known as the cell boundary. The die area represents the silicon boundary of a chip and encloses all objects of a design, such as pads, I/O pins, and cells.

The following example shows a die area definition in a DEF file:

```
DEF
  UNITS DISTANCE MICRONS 1000 ;
  DIEAREA ( 0 0 ) ( 0 60000 ) ( 39680 60000 ) ( 39680 40000 ) \
            ( 59360 40000 ) ( 59360 0 ) ;
```

The following example shows how Design Compiler in topographical mode translates the DEF definition into Tcl when you write out your physical constraints using the `write_floorplan -all` command:

```
create_die_area -polygon { { 0.000 0.000 } { 0.000 60.000 } \
                           { 39.680 60.000 } { 39.680 40.000 } { 59.360 40.000 } \
                           { 59.360 0.000 } { 0.000 0.000 } }
```

Placement Area

Placement area is computed as the rectangular bounding box of the site rows.

Macro Location and Orientation

When you use the `extract_physical_constraints` command, for each cell with a location and the `FIXED` attribute specified in the DEF, Design Compiler sets the location on the corresponding cell in the design. [Example 9-2](#) shows DEF macro location and orientation information.

Note:

E = east rotation and W = west rotation

Example 9-2 DEF Macro Location and Orientation Information

```
COMPONENTS 2 ;
  - macro_cell_abx2 + FIXED ( 4350720 8160 ) E ;
  - macro_cell_cdy1 + FIXED ( 4800 8160 ) W ;
END COMPONENTS
```

The Tcl equivalent commands are shown in [Example 9-3](#).

Example 9-3 Tcl Equivalent Macro Location and Orientation Information

```
set_cell_location macro_cell_abx2 -coordinates { 4350.720 8.160 } \
    -orientation E -fixed
set_cell_location macro_cell_cdy1 -coordinates { 4.800 8.160 } \
    -orientation W -fixed
```

Hard, Soft, and Partial Placement Blockages

The `extract_physical_constraints` command can import hard, soft, and partial placement blockages defined in the DEF file.

[Example 9-4](#) shows DEF hard placement blockage information.

Example 9-4 DEF Hard Placement Blockage Information

```
BLOCKAGES 50 ;
...
  - PLACEMENT RECT ( 970460 7500 ) ( 3247660 129940 )
...
END BLOCKAGES
```

The Tcl equivalent command is shown in [Example 9-5](#).

Example 9-5 Tcl Equivalent Hard Placement Blockage Information

```
create_placement_blockage -name def_obstruction_23 \
    -bbox { 970.460 7.500 3247.660 129.940 }
```

For a soft placement blockage, if your extracted DEF information is as shown in [Example 9-6](#) (DEF version 5.6) or [Example 9-7](#) (DEF version 5.7), then the Tcl equivalent command is shown in [Example 9-8](#).

Note:

If your floorplanning tool creates a DEF file with DEF version 5.6, you need to manually add the `#SNPS_SOFT_BLOCKAGE` directive to specify a soft blockage, as shown in [Example 9-6](#).

Example 9-6 DEF Version 5.6 Soft Placement Blockage Information

```
BLOCKAGES 50 ;
...
```

```
- PLACEMENT RECT ( 970460 7500 ) ( 3247660 129940 ) ; #SNPS_SOFT_BLOCKAGE
...
END BLOCKAGES
```

Example 9-7 DEF Version 5.7 Soft Placement Blockage Information

```
BLOCKAGES 50 ;
...
- PLACEMENT + SOFT RECT ( 970460 7500 ) ( 3247660 129940 ) ;
...
END BLOCKAGES
```

Example 9-8 Tcl Equivalent Soft Placement Blockage Information

```
create_placement_blockage -name def_obstruction_23 \
    -bbox { 970.460 7.500 3247.660 129.940 } \
    -type soft
```

[Example 9-9](#) shows DEF partial placement blockage information. The Tcl equivalent command is shown in [Example 9-10](#).

Note:

DEF versions prior to version 5.7 did not support partial blockages.

Example 9-9 DEF Partial Placement Blockage Information

```
BLOCKAGES 50 ;
...
- PLACEMENT + PARTIAL 80 RECT ( 970460 7500 ) ( 3247660 129940 ) ;
...
END BLOCKAGES
```

Example 9-10 Tcl Equivalent Partial Placement Blockage Information

```
create_placement_blockage -name def_obstruction_23 \
    -bbox { 970.460 7.500 3247.660 129.940 } \
    -type partial \
    -blocked_percentage 80
```

Wiring Keepouts

For wiring keepouts defined in the DEF, Design Compiler creates wiring keepouts on the design.

[Example 9-11](#) shows DEF wiring keepout information.

Example 9-11 DEF Wiring Keepout Information

```
BLOCKAGES 30 ;
...
- LAYER METAL6 RECT ( 0 495720 ) ( 4050 1419120 );
...
END BLOCKAGES
```

Placement Bounds

If REGIONS defining bounds exist in the DEF, `extract_physical_constraints` imports placement bounds. Also, if there are cells in the related GROUP attached to the region, these cells are matched with the ones in the design using rule-based name matching, and the matched cells are attached to the bounds in the following two ways:

- If there are regions in the design with the same name as in the DEF, the cells in the related group are attached to the region by the `update_bounds` command in incremental mode.
- If the region does not exist in the design, it is created with the same name as in the DEF file by applying the `create_bounds` command; matched cells in the related group are also attached.

[Example 9-12](#) shows imported placement bounds information.

Example 9-12 DEF Placement Bounds Information

```
REGIONS 1 ;
- c20_group ( 201970 40040 ) ( 237914 75984 ) + TYPE FENCE ;
END REGIONS

GROUPS 1 ;
- c20_group
  cell_abcl
  cell_sm1
  cell_sm2
+ SOFT
+ REGION c20_group ;
END GROUPS
```

The Tcl equivalent commands are shown in [Example 9-13](#).

Example 9-13 Tcl Equivalent Placement Bounds Information

```
create_bounds \
  -name "c20_group" \
  -coordinate {201970 40040 237914 75984} \
  -exclusive \
  {cell_abcl cell_sm1 cell_sm2}
```

Port Location

When you use the `extract_physical_constraints` command, for each port with the location specified in the DEF, Design Compiler sets the location on the corresponding port in the design.

[Example 9-14](#) shows imported port location information.

Example 9-14 DEF Port Information

```
PINS 2 ;
-Out1 + NET Out1 + DIRECTION OUTPUT + USE SIGNAL +
  LAYER M3 (0 0) (4200 200) + PLACED (80875 0) N;
```

```

-Sel0 + NET Sel0 + DIRECTION INPUT + USE SIGNAL +
    LAYER M4 ( 0 0 ) (200 200) + PLACED (135920 42475) N;
END PINS

```

The Tcl equivalent commands are shown in [Example 9-15](#).

Example 9-15 Tcl Equivalent Port Information

```

create_terminal -name Out1 -port Out1 -layer M3 -bbox {{80.875 0} {85.075 0.200}}
create_terminal -name Sel0 -port Sel0 -layer M4 \
    -bbox {{135.920 42.475} {136.120 42.675}}

```

Ports with changed names and multiple layers are supported. [Example 9-16](#) shows DEF information for such a case.

Example 9-16 DEF Port Information

```

PINS 2 ;
- sys_addr\[23\].extra2 + NET sys_addr[23] + DIRECTION INPUT + USE SIGNAL +
    LAYER METAL4 ( 0 0 ) ( 820 5820 ) + FIXED ( 1587825 2744180 ) N ;
- sys_addr[23] + NET sys_addr[23] + DIRECTION INPUT + USE SIGNAL + LAYER
    METAL3 ( 0 0 ) ( 820 5820 ) + FIXED ( 1587825 2744180 ) N ;
END PINS

```

The corresponding Tcl commands are shown in [Example 9-17](#).

Example 9-17 Tcl Equivalent Port Information

```

create_terminal -name {sys_addr[23]} -port {sys_addr[23]} -layer METAL3 \
    -bbox {{1587.825 2744.180} {1588.645 2750.000}}
create_terminal -name {sys_addr[23]_1} -port {sys_addr[23]} -layer METAL4 \
    -bbox {{1587.825 2744.180} {1588.645 2750.000}}

```

Port orientation is also supported. [Example 9-18](#) shows DEF information for such a case.

Example 9-18 DEF Port Information

```

PINS 1;
- OUT + NET OUT + DIRECTION INPUT + USE SIGNAL
    + LAYER m4 ( -120 0 ) ( 120 240 )
    + FIXED ( 4557120 1726080 ) S ;
END PINS

```

The corresponding Tcl commands are shown in [Example 9-19](#).

Example 9-19 Tcl Equivalent Port Information

```

create_terminal -name OUT -port OUT -layer m4 \
    -bbox {{4557.000 1725.840} {4557.240 1726.080}}

```

Preroutes

Design Compiler extracts preroutes that are defined in the DEF file.

[Example 9-20](#) shows imported preroute information.

Example 9-20 DEF Preroute Information

```
SPECIALNETS 2 ;
- vdd
+ ROUTED METAL3 10000 + SHAPE STRIPE ( 10000 150000 ) ( 50000 * )
+ USE POWER ;
...
END SPECIALNETS
```

The Tcl equivalent commands are shown in [Example 9-21](#).

Example 9-21 Tcl Equivalent Preroute Information

```
create_net_shape -no_snap -type path -net vdd -datatype 0 -path_type 0 \
    -route_type pg_strap -layer METAL3 -width 10.000 \
    -points {{10.000 150.000} {50.000 150.000}}
```

Site Array Information

Design Compiler imports site array information that is defined in the DEF file. Site arrays in the DEF file define the placement area.

[Example 9-22](#) shows imported site array information.

Example 9-22 DEF Site Array Information

```
ROW ROW_0 core 0 0 N DO 838 BY 1 STEP 560 0;
```

The Tcl equivalent commands are shown in [Example 9-23](#).

Example 9-23 Tcl Equivalent Site Array Information

```
create_site_row -name ROW_0 -coordinate {0.000 0.000} \
    -kind core -orient 0 -dir H -space 0.560 -count 838
```

Vias

The `extract_physical_constraints` command extracts vias that are defined in the DEF file. Vias are stored in the .ddc file in the same way as other physical constraints.

[Example 9-24](#) shows how Design Compiler in topographical mode translates the DEF definition into Tcl when you write out your physical constraints using the `write_floorplan -all` command.

Example 9-24 Tcl Equivalent Via Information

```
create_via -type via -net VDD -master VIA67 -route_type pg_strap \
    -at {746.61 2279} -orient N
create_via -type via_array -net VDD -master FATVIA45 -route_type pg_strap \
    -at {1491.79 2127.8} -orient N -col 5 -row 2 -x_pitch 0.23 -y_pitch 0.23
```

See Also

- [Creating Vias](#)

Routing Tracks

The `extract_physical_constraints` command extracts any track information that is defined in the DEF file. Tracks define the routing grid for standard cell-based designs. They can be used during routing, and track support can enhance congestion evaluation and reporting in Design Compiler in topographical mode to make congestion routing more precise and match more closely with IC Compiler or IC Compiler II. Track information is stored in the .ddc file in the same way as other physical constraints. If you have a floorplan with track information, such as track type and location, the track information is passed to IC Compiler or IC Compiler II during floorplan exploration.

The following example shows track data in a DEF file:

```
TRACKS X 330 DO 457 STEP 660 LAYER METAL1 ;
TRACKS Y 280 DO 540 STEP 560 LAYER METAL1 ;
```

[Example 9-25](#) shows how Design Compiler in topographical mode translates the DEF definition into Tcl when you write out your physical constraints using the `write_floorplan -all` command.

Example 9-25 Tcl Equivalent Track Information

```
create_track \
    -layer metal1 \
    -dir X \
    -coord 0.100 \
    -space 0.200 \
    -count 11577 \
    -bounding_box {{0.000 0.000} {2315.400 2315.200}}
create_track \
    -layer metal1 \
    -dir Y \
    -coord 0.200 \
    -space 0.200 \
    -count 11575 \
    -bounding_box {{0.000 0.000} {2315.400 2315.200}}
```

See Also

- [Creating Routing Tracks](#)

Keepout Margins

The `extract_physical_constraints` command extracts keepout margins that are defined in the DEF file. Keepout margins are stored in the .ddc file in the same way as other physical constraints.

The following example shows a keepout margin definition in a DEF file:

```
COMPONENTS 2 ;
- U542 OAI21XL + FIXED ( 80000 80000 ) FN + HALO 10000 10000 50000 50000 ;
- U543 OAI21XL + FIXED ( 10000 20000 ) FN + HALO SOFT 15000 15000 15000 15000 ;
END COMPONENTS
```

See Also

- [Creating Keepout Margins](#)

Extracting Physical-Only Cells From a DEF File

To extract physical-only cells from a DEF file, use the `extract_physical_constraints` command with the `-allow_physical_cells` option.

See Also

- [Including Physical-Only Cells](#)

Macro and Port Name Matching With the `extract_physical_constraints` Command

When the `extract_physical_constraints` command applies physical constraints, it automatically matches macros and ports in the DEF file with macros and ports in memory. When it does not find an exact match, it uses the rule-based name-matching capability.

Name mismatches can be caused by automatic ungrouping and the `change_names` command. Typically, hierarchy separators and bus notations are sources of these mismatches.

For example, automatic ungrouping by the `compile_ultra` command followed by the `change_names` command might result in the forward slash (/) separator being replaced with an underscore (_) character. Therefore, a macro named a/b/c/macro_name in the RTL might be named a/b_c_macro_name in the mapped netlist, which is the input to the back-end tool. The `extract_physical_constraints` command automatically resolves these name differences when extracting physical constraints from the DEF file.

By default, the following characters are considered equivalent:

- Hierarchical separators { / _ . }

For example, a cell named a_b_c/d_e is automatically matched with the string a/b_c/d/e in the DEF file.

- Bus notations { [] __ () }

For example, a cell named a[4][5] is automatically matched with the string a_4__5_ in the DEF file.

When you use the `-verbose` option with the `extract_physical_constraints` command, the tool displays an informational message indicating that rule-based name matching has matched the cell with an instance in the DEF file.

You can define the rules used by the rule-based name-matching capability by using the `set_query_rules` command. Use the `set_query_rules -show` command to display the default query rules.

To disable rule-based name matching, specify the `-exact` option with the `extract_physical_constraints` command. In this case, the tool matches objects in the netlist in memory exactly with the corresponding objects in the DEF file.

See Also

- [Site Name Matching](#)

Site Name Matching

When you use the `extract_physical_constraints` command to read a DEF file, the tool automatically matches the site name in the floorplan with the name of the tile in the Milkyway reference libraries. Milkyway reference libraries usually have the site name set to unit by default. However, you might still need to define the name mappings if the site dimension does not match unit or if more than one site type is used in the DEF files. To define the name mappings, use the `mw_site_name_mapping` variable. For example,

```
dc_shell-topo> set mw_site_name_mapping { {def_site_name1 mw_ref_lib_site_name} \
{def_site_name2 mw_ref_lib_site_name} }
```

You can specify multiple pairs of values for the `mw_site_name_mapping` variable.

Using the `write_floorplan` Command in IC Compiler

To improve timing, area, and power correlation between Design Compiler and IC Compiler, you can read your mapped Design Compiler netlist into IC Compiler, create a basic floorplan in IC Compiler, export this floorplan from IC Compiler, and read the floorplan back into Design Compiler.

To export floorplan information from IC Compiler into a file that can be used in Design Compiler in topographical mode, use the `write_floorplan` command in IC Compiler. The command writes a Tcl script that you read into Design Compiler to re-create elements of the floorplan for the specified design.

[Example 9-26](#) uses the `write_floorplan` command to write out all placed standard cells to a file called `placed_std.fp`.

Example 9-26 Using write_floorplan to Export Physical Data From IC Compiler

```
icc_shell> write_floorplan -placement {io terminal hard_macro soft_macro}\
```

```
-create_terminal -row -create_bound \
-preroute -track floorplan_for_DC.fp
```

See Also

- [Reading the Floorplan Script in Design Compiler](#)

Reading the Floorplan Script in Design Compiler

The IC Compiler `write_floorplan` command generates a Tcl script that contains commands that describe floorplan information for the current design or a design that you specify. You can use this file to re-create elements of the floorplan in Design Compiler. Run the `read_floorplan` command in topographical mode to read the script generated by the `write_floorplan` command and restore the floorplan information.

The imported physical constraints are automatically saved to your `.ddc` file. To save the physical constraints in a separate file, use the `write_floorplan` command after extraction. The command saves the floorplan information so you can read the floorplan back into Design Compiler.

You can also use the `source` command to import the floorplan information. However, the `source` command reports errors and warnings that are not applicable to Design Compiler. The `read_floorplan` command removes these unnecessary errors and warnings. In addition, you need to enable rule-based name matching manually when you use the `source` command. The `read_floorplan` command automatically enables rule-based name matching.

The `read_floorplan` command should not be used to perform incremental floorplan modifications. The `read_floorplan` command imports the entire floorplan information exported from IC Compiler with the `write_floorplan` command and overwrites any existing floorplan. The `write_floorplan` command usually includes commands to remove any existing floorplan.

You can visually inspect your imported physical constraints by using the layout view in the Design Vision layout window.

To learn more about imported physical constraints and port and macro name matching considerations, see

- [Physical Constraints Imported in the Floorplan File](#)
- [Macro and Port Name Matching With the `read_floorplan` Command](#)

Physical Constraints Imported in the Floorplan File

The following physical constraints are imported from the floorplan file (Tcl script) when you run the `read_floorplan` command:

- Voltage areas

- Die Area

The die area represents the silicon boundary of a chip and encloses all objects of a design, such as pads, I/O pins, and cells.

- Placement Area

- Macro Location and Orientation

For each cell with a location and the FIXED attribute specified, Design Compiler sets the location on the corresponding cell in the design.

- Hard and Soft Placement Blockages

For defined placement blockages, Design Compiler creates placement blockages on the design.

- Wiring Keepouts

Wiring keepout information is imported from the floorplan file. The `create_route_guide` command creates a wiring keepout.

- Placement Bounds

Placement bounds are extracted from the floorplan file in the following two ways:

1. If there are regions in the design with the same name as in the floorplan file, the cells in the related group are attached to the region by the `update_bounds` command in incremental mode.

2. If the region does not exist in the design, it is created with the same name as in the floorplan file by applying the `create_bounds` command. Matched cells in the related group are also attached.

- Port Locations

For each port with the location specified in the floorplan file, Design Compiler sets the location on the corresponding port in the design.

Ports with changed names and multilayers are supported.

- Preroutes

Design Compiler imports preroutes that are defined in the floorplan file. A preroute is represented with the `create_net_shape` command.

- User Shapes

Design Compiler imports user shapes that are defined in the floorplan file and includes them in the preroute section.

- Site Array Information

Design Compiler extracts site array information that is defined in the floorplan file. Site arrays define the placement area.

- Vias

Design Compiler imports vias that are defined in the floorplan file and includes them in the preroute section.

- Tracks

Design Compiler imports any track information that is defined in the floorplan file.

Macro and Port Name Matching With the `read_floorplan` Command

When the `read_floorplan` command applies physical constraints in topographical mode, it automatically matches macros and ports in the floorplan file with macros and ports in memory. When it does not find an exact match, it uses the rule-based name-matching capability.

Name mismatches can be caused by automatic ungrouping and the `change_names` command. Typically, hierarchy separators and bus notations are sources of these mismatches.

For example, automatic ungrouping by the `compile_ultra` command followed by the `change_names` command might result in the forward slash (/) separator being replaced with an underscore (_) character. Therefore, a macro named a/b/c/macro_name in the RTL might be named a/b_c_macro_name in the mapped netlist, which is the input to the back-end tool. The `extract_physical_constraints` command automatically resolves these name differences when extracting physical constraints from the DEF file.

By default, the following characters are considered equivalent:

- Hierarchical separators { / _ . }

For example, a cell named a.b_c/d_e is automatically matched with the string a/b_c.d/e in the floorplan file.

- Bus notations { [] __ () }

For example, a cell named a[4][5] is automatically matched with the string a_4__5_ in the floorplan file.

To define the rules used by the rule-based name-matching capability, use the `set_query_rules` command.

Note:

Using the `source` command is not recommended. However, if you use the `source` command to import your floorplan information, you must enable rule-based name matching manually by setting the `enable_rule_based_query` variable to `true` before you source the floorplan file. For example,

```
dc_shell-topo> set_query_rules...
dc_shell-topo> set_app_var enable_rule_based_query true
dc_shell-topo> source design.fp
dc_shell-topo> set enable_rule_based_query false
```

Reading and Writing Preroute Information for Power and Ground Nets and Physical-Only Cells

The floorplan that is written from IC Compiler using the `write_floorplan` command can include preroute information and the location of physical-only cells. However, when the floorplan file is read into Design Compiler in topographical mode, the information is not used during optimization because the power and ground nets and physical cells do not exist in the logical netlist.

To use preroute information and physical-only cells in Design Compiler, you need to create the power and ground nets and physical-only cells before reading the floorplan file. When you run the `write_floorplan -preroute` command, Design Compiler writes out the preroute information and the location of the physical-only cells, and the command generates a secondary floorplan. This secondary floorplan file is generated with an `.objects` suffix added to the floorplan file name. The secondary floorplan file contains Tcl commands that the tool will use to create the power and ground nets and physical-only cells.

The following example creates the floorplan files, `floorplan.tcl` and `floorplan.tcl.objects`:

```
dc_shell-topo> write_floorplan -preroute floorplan.tcl
```

The following example shows a `floorplan.tcl.objects` file:

```
if {! [sizeof_collection [get_nets -quiet -all "VDD"]]} {
create_net -power VDD
}
if {! [sizeof_collection [get_nets -quiet -all "VSS"]]} {
create_net -ground VSS
}
if {! [sizeof_collection [get_cells -quiet -all "FIL_1"]]} {
create_cell -only_physical FIL_1 FILLERX1
}
```

In the UPF flow, the power and ground nets are typically specified in the UPF file. If you are using the UPF flow, you must read the UPF file before you read the floorplan files.

If your power and ground nets are not defined yet, it is important that you read the secondary floorplan file before you read the main floorplan file, as shown in [Example 9-27](#). When you read the secondary floorplan file, the tool creates the power and ground nets and physical-only cells in the logical netlist. When you read the main floorplan file, the tool applies the floorplan information for the power and ground nets and the physical-only cells to the corresponding objects. This allows the tool to use the preroute information for the power and ground nets and physical-only cells during optimization.

Example 9-27 Creating and Applying Power and Ground Nets and Physical-Only Cells

```
dc_shell-topo> read_floorplan floorplan.tcl.objects
1
dc_shell-topo> read_floorplan floorplan.tcl
Information: Successfully applied 2 of the 2 'create_net_shape'
commands read. (DCT-143)
1
```

Manually Defined Physical Constraints

If you do not provide any physical constraints from a floorplanning tool, either in a DEF file or in a Tcl script, Design Compiler uses the following default physical constraints:

- Aspect ratio of 1.0 (that is, a square placement area)
- Utilization of 0.6 (that is, forty percent of empty space in the core area)

You can also manually define physical constraints as described in the following topics:

- [Defining Physical Constraints Overview](#)
- [Defining the Die Area](#)
- [Defining the Core Placement Area With the create_site_row Command](#)
- [Defining Placement Area With the set_aspect_ratio and set_utilization Commands](#)
- [Defining Port Locations](#)
- [Defining Macro Location and Orientation](#)
- [Defining Placement Blockages](#)
- [Defining Voltage Areas](#)
- [Defining Placement Bounds](#)
- [Creating Wiring Keepouts](#)
- [Creating Preroutes](#)
- [Creating User Shapes](#)
- [Defining Physical Constraints for Pins](#)
- [Creating Design Via Masters](#)
- [Creating Vias](#)
- [Creating Routing Tracks](#)
- [Creating Keepout Margins](#)

- Computing Polygons

Defining Physical Constraints Overview

This topic describes the commands you can use to define the floorplan's physical constraints manually. Design Compiler in topographical mode can read physical constraints (floorplan information) from a Tcl-based script of physical commands.

You manually define physical constraints when you cannot obtain this information from a DEF file using the `extract_physical_constraints` command or from a Tcl script using the `read_floorplan` command. For details, see [Reading DEF Information in Design Compiler](#) and [Reading the Floorplan Script in Design Compiler](#).

After you have manually defined your physical constraints in a Tcl script file, use the `source` command to apply these constraints. Keep the following points in mind when you manually define physical constraints:

- You must read in the design before applying user-specified physical constraints.
- You must apply user-specified physical constraints during the first topographical mode session.

[Table 9-1](#) lists the commands that you use to set physical constraints.

Table 9-1 Commands That Define Physical Constraints

To define this physical constraint	Use these commands
Die Area For details, see Defining the Die Area .	<code>create_die_area</code>
Floorplan estimate when exact area is not known For details, see Defining Placement Area With the set_aspect_ratio and set_utilization Commands .	<code>set_aspect_ratio</code> and <code>set_utilization</code>
Exact core area For details, see Defining the Core Placement Area With the create_site_row Command .	<code>create_site_row</code>
Relative port locations For details, see Defining Relative Port Locations .	<code>set_port_side</code>
Exact port locations For details, see Defining Exact Port Locations .	<code>create_terminal</code>

Table 9-1 Commands That Define Physical Constraints (Continued)

To define this physical constraint	Use these commands
Macro location and orientation For details, see Defining Macro Location and Orientation .	<code>set_cell_location</code>
Placement keepout (blockages) For details, see Defining Placement Blockages .	<code>create_placement_blockage</code>
Voltage area For details, see Defining Voltage Areas .	<code>create_voltage_area</code>
Placement bounds For details, see Defining Placement Bounds .	<code>create_bounds</code>
Wiring keepouts For details, see Creating Wiring Keepouts .	<code>create_route_guide</code>
Preroutes For details, see Creating Preroutes .	<code>create_net_shape</code>
User shapes For details, see Creating User Shapes .	<code>create_user_shape</code>
Pin physical constraints For details, see Defining Physical Constraints for Pins .	<code>set_pin_physical_constraints</code> <code>create_pin_guide</code>
Design via masters For details, see Creating Design Via Masters .	<code>create_via_master</code>
Vias For details, see Creating Vias .	<code>create_via</code>
Tracks For details, see Creating Routing Tracks .	<code>create_track</code>
Keepout margins For details, see Creating Keepout Margins .	<code>set_keepout_margin</code>
Polygons For details, see Computing Polygons .	<code>compute_polygons</code>

Defining the Die Area

Design Compiler topographical technology allows you to manually define the die area, also known as the cell boundary. The die area represents the silicon boundary of a chip, and it encloses all objects of a design, such as pads, I/O pins, and cells. There should be only one die area in a design.

Typically, you create floorplan constraints, including the die area, in your floorplanning tool, and import these physical constraints into Design Compiler topographical mode. However, if you are not using a floorplanning tool, you need to manually create your physical constraints. The `create_die_area` command allows you to define your die area from within Design Compiler in topographical mode.

When using the `create_die_area` command, you can use the `-coordinate` option if your die area is a rectangle. If your die area is rectilinear, you must use the `-polygon` option. You can use the `-polygon` option to specify rectangles, but you cannot use the `-coordinate` option to specify rectilinear die areas. For example, the following commands create the same rectangular die area:

```
create_die_area -coordinate { 0 0 100 100 }
create_die_area -coordinate { {0 0} {100 100} }
create_die_area -polygon { {0 0} {100 0} {100 100} {0 100} }
```

You can use the `get_die_area` command to return a collection containing the die area of the current design. If the die area is not defined, the command returns an empty string.

Use the `get_attribute` command to get information about the die area, such as the object class, bounding box information, coordinates of the current design's die area, die area boundary, and die area name.

You can use the `report_attribute` command to browse attributes, such as `object_class`, `bbox`, `boundary`, and `name`.

For example, you can create a new die area, as shown:

```
dc_shell-topo> create_die_area -polygon {{0 0} {0 400} {200 400} {200 200} \
{400 200} {400 0}}
```

Then, use the `get_attribute` command to get the following information about the die area:

- To return the object class, use the `object_class` attribute, as shown:

```
dc_shell-topo> get_attribute [get_die_area] object_class
die_area
```

- To return the bounding box information and the coordinates of the current design's die area, use the `bbox` attribute, as shown:

```
dc_shell-topo> get_attribute [get_die_area] bbox
{0.000 0.000} {400.000 400.000}
```

- To return the die area boundary, use the `boundary` attribute, as shown:

```
dc_shell-topo> get_attribute [get_die_area] boundary
{0.000 0.000} {0.000 400.000} {200.000 400.000} {200.000 200.000}
{400.000 200.000} {400.000 0.000} {0.000 0.000}
```

- To return the die area name, use the `name` attribute, as shown:

```
dc_shell-topo> get_attribute [get_die_area] name
{my_design_die_area}
```

Defining the Core Placement Area With the `create_site_row` Command

You can define the core placement area with the `create_site_row` command. The core placement area is a box that contains all rows. It represents the placeable area for standard cells. The core area is smaller than the cell boundary (the die area). Pads, I/O pins, and top-level power and ground rings are found outside the core placement area. Standard cells, macros, and wire tracks are typically found inside the core placement area. There should be only one core area in a design.

You use the `create_site_row` command to create a row of sites or a site array at a specified location. A site is a predefined valid location where a leaf cell can be placed; a site array is an array of placement sites and defines the placement core area.

To create a horizontal row of 100 CORE_2H sites with a bottom-left corner located at (10,10) and rows spaced 5 units apart, enter

```
dc_shell-topo> create_site_row -count 100 -kind CORE_2H -space 5 \
-coordinate {10 10}
```

To report physical information, such as core area, aspect ratio, utilization, total fixed cell area, and total movable cell area, use the `report_area -physical` command.

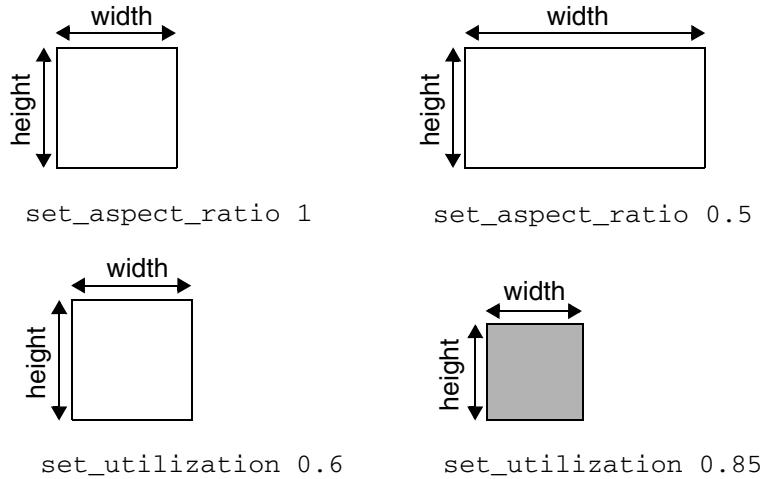
Defining Placement Area With the `set_aspect_ratio` and `set_utilization` Commands

If you have not defined your die area with the `create_die_area` command, defined a floorplan area with the `create_site_row` command, or imported floorplan information from a floorplanning tool, you can use the `set_aspect_ratio` and `set_utilization` commands to estimate the placement area.

The aspect ratio is the height-to-width ratio of a block; it defines the shape of a block. Utilization specifies how densely you want cells to be placed within the block. Increasing utilization reduces the core area.

[Figure 9-2](#) illustrates how to use these commands.

Figure 9-2 Using the set_aspect_ratio and set_utilization Commands



Defining Port Locations

You can define constraints that restrict the placement and sizing of ports by using the `set_port_side` command or `create_terminal` command. You can specify relative or exact constraints.

To learn how to define relative and exact port locations, see

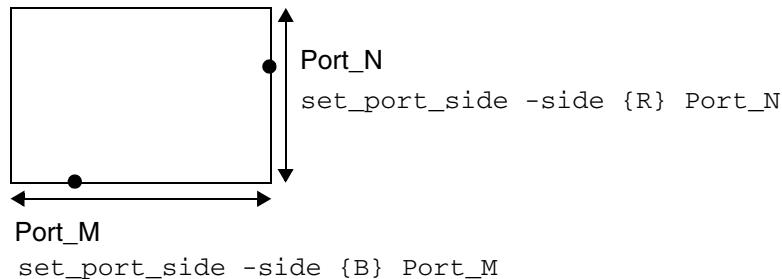
- [Defining Relative Port Locations](#)
- [Defining Exact Port Locations](#)

Defining Relative Port Locations

Use the `set_port_side` command to define relative port locations as follows:

```
set_port_side port_name -side {L|R|T|B}
```

Valid sides are left (L), right (R), top (T), or bottom (B). A port can be placed at any location along the specified side. If the port side constraints are provided, the ports are snapped to the specified side. Otherwise, by default, the ports are snapped to the side nearest to the port location assigned by the coarse placer. [Figure 9-3](#) shows how you define port sides by using the `set_port_side` command.

Figure 9-3 Setting Relative Port Sides

Defining Exact Port Locations

Use the `create_terminal` command to create a terminal for the specified logical port, specifying the bounding box of the terminal as follows:

```
dc_shell-topo> create_terminal -bbox {x1 y1 x2 y2} \
-layer layer_name -port port_name
```

The `-bbox` option specifies the bounding box of the terminal; the `-layer` option specifies the layer of the terminal; and the `-port` option specifies the name of the associated logical port of the terminal.

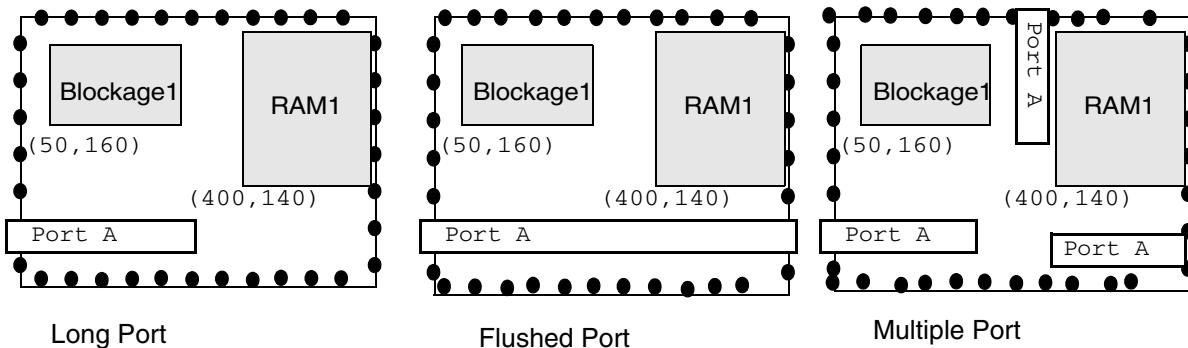
The following example creates a terminal for the Z port, a top-level port in the current design, with a bounding box of {0 0 100 500} on metal layer METAL1:

```
dc_shell-topo> create_terminal -port Z -bbox {0 0 100 500} -layer METAL1
```

These options allow you to define long ports so that connections to the long port can be made along any point on the specified metal layer during virtual-layout based optimization. A long port can be an input, output, or bidirectional port. You specify a long port location by using a dimension greater than the minimum metal area. [Figure 9-4](#) illustrates the different types of port dimensions: Long port, flushed port, and multiple port.

- A long port has the metal area defined with a single metal layer and touches only one side of the block core area boundary.
- A flushed port has the metal area defined with a single metal layer and touches the block core area boundary on two sides.
- A multiple port has more than one connection to the block and can touch the block boundary on two or more sides and can use one or more layers.

For multiple terminals associated with the same port, the first terminal has the same name as the port and subsequent terminal names have the format `port_name_number` where `port_name` is the port name and `number` is a unique number.

Figure 9-4 Types of Port Dimension Specifications

Defining Macro Location and Orientation

You can define exact macro locations by setting the following command:

```
set_cell_location cell_name -coordinates {x y} -fixed
                      -orientation {N|S|E|W|FN|FS|FE|FW}
```

where the `-coordinates` option specifies the coordinate of the lower-left corner of the cell's bounding box. Specify the coordinates in microns relative to the block orientation. The orientation value is one of the rotations listed in the following table.

Orientation	Rotation
N (default)	Nominal orientation, 0-degree rotation (north)
S	180 degrees (south)
E	270 degrees counterclockwise (east)
W	90 degrees counterclockwise (west)
FN	Reflection in the y-axis (flipped north)
FS	180 degrees, followed by reflection in the y-axis (flipped south)
FE	270 degrees counterclockwise, followed by reflection in the y-axis (flipped east)
FW	90 degrees counterclockwise, followed by reflection in the y-axis (flipped west)

Defining Placement Blockages

Use the `create_placement_blockage` command to define placement blockages. The `-bbox` option specifies the coordinates of the bounding box of the blockage. Use the `-type` option to specify the type of placement blockage to be created—hard, soft, or partial.

The `-type` values are described as follows:

- A `hard` placement blockage prevents the placer from placing any standard cells or hard macros in the specified region. This is the `-type` option default.
- A `soft` placement blockage prevents the coarse placer from placing any standard cells or hard macros in the specified region. However, cells might be placed there during optimization or legalization.
- A `partial` placement blockage limits the amount of area used to place cells to the specified percent of the blockage area. When you create a partial blockage, you must use the `-blocked_percentage` option to specify the blocked percentage value.

During coarse placement, Design Compiler can evenly distribute cells outside the partial blockages and prevent the clumping of cells around their perimeters. To do this, set the `placer_enable_redefined_blockage_behavior` variable to `true` if your design contains partial blockages created by using the `-type partial` option.

You can define a blockage area where only buffers or inverters can be placed by specifying the `-buffer_only` option with the `-type partial` option. You must set the `placer_enable_redefined_blockage_behavior` variable to `true` when using the `-buffer_only` option.

The following example shows a blockage created with the `-buffer_only` option. The blocked percentage is 50 percent:

```
dc_shell-topo> create_placement_blockage -buffer_only -name my_blockage \
    -type partial -blocked_percentage 50 -bbox {{0 0} {100 100}}
```

The `-buffer_only` option cannot be used with the `-type hard`, `-type soft`, `-no_hard_macro`, `-no_rp_group`, `-no_register`, `-no_pin`, or `-category` options.

You can specify either to include specific library cells or cell instances in a blockage area or to exclude them from a blockage area by using the `-category` option with the `-type partial` option. You must set the `placer_enable_redefined_blockage_behavior` variable to `true` when using the `-category` option.

To create a predefined category of cells, you must first define an attribute for the library cells or cell instances by using the `define_user_attribute` command. Then apply the attribute to the desired library cells or cell instances by using the `set_attribute` command. After you do this, you can create a category blockage by using the attribute name as the argument to the `-category` option.

To prevent library cells or cell instances from being placed within the specified blockage area, set your user-defined attribute to `true` for the specific library cells or cell instances. By default, all cells are allowed within the specified blockage area. This is equivalent to the user-defined attribute set to `false`.

In the following example, Design Compiler blocks all cells except isolation (`bfiso*`) cells:

```
dc_shell-topo> define_user_attribute -type boolean -classes lib_cell only_iso
dc_shell-topo> set_attribute [get_lib_cell mylib/*] only_iso true
dc_shell-topo> set_attribute [get_lib_cell mylib/bfiso*] only_iso false
dc_shell-topo> create_placement_blockage -bbox {{230 795} {830 1940}} \
    -name blk_VA_edge -type partial -blocked_percentage 20 -category only_iso
```

Each `cell` or `lib_cell` class can have multiple user attributes defined for it, controlling the library cell or cell instance behavior in multiple regions. If a cell instance has attributes defined for it and its library cell has different attributes defined for it, the cell instance's attribute takes precedence.

User attributes defined on cell instances are saved when the design is written in `.ddc` format; however, user attributes defined on library cells are not saved when the design is written. You need to re-create the `lib_cell` user-attribute and reapply it to the library cell if you reload your design in another session.

Multiple blockages can use the same attribute. However, each blockage can accept only one attribute. The `-blocked_percentage` option setting affects all cells in the blockage area, regardless of the attribute.

The `-category` option cannot be used with the `-type hard`, `-type soft`, `-no_hard_macro`, `-no_rp_group`, `-no_register`, `-no_pin`, or `-buffer_only` options.

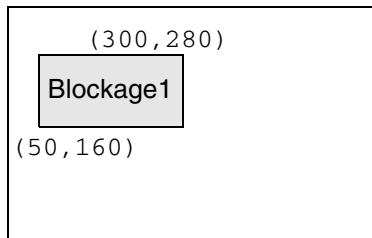
Use the `-no_rp_group` option to specify relative placement group blockages. This option prevents the placement of relative placement groups within the specified area. The option only affects cells that belong to relative placement groups. The `-no_rp_group` option must be used with the `-type partial` option, and it cannot be used with the `-no_hard_macro`, `-no_register`, and `-no_pin` options.

To restrict Design Compiler from placing any register cells within the specified partial blockage area during coarse placement, specify the `-no_register` option with the `-type partial` option. However, this restriction does not apply to optimization performed after coarse placement. If you specify the `-retime` option with the `compile_ultra` command, the registers that are moved during optimization can be placed in this area.

Figure 9-5 uses the `create_placement_blockage` command to define a hard placement blockage named Blockage1.

Figure 9-5 Defining a Hard Placement Blockage

```
create_placement_blockage\  
-name Blockage1 -type hard \  
-bbox {50 160 300 280}
```



You can use the `get_placement_blockages` command to return a collection of placement blockages from the current design. The command returns a collection of placement blockages if one or more blockages meet the specified criteria. If no blockages match the criteria, it returns an empty string. You can use the `get_placement_blockages` command as an argument to another command or assign its result to a variable.

To create a collection containing all blockages within a specified region, use the `-within region` option with the `get_placement_blockages` command. The region boundary can be a rectangle or a polygon. In the following example, Design Compiler returns one rectangular placement blockage based on the specified region. In the example, the first xy pair {2 2} represents the lower-left corner of the rectangle, and the second xy pair {25 25} represents the upper-right corner of the rectangle:

```
dc_shell-topo> get_placement_blockages * -within {{2 2} {25 25}}  
{ "PB#5389" }
```

To filter the collection with an expression, use the `-filter expression` option with the `get_placement_blockages` command. In the following example, Design Compiler returns all placement blockages that have an area greater than 900:

```
dc_shell-topo> get_placement_blockages * -filter "area > 900"  
{ "PB#4683" }
```

Defining Voltage Areas

You define voltage areas by using the `create_voltage_area` command. This command enables you to create a voltage area on the core area of the chip. The voltage area is associated with hierarchical cells. The tool assumes the voltage area to be an exclusive,

hard move bound and tries to place all the cells associated with the voltage area within the defined voltage area, as well as place all the cells not associated with the voltage area outside the defined voltage area.

[Example 9-28](#) uses the `create_voltage_area` command to constrain the instance `INST_1` to lie within the voltage area whose coordinates are lower-left corner (100 100) upper-right corner (200 200).

Example 9-28 Using the `create_voltage_area` Command With the `-name` Option

```
dc_shell-topo> create_voltage_area -name my_design \
    -coordinate {100 100 200 200} INST_1
```

[Example 9-29](#) uses the `create_voltage_area` command to constrain cells in power domain `PD1` to lie within the voltage area whose coordinates are lower-left corner (100 100) upper-right corner (200 200).

Example 9-29 Using the `create_voltage_area` Command With the `-power_domain` Option

```
dc_shell-topo> create_voltage_area -power_domain PD1 \
    -coordinate {100 100 200 200}
```

Note:

The `-name` option and the `-power_domain` option of the `create_voltage_area` command are mutually exclusive.

In addition to the `create_voltage_area` command, the following commands are supported: `remove_voltage_area` and `report_voltage_area`

Voltage areas are automatically defined when you import your floorplan information from IC Compiler or IC Compiler II with the `read_floorplan` command. For details, see [Reading the Floorplan Script in Design Compiler](#). You need to define voltage areas manually when importing floorplan information from a DEF file.

To visually inspect your defined voltage areas, use the visual mode in the Design Vision layout window.

Defining Placement Bounds

To learn how to create and use placement bounds, see

- [Placement Bounds Overview](#)
- [Creating Placement Bounds](#)
- [Order for Creating Placement Bounds](#)
- [Guidelines for Defining Placement Bounds Effectively](#)

- [Returning a Collection of Bounds](#)

Placement Bounds Overview

A placement bound is a rectangular or rectilinear area within which to place cells and hierarchical cells. You use the `create_bounds` command to specify placement constraints for coarse placement.

Defining a placement bound enables you to group cells such as clock-gating cells or extremely timing-critical groups of cells that you want to guarantee will not be disrupted during placement by other logic. During placement, the tool ensures that the cells you grouped remain together. Placement bounds are placement constraints.

Usually, it is best not to impose bounds constraints and instead allow the tool full flexibility to optimize placement for timing and routability. However, if QoR does not meet your requirements, you might improve QoR by using the `create_bounds` command.

To use placement bounds effectively, make the number of cells you define in placement bounds relatively small compared with the total number of cells in the design. When you define placement bounds, the solution space available to the placer to reach the optimal result gets smaller.

Creating Placement Bounds

Use the `create_bounds` command to specify placement bounds. You can specify two different types of bounds: move bounds and group bounds. Move bounds restrict the placement of cells to a specific region of the core area. Group bounds are floating region constraints. Cells in the same group bound are placed within a specified bound but the absolute coordinates are not fixed. Instead, they are optimized by the placer. For more information, see the following topics:

- [Move Bounds](#)
- [Group Bounds](#)
- [Repelling Bounds](#)

Move Bounds

Move bounds require absolute coordinates to be specified using the `-coordinate` option. Move bounds can be soft, hard, or exclusive:

- Soft bounds specify placement goals, with no guarantee that the cells will be placed inside the bounds. If timing or congestion cost is too high, cells might be placed outside the region. This is the default.

```
dc_shell-topo> create_bounds -coordinate {0 0 10 10} \
-name mb_soft INST2
```

- Hard bounds force placement of the specified cells inside the bounds. To specify hard bounds, use the `-type hard` option. Note that overusing hard bounds can lead to inferior placement solutions.

```
dc_shell-topo> create_bounds -type hard -coordinate {0 0 10 10} \
-name mb_hard INST4
```

- Exclusive move bounds force the placement of the specified cell inside the bounds. All other cells must be placed outside the bounds. To specify exclusive bounds, use the `-exclusive` option.

```
dc_shell-topo> create_bounds -exclusive -type hard \
-coordinate {0 0 10 10} -name mb_excl INST5
```

For the GUI, you can specify color for move bounds by using the following options with the `create_bounds` command:

- `-color`

Specify the move bound color either by specifying an integer value between 0 and 63 or by specifying a color name string with one of the following values: black, blue, green, cyan, brown, purple, red, magenta, salmon, orange, yellow, or white. The default is `no_color`, meaning that no color is applied.

- `-cycle_color`

The `-cycle_color` option allows the tool to automatically assign a move bound color. By default, this option is turned off.

Group Bounds

Group bounds are floating region constraints that you specify with the `-dimension` option set to `{width height}`. Cells in the same group bound are placed within the specified bound, but the absolute coordinates are not fixed. Instead, they are optimized by the placer. If you do not use the `-dimension` option, the tool creates a group bound with the bounding box computed internally by the tool.

Group bounds can be soft or hard:

- Soft bounds specify placement goals, with no guarantee that the cells will be placed inside the bounds. If timing or congestion cost is too high, cells might be placed outside the region. This is the default.

```
dc_shell-topo> create_bounds -dimension {100 100} -name gb_soft INST1
```

- Hard bounds force placement of the specified cells inside the bounds. To specify hard bounds, use the `-type hard` option with the `-dimension` option set to `{width height}`. Note that overusing hard bounds can lead to inferior placement solutions.

```
dc_shell-topo> create_bounds -type hard -dimension {100 100} \
-name gb_hard INST3
```

You can also create diamond bounds by using the `-diamond` option. A diamond bound is a soft group bound that constrains the placement of the cells associated with the bound to be within a specified Manhattan distance from a specified cell, port, or pin. In this case, specify a single value with the `-dimension` option.

The following example shows how to create a diamond bound to constrain the placement of the cells of the INST6 hierarchical instance so that they are placed within a 50-micron Manhattan distance from the data1 pin:

```
dc_shell-topo> create_bounds -diamond [get_pins data1] \
    -dimension 50 -name db_soft INST6
```

Repelling Bounds

Repelling bounds are similar to group bounds. However, instead of specifying a floating area where objects should be placed, a repelling bound specifies a floating area between a pair of objects to keep them apart in the bound.

To create a repelling bound, use the `-repelling` option with the `create_bounds` command. You can specify a diamond-shaped bound (`diamond`) or a rectangle-shaped bound (`rect`).

The following example creates a rectangular hard repelling bound with a height and width of 100:

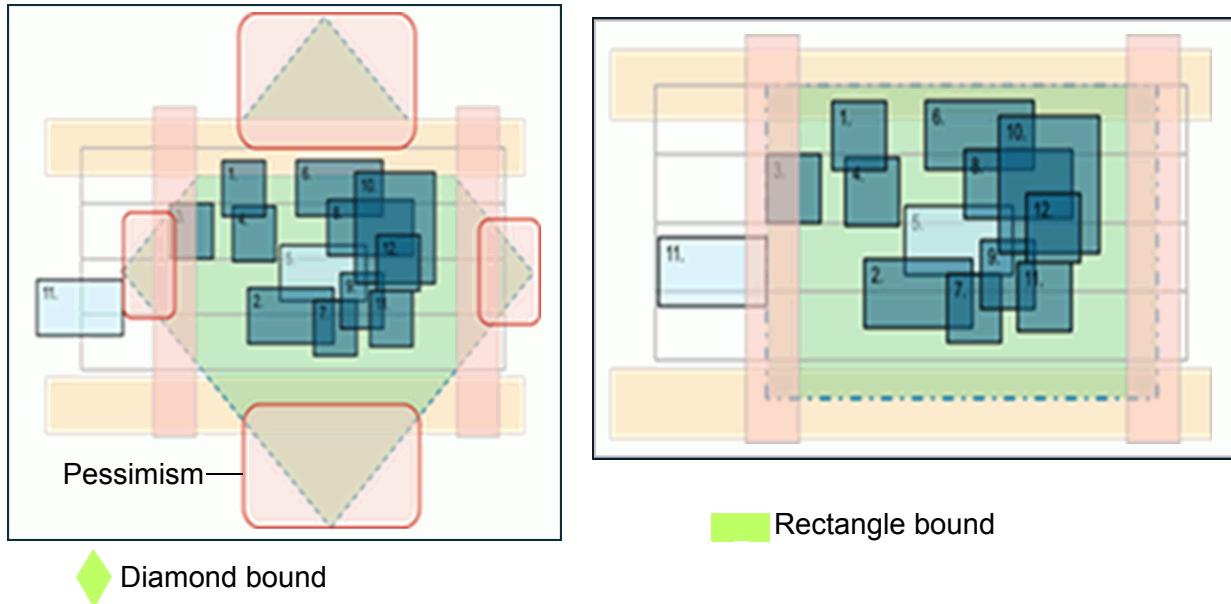
```
dc_shell-topo> create_bounds -repelling rect -dimension { 100 100 } \
    -type hard -name my_bound [get_cells {RG1 RG2}]
```

The following example creates a diamond soft repelling bound with a Manhattan length of 50:

```
dc_shell-topo> create_bounds -repelling diamond -dimension { 50 } \
    -type soft -name my_bound [get_cells {RG1 RG2}]
```

[Figure 9-6](#) shows how the bound contents are spread apart in the diamond-shaped and rectangle-shaped bound regions.

Figure 9-6 Diamond-Shaped Bound and Rectangle-Shaped Bound Regions



Diamond bound

Rectangle bound

To report bounds in the design, use the `report_bounds` command.

To hand off information to the following tools, write the repelling bound information to the `floorplan.tcl` script file in ASCII format by using the following commands:

- IC Compiler II: Use the `write_icc2_files` or `write_floorplan -format icc2` command.
- Another Design Compiler Graphical session: Use the `write_floorplan` command.

Order for Creating Placement Bounds

If you impose placement bounds constraints, create them in the following order:

1. Floating group bounds where the location and dimension are optimized by the tool. For these bounds, do not specify dimensions.
2. Floating group bounds with fixed dimensions that are defined by the `-dimension` option.
3. Fixed move bounds with a fixed location and dimension that are defined by the `-coordinate` option.

Guidelines for Defining Placement Bounds Effectively

Use the following guidelines when you create placement bounds:

- Use soft and hard move bounds sparingly in your design.

- Avoid placing cells in more than one bound.
- Be aware that including small numbers of fixed cells in group bounds can move the bound.
- Do not use placement bounds as keepouts.
- Maintain even cell density distribution over the chip.

Returning a Collection of Bounds

You can use the `get_bounds` command to return a collection of bounds from the current design based on the criteria you specify. You can use the `get_bounds` command at the command prompt, or you can nest it as an argument to another command, such as the `query_objects` or `report_bounds` command. In addition, you can assign the `get_bounds` result to a variable.

When issued from the command prompt, the `get_bounds` command behaves as though you have called the `query_objects` command to report the objects in the collection. By default, it displays a maximum of 100 objects. You can change this maximum by using the `collection_result_display_limit` variable.

In the following example, Design Compiler returns all bounds that have a name starting with `my_bound`:

```
dc_shell-topo> get_bounds my_bound*
```

Creating Wiring Keepouts

You can create wiring keepouts by using the `create_route_guide` command.

[Example 9-30](#) uses the `create_route_guide` command to create a keepout named `my_keepout_1` in the METAL1 layer at coordinates {12 12 100 100}.

Example 9-30

```
dc_shell-topo> create_route_guide -name "my_keepout_1" \
                    -no_signal_layers "METAL1" \
                    -coordinate {12 12 100 100}
```

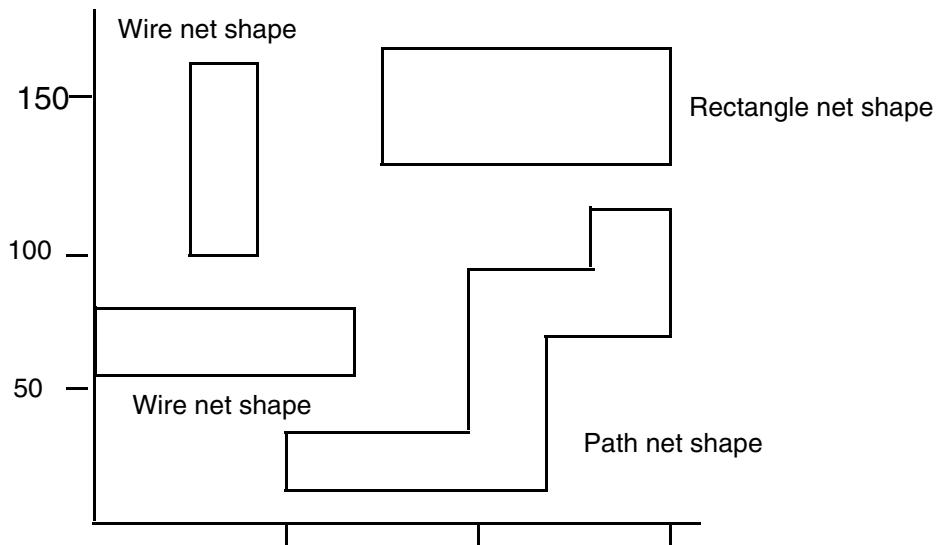
You can control the naming convention used by the `create_route_guide` command by using the `route_guide_naming_style` variable. The value must contain one %s and one %d in the character sequence, where %s indicates the user-defined route guide name and %d indicates an integer assigned by the tool to make the name unique. To use a percent sign in the route guide name, you must specify two of them in the string (%%). If the value of the variable is invalid, the tool uses the default naming convention, which is %s_%d. Using the variable helps to prevent issues caused by a conflict between a user route guide and the wiring keepouts that are automatically derived by the tool from other route guides.

Creating Preroutes

You can preroute a group of nets, such as clock nets, before routing the rest of the nets in the design. During global routing, the tool considers these preroutes while computing the congestion map; this map is consistent with IC Compiler and IC Compiler II. This feature also addresses correlation issues caused by inconsistent floorplan information.

To define preroutes, you use the `create_net_shape` command. You can create three different types of net shapes as shown in [Figure 9-7](#): path, wire (horizontal and vertical), and rectangle.

Figure 9-7 Types of Preroutes



The following examples show how to create the net shapes shown in [Figure 9-7](#).

[Example 9-31](#) uses the `create_net_shape` command to create two wire net shapes.

Example 9-31 Creating Two Wire Net Shapes

```
dc_shell-topo> create_net_shape -type wire -net VSS \
    -bbox {0 60 80 80} \
    -layer METAL5 \
    -route_type pg_strap \
    -net_type ground

dc_shell-topo> create_net_shape -type wire -net VSS \
    -origin {50 100} \
    -width 20 \
    -length 80 \
    -layer METAL4 -route_type pg_strap -vertical
```

[Example 9-32](#) uses the `create_net_shape` command to create the path net shape.

Example 9-32 Creating a Path Net Shape

```
dc_shell-topo> create_net_shape -type path -net vss \
    -points {70 30 110 30 110 90 150 90 150 110} \
    -width 0.20 -layer M3 \
    -route_type pg_std_cell_pin_conn
```

[Example 9-33](#) uses the `create_net_shape` command to create the rectangle net shape.

Example 9-33 Creating a Rectangle Net Shape

```
dc_shell-topo> create_net_shape -type rect -net vss \
    -bbox {{80 140} {160 180}} \
    -layer METAL1 -route_type pg_strap
```

Creating Preroutes for Power and Ground Nets

Design Compiler uses preroute data for congestion analysis during optimization only for preroutes that are created on nets that exist in the design. When preroute data is extracted from a DEF file by the `extract_physical_constraints` command, Design Compiler also extracts the power and ground nets from the DEF file and creates the preroutes on them.

Design Compiler takes the power and ground nets into account when you run the `compile_ultra` command. However, when preroutes are created by the `create_net_shape` command, you need to create the power and ground nets before creating the preroutes on them because the power and ground nets are not in the logical netlist.

You can create power and ground nets in Design Compiler in topographical mode by using the `create_net` command. This allows the tool to use the preroute information created by the `create_net_shape` command for the power and ground nets.

In the following example, the tool creates the power and ground nets and then creates preroutes on the power and ground nets:

```
dc_shell-topo> create_net -power vdd
1
dc_shell-topo> create_net -power vss
1
dc_shell-topo> create_net_shape -net vdd -points {{0 20} {80 20}} \
    -width 10 -layer M1 -route_type pg_ring -no_snap
Information: net shape with type path will be created during compile.
(DCT-020)
1
dc_shell-topo> create_net_shape -net vss -points {{20 0} {20 80}} \
    -width 10 -layer M3 -route_type pg_ring -no_snap
Information: net shape with type path will be created during compile.
(DCT-020)
1
```

When a preroute is created on a net that does not exist in the design, Design Compiler does not use the preroute data when you run the `compile_ultra` command. It removes the preroute data.

This behavior is consistent with the `place_opt` command in IC Compiler and IC Compiler II.

If you want to preserve preroute data for nets that do not exist in the design, set the `dct_preserve_all_preroutes` variable to `true` before you run the `compile_ultra` command. When this variable is set to `true`, Design Compiler does not remove the preroutes during optimization, but it does not use the preroute data on the non-existing net during optimization. The default for the variable is `false`. After you run the `compile_ultra` command, generate the floorplan file using the `write_floorplan` command.

Creating User Shapes

You can create user shapes by using the `create_user_shape` command. A user shape is a metal shape that is not associated with a net. You can specify the following types of user shapes by using the `-type` option: wire (horizontal or vertical), path, rectangle, polygon, and trapezoid.

If you do not specify the `-type` option, the user shape type is determined by the following rules, in order of precedence:

1. If you use the `-origin` option, the user shape is a wire. The wire is horizontal unless you also specify the `-vertical` option.
2. If you use the `-bbox` option, the user shape is a wire if you also use the `-path_type`, `-route_type`, or `-vertical` options. If you do not use any of these additional options, the user shape is a rectangle.
3. If you use the `-points` option, the user shape is a path.

In the following example, Design Compiler creates a wire user shape:

```
dc_shell-topo> create_user_shape -type wire \
    -origin {0 0} -length 10 -width 2 -layer M1
```

Use the `write_floorplan -user_shape` command to save user shapes to the generated floorplan file. Alternatively, use the `write_floorplan -preroute` command to save user shapes and net shapes to the floorplan file.

The `create_user_shape` commands are saved in the Preroute section of the floorplan file, as shown in the following example:

```
*****
# SECTION: Preroutes, with number: 6
*****
remove_user_shape *
create_user_shape -no_snap -type wire -layer METAL1 -datatype 0 \
```

```

-path_type 0 -width 2000 -route_type signal_route -length 3000 -origin {100 1500}
create_user_shape -no_snap -type wire -layer METAL2 -datatype 0 -path_type 0 \
-width 2000 -route_type signal_route -length 3000 -origin {100 1500}
create_user_shape -no_snap -type path -layer METAL4 -datatype 0 -path_type 0 \
-width 200 -route_type signal_route -points {{100 100} {400 100} {400 400} {600 400}}
create_user_shape -no_snap -type rect -layer METAL5 -datatype 0 \
-route_type user_enter -bbox {{100 100} {600 600}}
create_user_shape -no_snap -type poly -layer METAL6 -datatype 0 -boundary {{100 100} \
{300 200} {600 600} {400 400} {200 400} {100 100}}
create_user_shape -no_snap -type trap -layer METAL3 -datatype 0 -boundary {{100 100} \
{600 100} {400 600} {200 600}}

```

To remove objects that are user shapes, use the `remove_user_shape` command. Use an asterisk (*) to indicate that all user shapes in the design be removed, or use the `user_shapes` argument to specify a list of user shapes to be removed:

```
dc_shell-topo> remove_user_shape *
```

You can use the layout window in Design Compiler Graphical to view user shapes in the floorplan. You control the visibility of user shapes in the active layout view by setting options on the View Settings panel. For more information about viewing user shapes in the floorplan, see the “Examining Preroutes” topic in Design Vision Help.

Defining Physical Constraints for Pins

Design Compiler in topographical mode supports the following commands to control the placement of I/O cells and terminals:

- `set_pin_physical_constraints`
- `create_pin_guide`

Note:

Design Compiler in topographical mode supports only the placement of top-level design ports. It does not support the plan group or macro pin placement that is supported in IC Compiler and IC Compiler II design planning.

You can set physical constraints, such as the width and depth, on specified pins by using the `set_pin_physical_constraints` command. The following example shows the typical usage for the `set_pin_physical_constraints` command:

```
dc_shell-topo> set_pin_physical_constraints -pin_name {CCEN} -width 1.2 \
-depth 1.0 -side 1 -offset 10 -order 2
```

In the example, the command sets constraints on the CCEN pin. The geometry width is 1.2 microns, the depth is 1.0 micron, and the pin abuts to the side 1 edge, which is the lower left-most vertical edge. The offset distance is 10 microns, and the relative placement order is 2. Therefore, the pin is placed at a location on the left-most edge.

You can also use the `create_pin_guide` command to create a pin guide to constrain the pin assignment to a specified bounding box. This allows you to specify which area the port should be placed. The following example creates a pin guide named abc for all ports whose name begins with the string xyz.

```
dc_shell-topo> create_pin_guide -bbox {{-20 50} {20 800}} \
[get_ports xyz*] -name abc
```

The constraints that you specify are honored during the `compile_ultra` run.

By default, Design Compiler in topographical mode snaps ports to tracks during pin placement. However, you can prevent the snapping of ports to tracks by setting the `dct_port_dont_snap_onto_tracks` variable to `true`.

The pin constraints are saved to the design in the .ddc file. When you read the .ddc file in to a new Design Compiler topographical session, the pin constraints are restored.

The `write_floorplan` and `report_physical_constraints` commands do not report the pin constraint information. However, you can use the layout window in Design Compiler Graphical to verify that the pin constraints are placed correctly in the floorplan. To view the pin constraints in the layout view, select the Pin Guide visibility option (Vis) on the View Settings panel in the layout window. For more information, see the “Examining Physical Constraints” topic in Design Vision Help.

Design Compiler Graphical does not pass the pin constraints to floorplan exploration, and floorplan exploration does not pass the pin constraints to Design Compiler Graphical.

Creating Design Via Masters

Design via masters are created when

- A design via master is not in the technology file but it is in the DEF file being read by Design Compiler.
- You use the `create_via_master` command.

Via instances that are created from design via masters defined in a DEF file can be written out by using the `write_floorplan` command and then read in to Design Compiler by using the `read_floorplan` command. After the design via master is created, you can use it anywhere in the design, similar to a ContactCode definition in the technology file. The design via master is stored as an object together with the design cell in the binary design database, and therefore it can be used only in that design. To add an instance of a via defined as a design via master, use the `create_via` command. For more information, see the [Creating Vias](#).

You can use the `report_physical_constraints` command to report design via masters. For a report example, see [Reporting Design Via Masters](#).

Creating Vias

You can create vias by using the `create_via` command if the master via is defined in the technology file. Use the `-type` option to specify the type of via you want to create. The following example shows the creation of a via instance with its center specified as {213 215} and with an orientation of E. The via's size is determined by the via master, via1, which is defined in the technology file:

```
dc_shell-topo> create_via -type via -net vdd -master via1 \
    -route_type pg_strap -at {213 215} -orient E
```

The following example shows the creation of a via array with four columns and three rows. The via master is via4:

```
dc_shell-topo> create_via -type via_array -net vdd -master via4 \
    -route_type signal_route -at {215 215} -orient W -col 4 -row 3 \
    -x_pitch 0.4 -y_pitch 0.5
```

Use the `-name` option to specify the via's name. If the via name is not specified, its name is automatically assigned internally in the Milkyway database, and the via name is passed to the .ddc file.

If a via is defined in the DEF file and it does not exist in the logic library, a FRAM view is created in the Milkyway design library to save that via master. When that via is instantiated as a preroute in the floorplan, its instance is saved as a via cell that references the via master in the FRAM view.

You can visually examine preroute vias in your floorplan by viewing them in the Design Vision layout window. You can display or hide vias in the active layout view by setting options on the View Settings panel. Vias are visible by default. For more information about controlling the visibility of vias in the active layout view, see Design Vision Help.

Creating Routing Tracks

You can create tracks for routing layers and polygon layers by using the `create_track` command. Tracks cover the entire die area. If the die area is a polygon, the tracks cover the die area in a rectangle, stretching outside the polygon die area.

The `create_track` command creates a group of tracks on the floorplan so the router can use them to perform detail routing. You must specify either a polygon layer or a routing layer for the tracks. Creating the track on a polygon layer is not intended to support routing on the polygon layer.

The following example creates routing tracks for a routing layer named m3 on the floorplan:

```
dc_shell-topo> create_track \
    -layer MET3 \
    -dir Y \
```

```

-coord 0.000 \
-space 0.290 \
-count 4827 \
-bounding_box {{0.000 0.000} {1460.000 1400.000} }

dc_shell-topo> create_track \
    -layer MET3 \
    -dir X \
    -coord 0.000 \
    -space 0.290 \
    -count 5034 \
    -bounding_box {{0.000 0.000} {1460.000 1400.000}}

```

To return a collection of tracks in the current design that meet your selection criteria, use the `get_tracks` command. For example, you can use the `-within rectangle` option to create a collection containing all tracks within the specified rectangle. The format of a rectangle specification is `{{llx lly} {urx ury}}`, which specifies the lower-left and upper-right corners of the rectangle. The coordinate unit is specified in the technology file.

The following example returns the tracks within the rectangle with the lower-left corner at `{2 2}` and the upper-right corner at `{25 25}`:

```
dc_shell-topo> get_tracks * -within {{2 2} {25 25}}
{ "USER_TRACK_5389" }
```

The following example returns all tracks:

```
dc_shell-topo> get_tracks "*TRACK_*"
{ "DEF_TRACK_4683" "USER_TRACK_5389" }
```

Table 9-2 summarizes the commands related to tracks. The `create_track` and `remove_track` commands are written out by the `write_floorplan` command, while the other commands listed in the table are not.

Table 9-2 Summary of Routing Track Commands

Command	Description
<code>create_track</code>	Creates tracks for routing layers and polygon layers.
<code>remove_track</code>	Removes tracks from the current design.
<code>report_track</code>	Reports the routing tracks for a specified layer or for all layers.
<code>get_tracks</code>	Returns a collection of tracks in the current design that meet the selection criteria.

Creating Keepout Margins

You can create a keepout margin for the specified cell or library cell by using the `set_keepout_margin` command. Use the `-type hard` option or the `-type soft` option to specify the type of keepout, either hard or soft. The default is hard. Use the `-all_macros` option to apply the keepout margins to all macros, the `-macro_masters object_list` option for all instances of specified macro masters, or the `-macro_instances object_list` option for specified instances of macros.

You can specify explicit keepout margins by using the `-outer {lx by rx ty}` option, where the four numbers are the left, bottom, right, and top margins. A value of 0 results in no keepout margin for that side. The `-north` option sets the margins with respect to the north orientation of the cell.

Instead of specifying the keepout margins explicitly, you can have them derived automatically by using the `-tracks_per_macro_pin value` option, as shown in the following example. In this case, the keepout margin is calculated from the track width, the number of macro pins, and the specified track-to-pin ratio, which is typically set to a value near 0.5. A larger value results in larger keepout margins.

```
dc_shell-topo> set_keepout_margin -tracks_per_macro_pin .6 \
    -min_padding_per_macro .1 -max_padding_per_macro 0.2
```

The derived keepout margin is always hard; the `-type` option setting is ignored. The `-all_macros`, `-macro_masters`, and `-macro_instances` options are not allowed for derived margins. The derived margins are subject to minimum and maximum values that are specified by the `-min_padding_per_macro` and `-max_padding_per_macro` options.

[Table 9-3](#) summarizes the commands related to keepout margins.

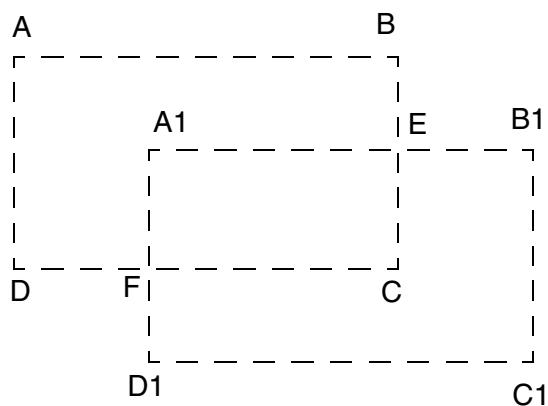
Table 9-3 Summary of Keepout Margin Commands

Command	Description
<code>set_keepout_margin</code>	Creates a keepout margin of the specified type for the specified cell or library cell
<code>remove_keepout_margin</code>	Removes keepout margins of a specified type for the specified cells or library cells in the design
<code>report_keepout_margin</code>	Reports keepout margins of a specified type for the specified cells in the design

Computing Polygons

The `compute_polygons` command returns a list or collection of polygons that exactly cover the region computed by performing a Boolean operation on the input polygons. The command performs an AND, OR, NOT, or XOR operation between two polygons or between two sets of polygons that are specified as lists of vertexes or as polygon collections. The typical scenario is to compute the geometry operations of physical objects, such as net shapes, user shapes, blockages, and so on. For example, assume you have two polygons, A-B-C-D-A and A1-B1-C1-D1-A1, as shown in [Figure 9-8](#).

Figure 9-8 Example of an OR Operation on Two Polygons

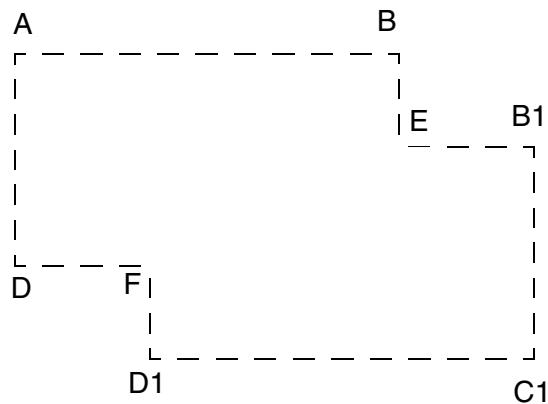


The following example performs the Boolean OR operation on these polygons. The result is the B-E-B1-C1-D1-F-D-A-B polygon.

```
dc_shell-topo> compute_polygons -boolean or \
{{0 30} {30 30} {30 10} {0 10} {0 30}} \
{{10 20} {40 20} {40 0} {10 0} {10 20}}
{{30.000 30.000} {30.000 20.000} {40.000 20.000} {40.000 0.000} {10.000 0.000}
{10.000 10.000} {0.000 10.000} {0.000 30.000} {30.000 30.000}}
```

[Figure 9-9](#) shows the resulting polygon.

Figure 9-9 Resulting Polygon



[Table 9-2](#) summarizes the commands related to polygons.

Table 9-4 Summary of Polygon Commands

Command	Description
compute_polygons	Returns a list or collection of polygons that exactly cover the region computed by performing a Boolean operation on the input polygons.
convert_to_polygon	Returns a polygon list or collection for the specified objects. Bounds and placement blockage objects are supported.
convert_from_polygon	Converts each polygon into a list or collection of mutually exclusive rectangles. You can specify a single polygon or multiple polygons.
resize_polygon	Resizes the input polygon. You can specify a single polygon or multiple polygons.
get_polygon_area	Calculates the area of the input polygon.

Including Physical-Only Cells

You can include physical cells that do not have logic functions in your floorplan. These cells are referred to as physical-only cells. Examples of physical-only cells include filler cells, tap cells, flip-chip pad cells, endcap cells, and decap cells. Although physical-only cells have no logic function, they create placement blockages that the tool considers during optimization and congestion analysis.

To learn about physical-only cells, see

- [Specifying Physical-Only Cells Manually](#)
 - [Extracting Physical-Only Cells From a DEF File](#)
 - [Creating Collections With Physical-Only Cells](#)
 - [Reporting Physical-Only Cells](#)
 - [Saving Physical-Only Cells](#)
-

Specifying Physical-Only Cells Manually

To specify physical-only cells manually, perform the following steps:

1. Create the physical-only cell by using the `create_cell` command with the `-only_physical` option:

```
dc_shell-topo> create_cell -only_physical MY_U_PO_CELL MY_FILL_CELL
Information: create physical-only cell 'MY_U_PO_CELL'
Warning: The newly created cell does not have location.
Timing will be inaccurate. (DCT-004)
```

This creates the physical-only cell `MY_U_PO_CELL`, which references the `MY_FILL_CELL` physical library cell, and sets the `is_physical_only` attribute on the created cell.

2. Assign a location to the cell.

Unlike physical cells with logic functions, Design Compiler topographical does not assign a location to a physical-only cell during synthesis.

To assign a location to a physical-only cell, use one of the following methods:

- Assign a location by using a DEF file.

You can define the location of a physical-only cell in a DEF file, as shown in the following example:

```
COMPONENTS 1 ;
- MY_U_PO_CELL MY_FILL_CELL + FIXED ( 3100000 700000 ) N ;
END COMPONENTS
```

After you have defined the location in a DEF file, apply the location to the physical-only cell by using the `extract_physical_constraints` command.

- Assign a location using Tcl commands.

You can define the location of physical-only cells in a Tcl floorplan file. For example, the following commands define the location of the MY_U_PO_CELL cell:

```
set obj [get_cells {MY_U_PO_CELL} -all]
set_attribute -quiet $obj orientation N
set_attribute -quiet $obj origin {3100.000 700.000}
set_attribute -quiet $obj is_fixed true
```

The `is_fixed` attribute must be assigned to the cell. Otherwise, the tool ignores the location specification. To assign the location to the physical-only cell, read the Tcl floorplan file into Design Compiler topographical mode using the `read_floorplan` command.

- Assign a location using the `set_cell_location` command.

You can define the location of physical-only cells by using the `set_cell_location` command in Design Compiler topographical mode, as shown:

```
set_cell_location -coordinates {3100.00 700.00} -orientation N \
    -fixed MY_U_PO_CELL
```

Extracting Physical-Only Cells From a DEF File

To extract physical-only cells from a DEF file and add them to the design, use the `-allow_physical_cells` option with the `extract_physical_constraints` command. The physical-only cell definition in the DEF file must contain the `+fixed` attribute. Otherwise, the tool ignores the location specification.

For example, if you want to extract the filler cells in the `my_design.def` file shown in [Example 9-34](#), use the following command:

```
dc_shell-topo> extract_physical_constraints \
    -allow_physical_cells my_design.def
```

Example 9-34 DEF Definitions for Two Filler Cells

```
COMPONENTS 2 ;
- fill_1 FILL_CELL + FIXED ( 3100000 700000 ) N ;
- fill_2 FILL_CELL + FIXED ( 3000000 600000 ) N ;
END COMPONENTS
```

The following types of cells are considered physical-only cells. Design Compiler marks these cells with the `is_physical_only` attribute when you run the `extract_physical_constraints -allow_physical_cells` command:

- Standard cell fillers
- Pad filler cells

- Corner cells
- Chip cells
- Cover cells
- Tap cells
- Cells containing only power and ground ports

In addition to extracting physical-only cells from the DEF file, the `extract_physical_constraints -allow_physical_cells` command also extracts some cells in the DEF file that are not identified as physical-only cells. These cells could be logic cells such as ROM or RAM cells. The tool updates the current design with these new logic cells and they are included in your Verilog netlist.

Design Compiler in topographical mode sets a `logical_cell_from_def` attribute on logic cells that are created from a DEF file. You can identify these logic cells by using the following command:

```
dc_shell-topo> get_cells -hierarchical \
                  -filter "logical_cell_from_def == true"
```

Note:

The physical-only cell definitions in the DEF file must include cell locations. Design Compiler does not assign locations to physical-only cells during synthesis. For more information about assigning a location to a physical-only cell, see [Specifying Physical-Only Cells Manually](#).

Creating Collections With Physical-Only Cells

To return a collection of all cells in the design, including physical-only cells, use the `get_cells` command with the `-all` option:

```
dc_shell-topo> get_cells -all
{fill_1 fill_2 U0 U1 U2 U3...}
```

To return a collection of all physical-only cells in the design, use the `all_physical_only_cells` command:

```
dc_shell-topo> all_physical_only_cells
{fill_1 fill_2}
```

See Also

- [Reporting Physical-Only Cells](#)

Reporting Physical-Only Cells

By default, the tool sets the `is_physical_only` attribute on all physical-only cells. To check if a cell is a physical-only cell, run the following command:

```
dc_shell-topo> get_attribute [get_cells -all $cell_name] is_physical_only
```

You can report physical-only cells in the following ways:

- To report physical-only cell information for the current design, use the `report_cell` command with the `-only_physical` option.

[Example 9-35](#) shows the report for the `fill_1` and `fill_2` physical-only cells.

Example 9-35 Reporting Physical-Only Cell Information

```
dc_shell-topo> report_cell -only_physical
*****
Report : cell
          -only_physical
Design : test
...
*****
Cell Reference Library Area Orient. Location
-----
fill_1 FILL_CELL xyz.mw 26.61 0 (3100.00, 700.00)
fill_2 FILL_CELL xyz.mw 26.61 0 (3000.00, 600.00)
-----
Total 2 cells 53.22
1
```

- To report the current design floorplan information that includes physical-only cell information, use the `report_physical_constraints` command.

[Example 9-36](#) shows a sample `report_physical_constraints` report.

Example 9-36 report_physical_constraints Output for Physical-Only Filler Cells

Cell2	Location	Orientation	Fixed
fill_1	{3100.000 700.000}	N	Yes
fill_2	{3000.00, 600.00}	N	Yes

See Also

- [Creating Collections With Physical-Only Cells](#)

Saving Physical-Only Cells

The `write_floorplan` command writes out the physical-only cell information while writing out the floorplan. [Example 9-37](#) shows the `write_floorplan` output for the two filler cells described in [Example 9-34](#).

Example 9-37 write_floorplan Output for Physical-Only Filler Cells

```
#*****  
# SECTION: Std Cells, with number: 2  
#*****  
set obj [get_cells {"fill_cell_1"} -all]  
set_attribute -quiet $obj orientation N  
set_attribute -quiet $obj origin {3100.000 700.000}  
set_attribute -quiet $obj is_fixed TRUE  
set obj [get_cells {"fill_cell_2"} -all]  
set_attribute -quiet $obj orientation N  
set_attribute -quiet $obj origin {3000 600.00}  
set_attribute -quiet $obj is_fixed TRUE  
The read_floorplan command extracts floorplan information
```

Similarly, the `read_floorplan` command reads the physical-only cell information while reading in the saved floorplan.

The physical-only cell information is saved in the `.ddc` file, along with other constraints, such as timing, and can be read back into Design Compiler.

Note:

Similar to all other physical information, the physical-only cell information is not written out in the ASCII netlist or the MilkyWay interface. It cannot be passed to IC Compiler through the `.ddc` file. The only way to pass physical information to IC Compiler is to use physical guidance or floorplan exploration in Design Compiler Graphical. Design Compiler in topographical mode does not pass any physical information to IC Compiler.

Specifying Relative Placement

The relative placement capability provides a way for you to create structures in which you specify the relative column and row positions of instances. These structures are called relative placement structures, which are placement constraints.

During placement and optimization, these structures are preserved and the cells in each structure are placed as a single entity.

To learn how to use relative placement, see

- [Relative Placement Overview](#)

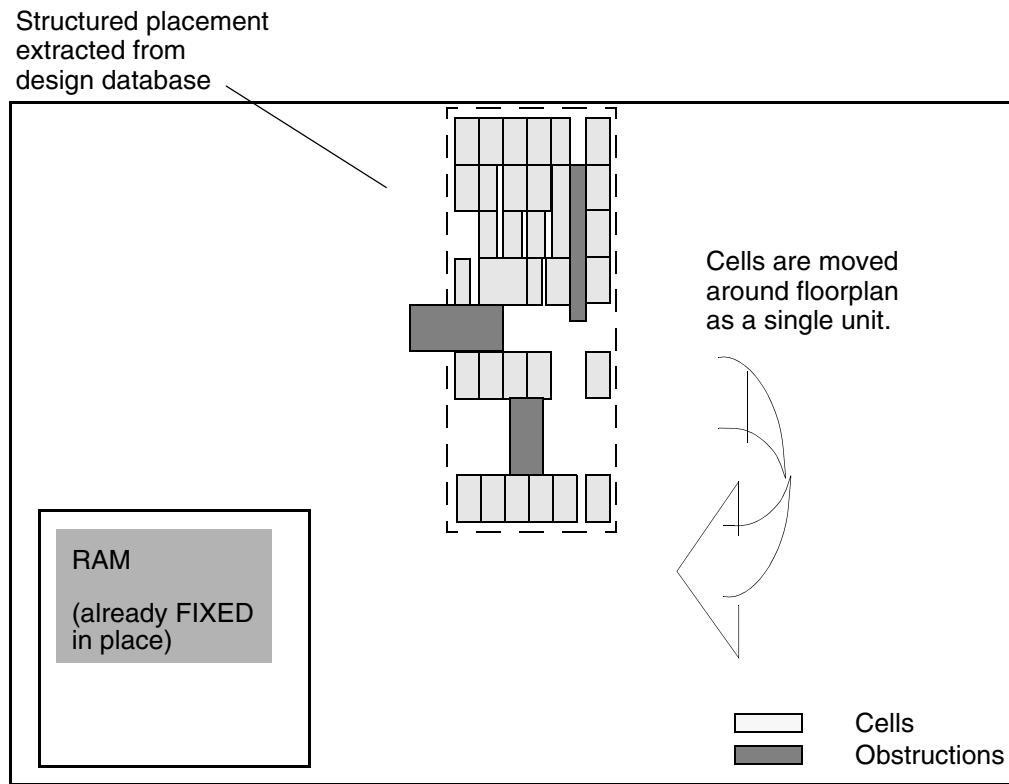
- Benefits of Relative Placement
- Methodology for the Relative Placement Flow
- Creating Relative Placement Using HDL Compiler Directives
- Summary of Relative Placement Tcl Commands
- Creating Relative Placement Groups
- Anchoring Relative Placement Groups
- Applying Compression to Relative Placement Groups
- Specifying Alignment
- Adding Objects to a Group
- Querying Relative Placement Groups
- Checking Relative Placement Constraints
- Saving Relative Placement Information
- Removing Relative Placement Group Attributes
- Sample Script for a Relative Placement Flow

Relative Placement Overview

Relative placement is usually applied to datapaths and registers, but you can apply it to any cells in your design, controlling the exact relative placement topology of gate-level logic groups and defining the circuit layout. You can use relative placement to explore QoR benefits, such as shorter wire lengths, reduced congestion, better timing, skew control, fewer vias, better yield, and lower dynamic and leakage power.

The relative placement constraints implicitly generate a matrix structure of instances and control the placement of these instances. You use the resulting annotated netlist for optimization, during which the tool preserves the structure and places it as a single entity or group, as shown in [Figure 9-10](#).

Figure 9-10 Relative Placement in a Floorplan



You can specify relative placement constraints in Design Compiler in topographical mode by using a dedicated set of Tcl commands, similar to the commands in IC Compiler. For information about the relative placement Tcl commands, see [Summary of Relative Placement Tcl Commands](#).

Design Compiler in topographical mode also supports relative placement information embedded within the Verilog or VHDL description. You specify relative placement data within Verilog and VHDL by using HDL compiler directives. For more information, see [Creating Relative Placement Using HDL Compiler Directives](#).

Benefits of Relative Placement

Relative placement provides the following benefits:

- Provides a method for maintaining structured placement for legacy or intellectual property (IP) designs
- Reduces the placement search space in critical areas of the design, which means greater predictability of QoR (wire length, timing, power)
- Correlates better with IC Compiler
- Can minimize congestion and improve routability

Methodology for the Relative Placement Flow

The relative placement flow follows these major steps:

1. Read one or more of the following files in Design Compiler in topographical mode:
 - An RTL design with relative placement constraints specified by HDL compiler directives.
 - A GTECH netlist with relative placement constraints specified by HDL compiler directives.
 - A mapped gate-level netlist with relative placement constraints specified by HDL compiler directives.
 - A mapped gate-level netlist without relative placement constraints; you will specify the constraints with Tcl commands in step 2.

For information about specifying relative placement using HDL compiler directives, including guidelines and restrictions, see the *HDL Compiler for VHDL User Guide* and the *HDL Compiler for Verilog User Guide*.

2. Define the relative placement constraints using Tcl commands. If the constraints have already been specified with HDL compiler directives, this step is optional. For information about specifying relative placement using HDL compiler directives, including guidelines and restrictions, see the *HDL Compiler for VHDL User Guide* and the *HDL Compiler for Verilog User Guide*.
 - a. Create the relative placement groups by using the `create_rp_group` command. See [Creating Relative Placement Groups](#).
 - b. Add relative placement objects to the groups by using the `add_to_rp_group` command. See [Adding Relative Placement Groups](#).
- Topographical mode annotates the netlist with the relative placement constraints and places an implicit `size_only` constraint on these cells.

3. Read floorplan information. For example, enter

```
dc_shell-topo> extract_physical_constraints floorplan.def
```

4. Check the relative placement by using the `check_rp_groups` command.

The command reports relative placement failures, such as:

Warning: The height '%f' of the RP group '%s' is more than
the height '%f' of the core area. (RPGP-028)

Warning: Relative placement leaf cell data may have been
lost. (RPGP-035)

Note:

The `check_rp_groups` command reports only the Tcl relative placement constraints at this stage in the flow. The command does not report constraints specified by HDL compiler directives until after the `compile_ultra` step.

5. Synthesize and optimize the design by using the `compile_ultra` command.
6. Visually verify the placement by using the layout view in the Design Vision layout window.

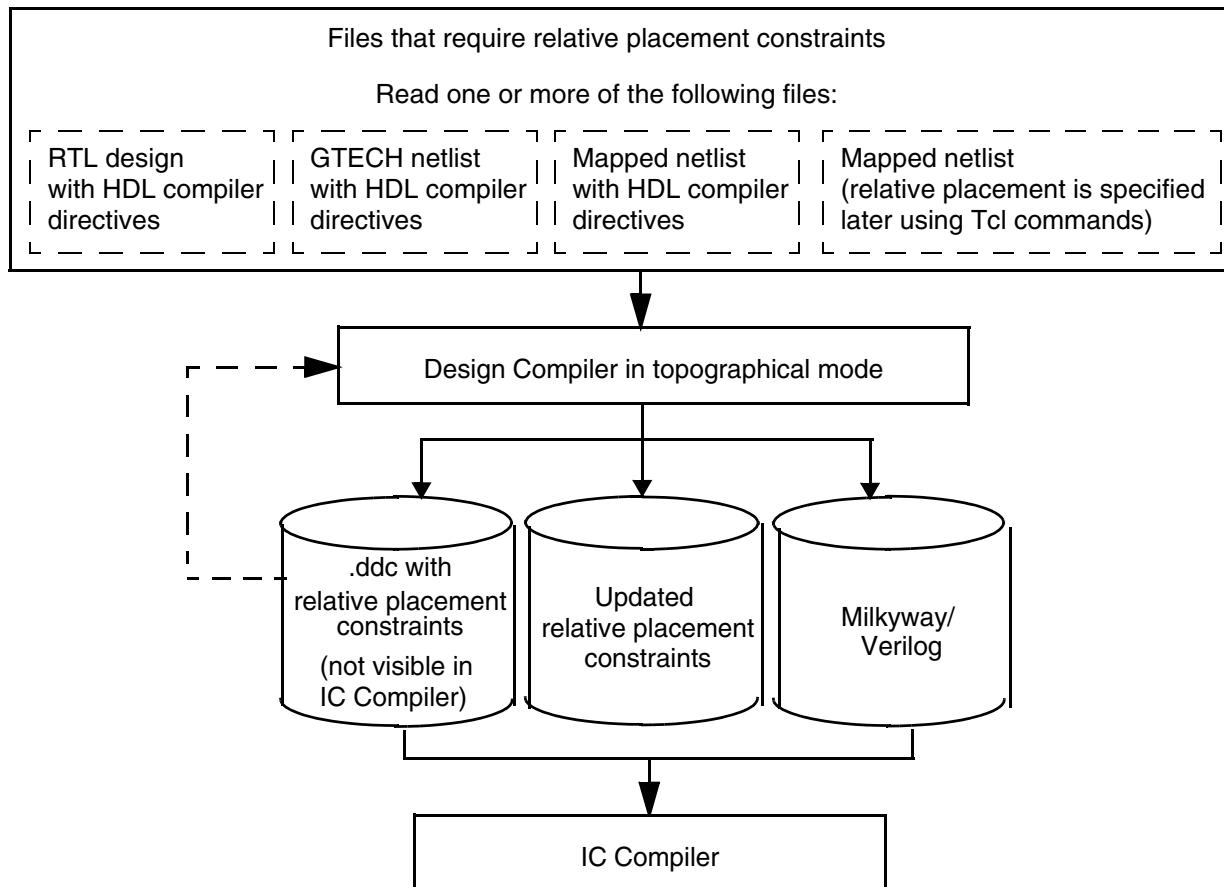
The Design Vision layout window allows you to visually verify that your floorplan is laid out according to your expectations. The layout view automatically displays floorplan constraints read in with `extract_physical_constraints` or read in with Tcl commands. You need to link all applicable designs and libraries to obtain an accurate floorplan.

For more information about using the GUI to view physical constraints, see the “Viewing the Floorplan” topic in Design Vision Help.

7. Write out the relative placement constraints to a Tcl file by using the `write_rp_groups` command. You can import this file into IC Compiler either by sourcing it or by using the `import_designs` command with the `-rp_constraint` option.

Relative Placement Flow Overview

[Figure 9-11](#) shows the relative placement flow in topographical mode.

Figure 9-11 Relative Placement Flow

Keep the following points in mind when you use relative placement:

- Topographical mode automatically places a `size_only` attribute on the relative placement cells to preserve the relative placement structure.
- Relative placement constraints are handled appropriately and preserved by using the `uniquify` and `ungroup` commands.
- Make sure that relative placement is applicable to your design. A design can contain both structured and unstructured elements. Some designs such as datapaths and pipelined designs are more appropriate for structured placement. Specifying relative placement constraints for cells that would be placed better by the tool can deliver poor results.
- Relative placement constraints are kept in the .ddc file for subsequent topographical mode sessions. These relative placement constraints are visible in Design Compiler in topographical mode only.

These relative placement constraints are not automatically transferred to IC Compiler. You can use the `write_rp_groups` command to write out the relative placement

constraints to a Tcl script that can be read into IC Compiler. For information about using relative placement in IC Compiler, see the *IC Compiler Implementation User Guide*.

Creating Relative Placement Using HDL Compiler Directives

Design Compiler topographical mode supports relative placement information embedded within the Verilog or VHDL description. This capability is enabled by HDL compiler directives that can specify and modify relative placement information. Using these compiler directives to specify relative placement increases design flexibility and simplifies relative placement because you no longer need to update the location of many cells in the design.

Using the embedded HDL compiler directives, you can place relative placement constraints in an RTL design, a GTECH netlist, or a mapped netlist. For information about specifying relative placement in Verilog and VHDL using HDL compiler directives, including guidelines and restrictions, see the *HDL Compiler for VHDL User Guide* and the *HDL Compiler for Verilog User Guide*.

Summary of Relative Placement Tcl Commands

You can specify relative placement constraints by using a dedicated set of Tcl commands, similar to the commands in IC Compiler. [Table 9-5](#) summarizes the Tcl commands available in Design Compiler topographical mode for relative placement.

Table 9-5 Summary of Relative Placement Tcl Commands

Command	Description
create_rp_group	Creates new relative placement groups.
add_to_rp_group	Adds items to relative placement groups.
set_rp_group_options	Sets relative placement group attributes.
report_rp_group_options	Reports attributes for relative placement groups.
get_rp_groups	Creates a collection of relative placement groups that match certain criteria.
write_rp_groups	Writes out relative placement information for specified groups.
all_rp_groups	Returns a collection of specified relative placement groups and all subgroups in their hierarchy.
all_rp_hierarchicals	Returns a collection of hierarchical relative placement groups that are ancestors of specified groups.

Table 9-5 Summary of Relative Placement Tcl Commands (Continued)

Command	Description
all_rp_inclusions	Returns a collection of hierarchical relative placement groups that include specified groups.
all_rp_instantiations	Returns a collection of hierarchical relative placement groups that instantiate specified groups.
all_rp_references	Returns a collection of relative placement groups that contain specified cells (either leaf cells or hierarchical cells that contain instantiated relative placement groups).
check_rp_groups	Checks relative placement constraints and reports failures.
remove_rp_groups	Removes a list of relative placement groups.
remove_rp_group_options	Reports attributes for the specified relative placement groups.
remove_from_rp_group	Removes an item (cell, relative placement group, or keepout) from the specified relative placement groups.
rp_group_inclusions	Returns collections for directly embedded included groups (added to a group by using the <code>add_to_rp_group -hierarchy</code> command) in all or specified groups.
rp_group_instantiations	Returns collections for directly embedded instantiated groups (added to a group by using the <code>add_to_rp_group -hierarchy -instance</code> command) in all or specified groups.
rp_group_references	Returns collections for directly embedded leaf cells (added to a group by using the <code>add_to_rp_group -leaf</code> command), directly embedded included cells that contain hierarchically instantiated cells (added to the included group by using the <code>add_to_rp_group -hierarchy -instance</code> command), or both in all or specified relative placement groups.

Creating Relative Placement Groups

A relative placement group is an association of cells, other groups, and keepouts. A group is defined by the number of rows and columns it uses. To create a relative placement group in Design Compiler topographical mode, use the `create_rp_group` command.

Topographical mode creates a relative placement group named `design_name::group_name`, where `design_name` is the design specified by the `-design`

option; if you do not use the `-design` option, the tool uses the current design. You must use this name or a collection of relative placement groups when referring to this group in other relative placement commands.

If you do not specify any options, the tool creates a relative placement group that has one column and one row. The group will not contain any objects. To add objects (leaf cells, relative placement groups, or keepouts) to a relative placement group, use the `add_to_rp_group` command, which is described in [Adding Objects to a Group](#).

For example, to create a group named `designA::rp1`, having six columns and six rows, enter

```
dc_shell-topo> create_rp_group rp1 -design designA -columns 6-rows 6
```

[Figure 9-12](#) shows the positions of columns and rows in a relative placement group.

Figure 9-12 Relative Placement Column and Row Positions

row 5	0 5	1 5	2 5	3 5	4 5	5 5
row 4	0 4	1 4	2 4	3 4	4 4	5 4
row 3		1 3	2 3	3 3	4 3	5 3
row 2	0 2	1 2	2 2	3 2	4 2	5 2
row 1	0 1	1 1	2 1	3 1		5 1
row 0	0 0	1 0	2 0	3 0	4 0	5 0
	col 0	col 1	col 2	col 3	col 4	col 5

In this figure,

- Columns count from column 0 (the leftmost column).
- Rows count from row 0 (the bottom row).
- The width of a column is the width of the widest cell in that column.
- The height of a row is determined by the height of the tallest cell in that row.
- It is not necessary to use all positions in the structure. For example, in this figure, positions 0 3 (column 0, row 3) and 4 1 (column 4, row 1) are not used.

Anchoring Relative Placement Groups

By default, topographical mode can place a relative placement group anywhere within the core area. You can control the placement of a top-level relative placement group by anchoring it.

To anchor a relative placement group, use the `create_rp_group` or the `set_rp_group_options` command with the `-x_offset` and `-y_offset` options. The offset values are float values, in microns, relative to the chip's origin.

If you specify both the x- and y-coordinates, the group is anchored at that location. If you specify only one coordinate, IC Compiler can determine the placement by sliding the group along the unspecified coordinate.

For example, to specify a relative placement group anchored at (100, 100), enter the following command:

```
dc_shell-topo> create_rp_group misc1 -design block1 \
              -columns 3 -rows 10 -x_offset 100 -y_offset 100
```

Applying Compression to Relative Placement Groups

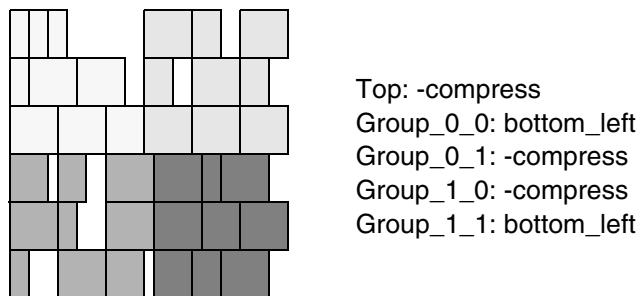
You can apply compression to a relative placement group in the horizontal direction during placement by using the `-compress` option with the `create_rp_group` or `set_rp_group_options` command. Setting this option enables bit-stack placement that places each row of a relative placement group without any gaps between leaf cells, lower-level hierarchical relative placement groups, or keepouts. Note that column alignment is not maintained when you use compression.

If you specify both the `-utilization` and `-compress` options, the utilization constraints are observed with gaps between leaf elements in a relative placement row. The `-compress` option does not propagate from a parent group to child groups. To disable relative placement with compression, use the `remove_rp_group_options -compress` command.

Supporting Compression with Mixed Alignment

Relative placement groups with alignment, such as bottom left, bottom right, or pin alignment, and the relative placement group that is created by using `-compress` can be placed on the same top-level groups as shown in [Figure 9-13](#). The individual groups of the top level are aligned with compression only if you specify the `-compress` option. Note that the compression specified at the top level does not propagate to the child groups. The default alignment of the top-level groups is bottom left and the `-compress` option is disabled by default.

Figure 9-13 Compression of Relative Placement Groups With Mixed Alignment



Specifying Alignment

To specify the default alignment method to use when placing leaf cells and relative placement groups, use the `-alignment` option with the `create_rp_group` or `set_rp_group_options` command.

```
dc_shell-topo> set_rp_group_options -alignment bottom-right \
[get_rp_groups *]
```

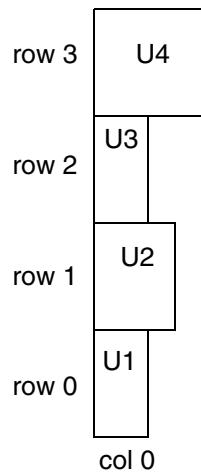
Controlling the cell alignment can improve the timing and routability of your design. You can specify a bottom-left, bottom-right or bottom-pin alignment. If you do not specify an option, the tool uses a bottom-left alignment.

The script in [Example 9-38](#) defines a relative placement group that is bottom-left aligned. The resulting structure is shown in [Figure 9-14](#). The `add_to_rp_group` commands are indented to show their relationship to the preceding `create_rp_group` command.

Example 9-38 Definition for Bottom-Left-Aligned Relative Placement Group

```
create_rp_group rpl -design pair_design -columns 1 -rows 4
  add_to_rp_group pair_design::rpl -leaf U1 -column 0 -row 0
  add_to_rp_group pair_design::rpl -leaf U2 -column 0 -row 1
  add_to_rp_group pair_design::rpl -leaf U3 -column 0 -row 2
  add_to_rp_group pair_design::rpl -leaf U4 -column 0 -row 3
```

Figure 9-14 Bottom-Left-Aligned Relative Placement Group



To align a group by pin location, use the `-alignment bottom-pin` and `-pin_align_name` options of the `create_rp_group` or `set_rp_group_options` command.

```
dc_shell-topo> set_rp_group_options -alignment bottom-pin \
    -pin_align_name align_pin
```

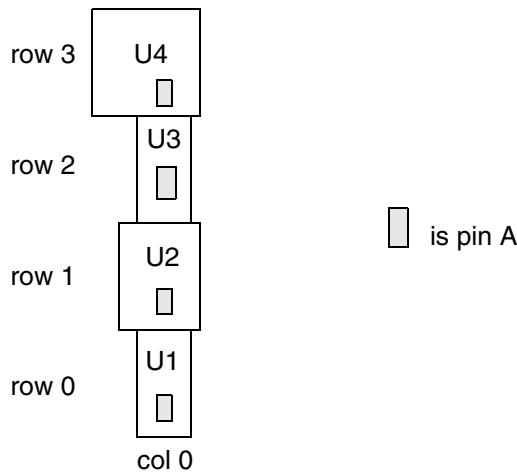
Topographical mode looks for the specified alignment pin in each cell in the column. If the alignment pin exists in a cell, the cell is aligned by use of the pin location. If the specified alignment pin does not exist in a cell, the cell is aligned at the bottom-left corner and the tool generates an information message. If the specified alignment pin does not exist in any cell in the column, the tool generates a warning message.

The script in [Example 9-39](#) defines a relative placement group that is aligned by pin A. The resulting structure is shown in [Figure 9-15](#).

Example 9-39 Definition for Relative Placement Group Aligned by Pins

```
create_rp_group rpl -design pair_design -columns 1 -rows 4 -pin_align_name A
add_to_rp_group pair_design::rpl -leaf U1 -column 0 -row 0
add_to_rp_group pair_design::rpl -leaf U2 -column 0 -row 1
add_to_rp_group pair_design::rpl -leaf U3 -column 0 -row 2
add_to_rp_group pair_design::rpl -leaf U4 -column 0 -row 3
```

Figure 9-15 Relative Placement Group Aligned by Pins



When you specify an alignment pin for a group, the pin applies to all cells in the group. You can override the group alignment pin for specific cells in the group by specifying the `-pin_align_name` option when you use the `add_to_rp_group` command to add the cells to the group.

Adding Objects to a Group

You can add leaf cells, other relative placement groups, and keepouts to relative placement groups (created with the `create_rp_group` command). You use the `add_to_rp_group` command to add objects.

When you add an object to a relative placement group, keep the following points in mind:

- The relative placement group to which you are adding the object must exist.
- The object must be added to an empty location in the relative placement group.

Adding Leaf Cells

To add a leaf cell to a relative placement group, use the `add_to_rp_group` command. In a relative placement group, a leaf cell can occupy multiple column positions or multiple row positions, which is known as leaf cell straddling. You can create a more compact relative placement group by straddling leaf cells. To define straddling, you specify multiple column or row positions by using the `-num_columns` or `-num_rows` options respectively. If you do not specify these options, the default is 1. For example, to create a leaf cell of two columns and one row, enter

```
dc_shell-topo> add_to_rp_group rp_group_name -leaf cell_name \
    -column 0 -num_columns 2 -row 0 -num_rows 1
```

You should not place a relative placement keepout at the same location of a straddling leaf cell. In addition, straddling is for leaf cells only, but not for hierarchical groups or keepouts.

Note:

You should not apply compression to a straddling leaf cell that has either multiple column positions, multiple row positions, or both. You can apply right alignment or pin alignment to a straddling leaf cell with multiple row positions, but not to a cell with multiple column positions.

Include the `-orientation` option with a list of possible orientations when you add the cells to the group with the `add_to_rp_group` command.

Aligning Leaf Cells Within a Column

You can align the leaf cells in a column of a relative placement group by using the following alignment methods:

- Bottom left (default)
- Bottom right
- Pin alignment

Controlling the cell alignment can improve the timing and routability of your design.

Aligning by Bottom-Left Corners

To align the leaf cells by aligning the bottom-left corners, use the `-alignment bottom-left` option with the `create_rp_group` command or the `set_rp_group_options` command:

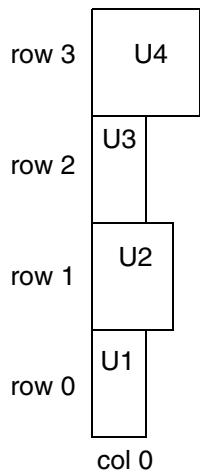
```
dc_shell-topo> set_rp_group_options -alignment bottom-left [get_rp_groups *]
```

The script in [Example 9-40](#) defines a relative placement group that is bottom-left aligned. The resulting structure is shown in [Figure 9-16](#).

Example 9-40 Definition for Bottom-Left Aligned Relative Placement Group

```
create_rp_group rpl -design pair_design -columns 1 -rows 4
  add_to_rp_group pair_design::rpl -leaf U1 -column 0 -row 0
  add_to_rp_group pair_design::rpl -leaf U2 -column 0 -row 1
  add_to_rp_group pair_design::rpl -leaf U3 -column 0 -row 2
  add_to_rp_group pair_design::rpl -leaf U4 -column 0 -row 3
```

Figure 9-16 Bottom-Left-Aligned Relative Placement Group



Aligning by Bottom-Right Corners

To align a group by aligning the bottom-right corners, use the `-alignment bottom-right` option with the `create_rp_group` command or the `set_rp_group_options` command:

```
dc_shell-topo> set_rp_group_options -alignment bottom-right \
    [get_rp_groups *]
```

Note:

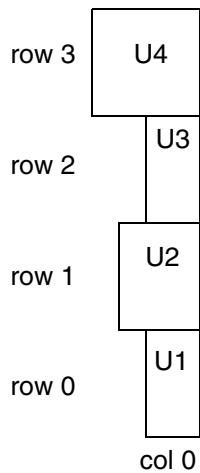
For hierarchical relative placement groups, the bottom-right alignment does not propagate through the hierarchy.

The script in [Example 9-41](#) defines a relative placement group that is bottom-right aligned. The resulting structure is shown in [Figure 9-17](#).

Example 9-41 Definition for Bottom-Right Aligned Relative Placement Group

```
create_rp_group rp1 -design pair_design -columns 1 -rows 4 \
-alignment bottom-right
add_to_rp_group pair_design::rp1 -leaf U1 -column 0 -row 0
add_to_rp_group pair_design::rp1 -leaf U2 -column 0 -row 1
add_to_rp_group pair_design::rp1 -leaf U3 -column 0 -row 2
add_to_rp_group pair_design::rp1 -leaf U4 -column 0 -row 3
```

Figure 9-17 Bottom-Right Aligned Relative Placement Group



Aligning by Pin Location

To align a group by pin location, use the `-alignment bottom-pin` and `-pin_align_name` options of the `create_rp_group` or `set_rp_group_options` command.

```
dc_shell-topo> set_rp_group_options -alignment bottom-pin \
    -pin_align_name align_pin
```

Design Compiler looks for the specified alignment pin in each cell in the column. If the alignment pin exists in a cell, the cell is aligned by use of the pin location. If the specified alignment pin does not exist in a cell, the cell is aligned at the bottom-left corner and Design Compiler generates an information message. If the specified alignment pin does not exist in any cell in the column, Design Compiler generates a warning message.

If you specify both pin alignment and cell orientation, Design Compiler resolves potential conflicts as follows:

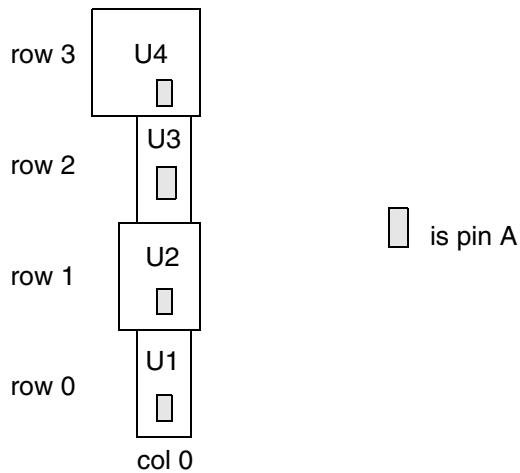
- User specifications for cell orientation take precedence over the pin alignment done by Design Compiler.
- Pin alignment done by Design Compiler takes precedence over the cell orientation optimization done by Design Compiler.

The script in [Example 9-42](#) defines a relative placement group that is aligned by pin A. The resulting structure is shown in [Figure 9-18](#).

Example 9-42 Definition for Relative Placement Group Aligned by Pins

```
create_rp_group rpl -design pair_design -columns 1 -rows 4 -pin_align_name A
add_to_rp_group pair_design::rpl -leaf U1 -column 0 -row 0
add_to_rp_group pair_design::rpl -leaf U2 -column 0 -row 1
add_to_rp_group pair_design::rpl -leaf U3 -column 0 -row 2
add_to_rp_group pair_design::rpl -leaf U4 -column 0 -row 3
```

Figure 9-18 Relative Placement Group Aligned by Pins



When you specify an alignment pin for a group, the pin applies to all cells in the group. You can override the group alignment pin for specific cells in the group by specifying the `-pin_align_name` option when you use the `add_to_rp_group` command to add the cells to the group.

Note:

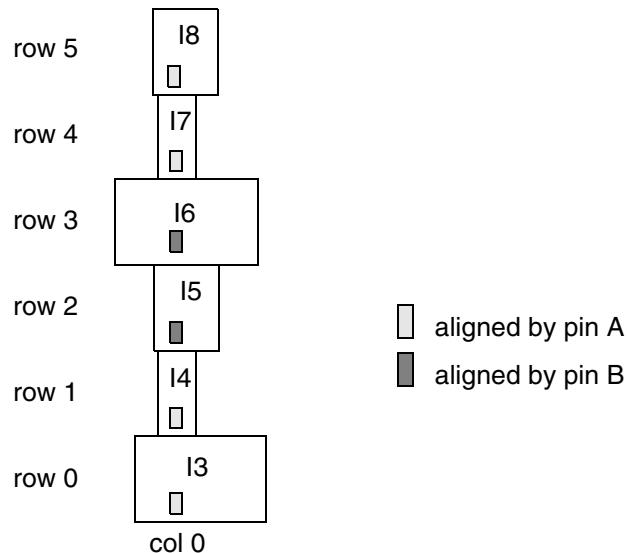
You cannot specify a cell-specific alignment pin when you add a leaf cell from the relative placement hierarchy browser.

The script in [Example 9-43](#) defines relative placement group misc1, which uses pin A as the group alignment pin; however, instances I5 and I6 use pin B as their alignment pin, rather than the group alignment pin. The resulting structure is shown in [Figure 9-19](#).

Example 9-43 Definition for Aligning a Group and Leaf Cells by Pins

```
create_rp_group misc1 -design block1 -columns 3 -rows 10 \
    -pin_align_name A
    add_to_rp_group block1::misc1 -leaf I3 -column 0 -row 0
    add_to_rp_group block1::misc1 -leaf I4 -column 0 -row 1
    add_to_rp_group block1::misc1 -leaf I5 -column 0 -row 2 \
        -pin_align_name B
    add_to_rp_group block1::misc1 -leaf I6 -column 0 -row 3 \
        -pin_align_name B
    add_to_rp_group block1::misc1 -leaf I7 -column 0 -row 4
    add_to_rp_group block1::misc1 -leaf I8 -column 0 -row 5
```

Figure 9-19 Relative Placement Group Aligned by Pins



Adding Relative Placement Groups

Hierarchical relative placement allows relative placement groups to be embedded within other relative placement groups. The embedded groups then are handled similarly to leaf cells. You can use hierarchical relative placement to simplify the expression of relative placement constraints. With hierarchical relative placement, you do not need to provide relative placement information multiple times for a recurring pattern.

There are two methods for adding a relative placement group to a hierarchical group. You can include the group or instantiate the group. You use the `add_to_rp_group` command for both methods:

- Include the group

If the relative placement group to be added is in the same design as its parent group, it is an included group. You can include groups in either flat or hierarchical designs. When you include a relative placement group in a hierarchical group, it is as if the included group is directly embedded within its parent group. An included group can be used only in a group of the same design and only one time. However, a group that contains an included group can be further included in another group in the same design or can be instantiated in a group of a different design.

- Instantiate the group

If the relative placement group to be added is in an instance of a subdesign of its parent group, it is an instantiated group. You can instantiate groups only in hierarchical designs.

The group specified in the `-hierarchy` option must be defined in the reference design of the instance specified in the `-instance` option. In addition, the specified instance must be in the

same design as the hierarchical group in which you are instantiating the specified group. Using an instantiated group is a useful way to replicate relative placement information across multiple instances of a design and to create relative placement relationships between those instances.

The script in [Example 9-44](#) creates a hierarchical group (rp2) that contains three instances of group rp1. Group rp1 is in the design pair_design and includes leaf cells U1 and U2. Group rp2 is a hierarchical group in the design mid_design that instantiates group rp1 three times (mid_design must contain at least three instances of pair_design). Group rp2 is treated as a leaf cell. You can instantiate rp2 multiple times and in multiple places, up to the number of times mid_design is instantiated in your netlist.

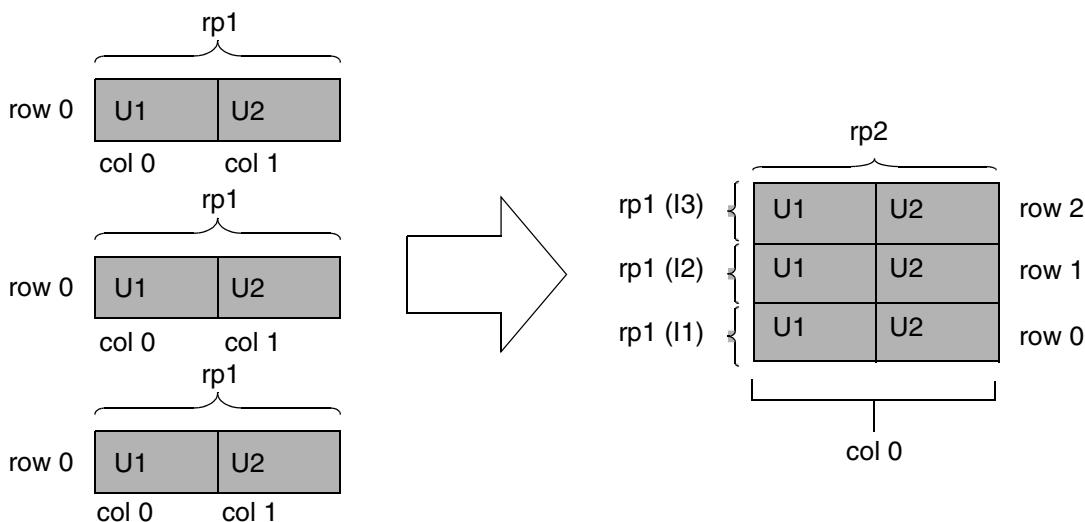
The resulting hierarchical relative placement group is shown in [Figure 9-20](#).

Example 9-44 Instantiating Groups in a Hierarchical Group

```
create_rp_group rp1 -design pair_design -columns 2 -rows 1
  add_to_rp_group pair_design::rp1 -leaf U1 -column 0 -row 0
  add_to_rp_group pair_design::rp1 -leaf U2 -column 1 -row 0

create_rp_group rp2 -design mid_design -columns 1 -rows 3
  add_to_rp_group mid_design::rp2 \
    -hierarchy pair_design::rp1 -instance I1 -column 0 -row 0
  add_to_rp_group mid_design::rp2 \
    -hierarchy pair_design::rp1 -instance I2 -column 0 -row 1
  add_to_rp_group mid_design::rp2 \
    -hierarchy pair_design::rp1 -instance I3 -column 0 -row 2
```

Figure 9-20 Instantiating Groups in a Hierarchical Group



Adding Keepouts

To add a keepout to a relative placement group, use the `add_to_rp_group` command. For example, to create a keepout named `gap1`, enter

```
dc_shell-topo> add_to_rp_group TOP::misc -keepout gap1 \
    -column 0 -row 2 -width 15 -height 1
```

where `TOP::misc` is the group list.

Querying Relative Placement Groups

[Table 9-6](#) lists the commands available for querying particular types of relative placement groups to annotate, edit, and view.

Table 9-6 Commands for Querying Relative Placement Groups

To return collections for this	Use this command
Relative placement groups that match certain criteria	<code>get_rp_groups</code>
All or specified relative placement groups and the included and instantiated groups they contain in their hierarchies	<code>all_rp_groups</code>
All or specified relative placement groups that contain included or instantiated groups	<code>all_rp_hierarchicals</code>
All or specified relative placement groups that contain included groups	<code>all_rp_inclusions</code>
All or specified relative placement groups that contain instantiated groups	<code>all_rp_instantiations</code>
All or specified relative placement groups that directly embed leaf cells (added to a group by <code>add_to_rp_group -leaf</code>) or instantiated groups (added to a group by <code>add_to_rp_group -hierarchy -instance</code>) in all or specified relative placement groups in a specified design or the current design	<code>all_rp_references</code>

For example, to create a collection of relative placement groups that start with the letter `g` in a design that starts with the letter `r`, enter

```
dc_shell-topo> get_rp_groups r*::g*
{ripple::grp_ripple}
```

To set the utilization to 95 percent for all relative placement groups, enter

```
dc_shell-topo> set_rp_group_options [all_rp_groups] -utilization 0.95
```

Checking Relative Placement Constraints

To check whether the relative placement constraints have been met, run the `check_rp_groups` command. The command reports relative placement failures such as the following:

Warning: The height '%f' of the RP group '%s' is more than the height '%f' of the core area. (RPGP-028)

Warning: Relative placement leaf cell data may have been lost. (RPGP-035)

The `check_rp_groups` command checks for the following failures:

- The relative placement group cannot be placed as a whole.
- The height or width of the relative placement group is greater than the height or width of the core area.
- The user-specified orientation cannot be met.

If a failure prevents the group from being placed as a single entity, as defined by the relative placement constraints, the failure is considered critical. If the failure does not prevent placement but causes the relative placement constraints to be violated, the failure is considered noncritical. The generated report contains separate sections for critical and noncritical failures.

Note that the `check_rp_groups` command does not check for the failure: the keepouts are not created correctly in the relative placement group.

You can check all relative placement groups by specifying the `-all` option or you can specify which relative placement groups to check. By default, the report is output to the screen. To save the generated report to a file, specify the file name by using the `-output` option.

For example, to check the relative placement constraints for groups `compare17::seg7` and `compare17::rp_group3` and to save the output in a file called `rp_failures.log`, run the following command:

```
dc_shell-topo> check_rp_groups "compare17::seg7 compare17::rp_group3" \
-output rp_failures.log
```

The generated report is similar to this:

```
*****
Report : The RP groups, which could not be placed.
...
No. of RP groups:1
*****
RP GROUP: compare17::seg7
-----
ERROR: Could not get clean area for placing RP group compare17::seg7.

*****
Report : The RP groups, not meeting all constraints but placed.
...
No. of RP groups:1
*****
RP GROUP: compare17::rp_group_3
-----
WARNING: Could not set user specified orientation for cell u[8].
```

To display a detailed report, use the `-verbose` option, as shown:

```
dc_shell-topo> check_rp_groups -all -verbose
```

The `-verbose` option reports all possible relative placement failures in the design.

Saving Relative Placement Information

To write out relative placement constraints, use the `write_rp_groups` command.

For example, to save all the relative placement groups to disk, remove the information from the design, and then re-create the information about the design, enter

```
dc_shell-topo> get_rp_groups
{mul::grp_mul ripple::grp_ripple example3::top_group}
dc_shell-topo> write_rp_groups -all -output my_groups.tcl
1
dc_shell-topo> remove_rp_groups -all -quiet
1
dc_shell-topo> get_rp_groups
Error: Can't find objects matching '**'. (UID-109)
dc_shell-topo> source my_groups.tcl
{example3::top_group}
dc_shell-topo> get_rp_groups
{example3::top_group ripple::grp_ripple mul::grp_mul}
```

By default, the `write_rp_groups` command writes out commands for creating the specified relative placement groups and to add leaf cells, hierarchical groups, and keepouts to these groups. The commands for generating subgroups within hierarchical groups are not written. The `write_rp_groups` command writes out updated relative placement constraints and includes names changes from unquification or ungrouping.

If you specified multiple column positions or multiple row positions for a cell using the `-num_columns` or `-num_rows` options with the `add_to_rp_group` command, the `write_rp_groups` command writes out the multiple-location cell as well.

You can modify the default behavior of the `write_rp_groups` command by using the options described in the man page.

Removing Relative Placement Group Attributes

To remove relative placement groups, run the `remove_rp_groups` command. You can remove all relative placement groups (by specifying the `-all` option), or you can specify which relative placement groups to remove. If you specify a list of relative placement groups, only the specified groups (and not groups included or instantiated within the specified group) are removed. To remove the included and instantiated groups of the specified groups, you must specify `-hierarchy`.

For example, to remove the relative placement group named `grp_ripple` and confirm its removal, enter

```
dc_shell-topo> get_rp_groups
{mul::grp_mul ripple::grp_ripple example3::top_group}
dc_shell-topo> remove_rp_groups ripple::grp_ripple
Removing rp group 'ripple::grp_ripple'
1
dc_shell-topo> get_rp_groups *grp_ripple
Error: Can't find object 'grp_ripple'. (UID-109)
dc_shell-topo> remove_rp_groups -all
Removing rp group 'mul::grp_mul'
Removing rp group 'example3::top_group'
1
```

To remove relative placement group attributes, run the `remove_rp_group_options` command. You must specify the group name and at least one option; otherwise, this command has no effect.

For example, to remove the `x_offset` attribute for the `block1::misc1` group, enter

```
dc_shell-topo> remove_rp_group_options block1::misc1 -x_offset
{block1::misc1}
```

The command returns a collection of relative placement groups for which attributes have been changed. If no attributes change for any object, an empty string is returned.

To remove objects from a relative placement group, use the `remove_from_rp_group` command. You can remove leaf cells (`-leaf`), included groups (`-hierarchy`), instantiated groups (`-hierarchy -instance`), and keepouts (`-keepout`).

For example, to remove leaf cell `carry_in_1` from `grp_ripple`, enter

```
dc_shell-topo> remove_from_rp_group ripple::grp_ripple -leaf carry_in_1
1
```

If you specified multiple column positions or multiple row positions for a cell using the `-num_columns` or `-num_rows` options with the `add_to_rp_group` command, the `remove_from_rp_group` command removes the cell from all its locations.

Sample Script for a Relative Placement Flow

[Example 9-45](#) is a sample script for running a relative placement flow.

Example 9-45 Sample Script for the Relative Placement Flow

```

# Set library and design paths
source setup.tcl
read_ddc design_name
current_design top
link

# Create relative placement constraints
create_rp_group grp_ripple -design ripple -rows 8
...
add_to_rp_group ripple::grp_ripple -leaf carry_in_1
...

# Apply design constraints
source constraints.tcl

# Read physical constraints
extract_physical_constraints top.def

# Check relative placement
check_rp_groups

# Perform synthesis
compile_ultra -scan

# Write out the netlist and relative placement
change_names -rules verilog
write_file -format verilog -hierarchy -output ripple
write_rp_groups -all -output ripple.rp.tcl

```

Magnet Placement

Magnet placement improves timing and congestion correlation between Design Compiler and IC Compiler or IC Compiler II. Magnet placement moves standard cells closer to objects specified as magnets. You can use magnet placement with any netlist that is fully mapped. Use the `magnet_placement` command to enable magnet placement if `magnet_placement` was used in IC Compiler or IC Compiler II.

The tool will only perform magnet placement if the standard cells are placed. If you use the `magnet_placement` command before the standard cells are placed, the tool will not pull any objects to the specified magnet. The tool will issue a warning for this condition.

To perform magnet placement, use the `magnet_placement` command with a specification of the magnets and options for any special functions you need to perform:

```
dc_shell-topo> magnet_placement [options] magnet_objects
```

To specify the fanout limit, use the `magnet_placement_fanout_limit` variable. If the fanout of a net exceeds the specified limit, the `magnet_placement` command does not pull cells of the net toward the magnet objects. The default setting is 1000.

By default, the magnet placement operation is terminated before the sequential cell when you use the `magnet_placement` command with the `-stop_by_sequential_cells` option. If you want to terminate the magnet placement operation after the sequential cell, set the `magnet_placement_stop_after_seq_cell` variable to `true`. The default is `false`.

Magnet placement allows cells to overlap by default. To prevent overlapping of cells, set the `magnet_placement_disable_overlap` variable to `true`.

To return a collection of cells that can be moved with magnet placement, use the `get_magnet_cells` command with the options you need:

```
dc_shell-topo> get_magnet_cells [options] magnet_list
```

Resetting Physical Constraints

To reset all physical constraints, use the `reset_physical_constraints` command. Use this command to clear all existing physical constraints before reading in a new or modified floorplan.

Saving Physical Constraints Using the `write_floorplan` Command

To save floorplan information, use the `write_floorplan` command, as described in the following sections. The `write_floorplan` command writes out individual floorplan commands relative to the top-level design, regardless of the current instance.

- [Saving Physical Constraints in IC Compiler Format](#)
- [Saving Physical Constraints in IC Compiler II Format](#)

Saving Physical Constraints in IC Compiler Format

By default, the `write_floorplan` command writes out physical constraint information in IC Compiler format. The output is a command script file that contains floorplan information such as bounds, placement blockages, route guides, plan groups, and voltage areas.

When writing out the floorplan from Design Compiler, you should use the `-all` option to write out the complete floorplan information. The `write_floorplan` command also provides options that allow you to write out partial floorplan information.

To use this output to create the floorplan in IC Compiler, you must read it in from the top level of the design with the `read_floorplan` command. You can also use the `source` command to import the floorplan information. However, the `source` command reports errors and warnings that are not applicable to Design Compiler in topographical mode. The `read_floorplan` command removes these unnecessary errors and warnings. In addition, you need to enable rule-based name matching manually when you use the `source` command. The `read_floorplan` command automatically enables rule-based name matching.

Saving Physical Constraints in IC Compiler II Format

To save physical constraints in IC Compiler II format, use the `write_floorplan -format icc2` command. The output is a directory that contains files in Tcl and DEF format. The following table lists the files generated by the command and the constraints transferred through these files.

Table 9-7 Files Generated by the `write_floorplan -format icc2` Command and Constraints Transferred to IC Compiler II

File name	Constraints transferred
<code>floorplan.def</code>	Die area, rows, tracks, components, pins, vias, special nets, keepout margins and so on
<code>floorplan.tcl</code>	Route guides, placement blockages, bounds, layer routing directions, voltage areas
<code>rp.tcl</code>	Relative placement groups
<code>routing_rule.layer.tcl</code>	Routing layer constraints on nets
<code>routing_rule.tcl</code>	Nondefault routing rule definitions
<code>routing_rule.net.tcl</code>	Routing rules on nets

When you specify the `-format icc2` option, Design Compiler Graphical writes out the complete floorplan information. Any `write_floorplan` command options that allow you to write out partial floorplan information are ignored.

Design Compiler Graphical specifies all layer promotion information as tool generated in the `routing_rule.layer.tcl` file. To accomplish this, Design Compiler Graphical writes out the `set_routing_rule` commands in the `routing_rule.layer.tcl` file with the `-min_layer_is_user` and `-max_layer_is_user` options set to `false`. Marking both the user-generated and tool-generated constraints as tool generated allows IC Compiler II to change the constraints.

To create the floorplan in IC Compiler II, read in the constraints from the top-level design in IC Compiler II by sourcing the `floorplan.tcl` file in the Design Compiler Graphical output directory.

Saving Physical Constraints Using the `write_def` Command

You can write out the physical constraints to a design exchange format (DEF) file by using the `write_def` command. The output DEF file contains floorplan information, such as bounds, placement blockages, route guides, plan groups, cell placement, pin locations, and so on. Design Compiler writes out the file using DEF syntax version 5.7 only.

If you reuse the DEF output, you must read it in from the top level of the design with the `extract_physical_constraints` command.

The following example uses the `write_def` command to write the placement information to a DEF file:

```
dc_shell-topo> write_def -components -placed -output dct.def
```

Design Compiler Graphical requires additional licensing to write out a DEF file. If you do not meet these licensing requirements, the DEF file that is written out cannot be read by third-party tools. For details about licensing, contact your Synopsys representative.

Note:

The `write_def -scan` option is not supported in Design Compiler. Use the `write_scan_def` command to capture the scan chain information.

Reporting Physical Constraints

To report any physical constraints that are applied to the design, use the `report_physical_constraints` command. The command reports the following physical constraints:

- Die area
- Placement area
- Utilization
- Aspect ratio
- Rectilinear outline
- Port side
- Port location
- Cell location
- Placement blockage
- Wiring keepouts
- Vias
- Design via masters
- Voltage area
- Site rows
- Bounds
- Preroutes
- User shapes
- Routing tracks
- Keepout margins

To report the `create_net_shape` commands (preroutes), use the `-pre_route` option with the `report_physical_constraints` command. If you do not want the report to show site row information, use the `-no_site_row` option.

For information about reporting physical constraints, see

- [Reporting Routing Tracks](#)
- [Reporting Preroutes](#)
- [Reporting Design Via Masters](#)
- [Reporting Keepout Margins](#)

Reporting Routing Tracks

You can use the `report_physical_constraints` command or the `report_track` command to report routing tracks. The following example shows the `report_physical_constraints` report format:

TRACK_NAME	LAYER	DIRECTION	COORDINATE	STEP	COUNT
USER_TRACK_1	METAL1	X	0.330	0.660	457
USER_TRACK_2	METAL1	Y	0.280	0.560	540

The `report_track` command reports the routing tracks for a specified layer or for all layers. The `-layer layer` option specifies which routing layer to report. You can specify a layer name, a layer number, or a collection containing one layer object. By default, Design Compiler reports the routing tracks on all layers.

You can use the `-dir` option to specify the routing direction. The valid values are either `X` or `Y`. By default, Design Compiler reports routing tracks in both directions.

To get a report similar to [Example 9-46](#) that shows the metal layer, the metal direction, the starting point, the number of tracks, the metal pitch, and the origin of the attributes, run the `report_track` command without any options:

```
dc_shell-topo> report_track
```

In [Example 9-46](#), all the attributes are defined from the DEF file, as indicated in the `Attr` column. However, the attributes could also be user defined (`usr`) or route defined (`rt`).

Example 9-46 Routing Track Report

```
*****
Report track
Design : frc_sys
...
...
*****
Layer      Direction     Start      Tracks      Pitch      Attr
-----
Attributes :
    usr : User defined
    rt  : Route66 defined
    def : DEF defined
```

metal1	X	0.100	11577	0.200	def
metal1	Y	0.200	11575	0.200	def
metal2	Y	0.200	11575	0.200	def

Reporting Preroutes

You can report preroutes, including user shapes, net shapes, vias, via arrays, via cells, and preroute net types by using the `-pre_route` option with the `report_physical_constraints` command.

The following example shows the report output from the `report_physical_constraints -pre_route` command:

```
report_physical_constraints -pre_route
*****
report_physical_constraints
...
*****
Design top
Physical Nets Number: 3
NAME      TYPE
vdd       power
vss       ground
net1      signal
1
```

Reporting Design Via Masters

You can use the `report_physical_constraints` command to report design via masters, as shown in the following example. The VIA MASTER section reports the via master definitions. The PREROUTES section includes the instantiation of vias using the via master definitions:

```
...
VIA MASTER total number: 35
INDEX NAME          RECT_NUM  LAYER  RECTANGLES/VIA RULES
#0   via23_array1    6        MET2 { -0.28 -0.35 0.28 0.35 }
                           MET3 { -0.28 -0.35 0.28 0.35 }
                           VIA2 { -0.22 -0.22 -0.08 -0.08 }
                           VIA2 { -0.22 0.08 -0.08 0.22 }
                           VIA2 { 0.08 -0.22 0.22 -0.08 }
                           VIA2 { 0.08 0.08 0.22 0.22 }

#1   via23_array2    6        MET2 { -0.28 -0.42 0.28 0.42 }
                           MET3 { -0.28 -0.42 0.28 0.42 }
                           VIA2 { -0.22 -0.22 -0.08 -0.08 }
                           VIA2 { -0.22 0.08 -0.08 0.22 }
                           VIA2 { 0.08 -0.22 0.22 -0.08 }
                           VIA2 { 0.08 0.08 0.22 0.22 }

...
```

```

PREROUTES total number: 352486
TYPE      LAYER      NETS      ROUTE_TYPE      DATA_TYPE      GEOMETRY      ATTRIBUTE
via       via23_array1  vdd!    pg_strap      at(375.5 438.5), N
via       via23_array2  vdd!    pg_strap      at(487.5 438.5), N
...

```

Reporting Keepout Margins

You can use the `report_physical_constraints` command or the `report_keepout_margin` command to report keepout margins. The `report_physical_constraints` command reports keepout margins in the following format:

CELL_KEEPOUT_MARGIN	TYPE	MARGIN_VALUE
U542	HARD	{10.000 10.000 50.000 50.000}
U543	SOFT	{15.000 15.000 15.000 15.000}

The `report_keepout_margin` command reports keepout margins of a specified type for the specified cells in the design. Use the `-type` option to specify the type of keepout to be reported. The valid values are either `hard` or `soft`. By default, both soft and hard keepouts are reported for the cells or library cells.

You can use the `-parameters` option to report the following pin-count-based parameter values:

- `tracks_per_macro_pin`
- `min_padding_per_macro`
- `max_padding_per_macro`

The following example reports hard keepouts for cells in an object list named `MY_CELL`:

```
dc_shell-topo> report_keepout_margin -type hard MY_CELL
```

The following example reports all the pin-count-based parameters to be used during the calculation of pin-count-based keepout margins for macro cells:

```
dc_shell-topo> report_keepout_margin -parameters
```


10

Compiling the Design

When you compile a design, Design Compiler reads the HDL source code and optimizes the design created from that description. The tool uses heuristics to implement a combination of library cells that meets the functional, speed, and area requirements of the design according to the attributes and constraints placed on it. The optimization process trades off timing and area constraints to provide the smallest possible circuit that meets the specified timing requirements.

To learn about compile flows and strategies, see the following topics:

- [Compile Commands](#)
- [Full and Incremental Compilation](#)
- [Compile Strategies](#)
- [Performing a Top-Level Compile](#)
- [Compile Log](#)
- [Resolving Multiple Instances of a Design Reference](#)
- [Test-Ready Compile](#)

Compile Commands

To compile a design, use the `compile` command if you are using DC Expert or the `compile_ultra` command if you are using DC Ultra or DC Graphical. Command options allow you to customize and control optimization.

- [The `compile` Command](#)
 - [The `compile_ultra` Command](#)
-

The `compile` Command

The `compile` command runs DC Expert. DC Expert synthesizes your HDL descriptions into optimized, technology-dependent, gate-level designs. It supports a wide range of flat and hierarchical design styles and can optimize both combinational and sequential designs for area, timing, and power.

The following list highlights `compile` command features:

- Command-line interface and graphical user interface
For more information, see [Running Design Compiler](#).
- Hierarchical compile (top down or bottom up)
For more information, see [Compile Strategies](#).
- Full and incremental compile techniques
For more information, see [Full and Incremental Compilation](#).
- Sequential optimization for complex flip-flops and latches
For more information, see [Optimizing the Design](#).
- Timing analysis
For more information, see [Analyzing Timing](#).

See Also

- [DC Expert](#)

The `compile_ultra` Command

Use the `compile_ultra` command for designs that have significantly tight timing constraints. The command is the best solution for timing-critical, high performance designs, and it allows you to apply the best possible set of timing-centric variables or commands during compile for critical delay optimization as well as QoR improvements.

The following list highlights `compile_ultra` command features that are in addition to the features listed in [The compile Command](#). Note that some features require additional licensing.

- DesignWare components

A DesignWare library is a collection of reusable circuit-design building blocks (components) that are tightly integrated into the Synopsys synthesis environment. When you use the `compile_ultra` command, Design Compiler selects the component with the best speed and area optimization from the DesignWare library during synthesis. For more information, see [DesignWare Libraries](#).

- Library-aware mapping and structuring

This capability characterizes your target logic library and creates a pseudolibrary called ALIB, which has mappings from Boolean functional circuits to actual gates from the target library. The ALIB file provides Design Compiler with greater flexibility and a larger solution space to explore tradeoffs between area and delay during optimization. For more information, see [Library-Aware Mapping and Synthesis](#).

- Automatic ungrouping

The `compile_ultra` command automatically ungroups certain logical hierarchies. Ungrouping merges subdesigns of a given level of the hierarchy into the parent cell or design. It removes hierarchical boundaries and allows Design Compiler to improve timing by reducing the levels of logic and to improve area by sharing logic. For more information about the automatic ungrouping of hierarchies, see [Automatic Ungrouping](#).

- Automatic boundary optimization

By default, the `compile_ultra` command optimizes across hierarchical boundaries. Boundary optimization is a strategy that can improve a hierarchical design by allowing the compile process to modify the port interface of lower-level designs. For more information about boundary optimization, see [Optimizing Across Hierarchical Boundaries](#).

- Automatic datapath extraction and optimization

Datapath extraction transforms arithmetic operators (for example, addition, subtraction, and multiplication) into datapath blocks. During datapath implementation, the tool uses a datapath generator to generate the best implementations for these extracted components. For more information, see [Datapath Extraction](#) and [Datapath Optimization](#).

- Register retiming

The `compile_ultra` command supports the `-retime` option, which enables Design Compiler to automatically perform register moves during optimization. This capability, called adaptive retiming, enables the tool to move registers and latches to improve timing. Adaptive retiming is intended for use in optimizing general designs; it does not replace the regular retiming engine available with the `optimize_registers` command. For more information, see [Adaptive Retiming](#).

- Topographical technology

When you run Design Compiler in topographical mode, the `compile_ultra` command uses topographical technology, which provides

- Accurate post-layout timing, area, and power predictions without the need for wire load model-based timing approximations, ensuring better correlation to the final physical design.
- Infrastructure to support multicore execution for faster runtimes.
- Support for multivoltage and multiple supply designs.
- Concurrent multicorner-multimode optimization, which reduces iterations and provides faster time-to-results.

For more information, see [Using Topographical Technology](#).

- Design Compiler NXT and Design Compiler Graphical

The Design Compiler NXT and Design Compiler Graphical tools, which are enabled with the `compile_ultra -spg` command, extends topographical technology by

- Enabling physical guidance technology, which includes enhanced placement and the ability to pass seed placement to IC Compiler to improve quality of results (QoR), correlation, and routability
- Reducing routing congestion during synthesis
- Enabling automatic layer optimization
- Allowing you to create and modify floorplans using floorplan exploration

For more information, see [Design Compiler NXT](#) and [Design Compiler Graphical](#).

See Also

- [DC Ultra](#)

Full and Incremental Compilation

Design Compiler supports the following compile flows:

- Full compile

During a full compile, Design Compiler maps and optimizes the entire design, resulting in a gate-level netlist of the design. Any mapped cells are unmapped to their logic function first; no attempt is made to preserve existing netlist structures.

- Incremental compile

During an incremental compile, Design Compiler can improve quality of results (QoR) by improving the structure of your design after the initial compile. Incremental mapping uses the existing gates from an earlier compilation as a starting point for the mapping process. It improves the existing design cost by focusing on the areas of the design that do not meet constraints and affects the second pass of compile. The existing structure is preserved where all constraints are already met. Mapping optimizations are accepted only if they improve the circuit speed or area.

To perform an incremental compile, use the `-incremental_mapping` option with the `compile` command or use the `-incremental` option with the `compile_ultra` command.

Keep the following points in mind when you do an incremental compile:

- The option enables only gate-level optimizations.
- Gates are not converted back to the generic technology (GTECH) level.
- Flattening and structuring are not done on the mapped portion of the design.
- Implementations for DesignWare operators are reselected if optimization costs can be improved.

In topographical mode, you can perform a second-pass, incremental compile to enable topographical-based optimization for post-topographical-based synthesis flows such as retiming, design-for-test (DFT), DFTMAX, and minor netlist edits. The primary focus in Design Compiler topographical mode is to maintain QoR correlation; therefore, only limited changes to the netlist can be made. Use the `-incremental` option with the `compile_ultra` command to run an incremental compile in topographical mode.

Compile Strategies

To optimize successfully, do the following before you compile the design:

- Ensure that partitioning is the best possible for synthesis. The quality of optimization results depends on how the HDL description is written. In particular, the partitioning of the hierarchy in the HDL, if done well, can enhance optimization.
- Define constraints as accurately as possible but do not overconstrain your design.
- Use a good synthesis library.
- Identify all multicycle and false paths.
- Instantiate clock gating elements.
- Optionally, define scan style.
- Determine the best strategy for optimizing your design.

Different pieces of your design require different compilation strategies, such as a top-down compile or bottom-up compile. You need to develop a compilation strategy before you compile. You can use the following compile strategies in Design Compiler to compile hierarchical designs in either wire load mode or topographical mode:

- [Top-Down Compilation](#)

The top-level design and all its subdesigns are compiled together.

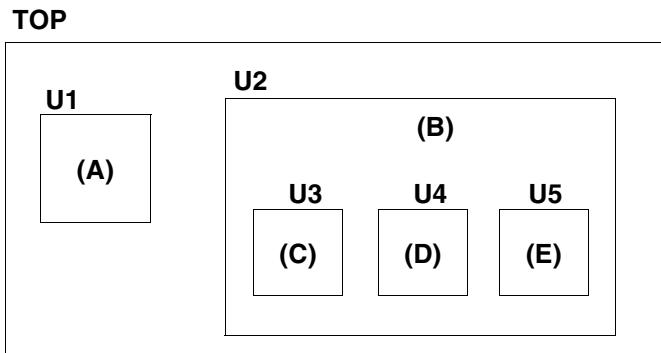
- [Bottom-Up Compilation](#)

The individual subdesigns are compiled separately, starting from the bottom of the hierarchy and proceeding up through the levels of the hierarchy until the top-level design is compiled.

- [Mixed Compile](#)

The top-down or bottom-up strategy, whichever is most appropriate, is applied to the individual subdesigns.

The top-down and bottom-up compile strategies are demonstrated using the simple design shown in [Figure 10-1](#).

Figure 10-1 Design to Illustrate Compile Strategies

The top-level, or global, specifications for this design, given in [Table 10-1](#), are defined by the script in [Example 10-1](#). These specifications apply to TOP and all its subdesigns.

Table 10-1 Design Specifications for Design TOP

Specification type	Value
Operating condition	WCCOM
Wire load model	"20x20"
Clock frequency	40 MHz
Input delay time	3 ns
Output delay time	2 ns
Input drive strength	drive_of (IV)
Output load	1.5 pF

Example 10-1 Constraints File for Design TOP (defaults.con)

```

set_operating_conditions WCCOM
set_wire_load_model "20x20"
create_clock -period 25 clk
set_input_delay 3 -clock clk \
    [remove_from_collection [all_inputs] [get_ports clk]]
set_output_delay 2 -clock clk [all_outputs]
set_load 1.5 [all_outputs]
set_driving_cell -lib_cell IV [all_inputs]
set_drive 0 clk
  
```

Note:

To prevent buffering of the clock network, the script sets the input drive resistance of the clock port (clk) to 0 (infinite drive strength).

Top-Down Compilation

In the top-down compile method, all environment and constraint settings are defined with respect to the top-level design. Although this strategy automatically takes care of interblock dependencies, the method is not practical for large designs because all designs must reside in memory at the same time. Therefore, you should only use the top-down compile strategy for designs that are not limited by memory or CPU.

You can use an alternate approach to compile a top-level design on a system that is memory limited. By replacing some of the subdesigns with hierarchical model representations, such as block abstractions, you can greatly reduce the memory requirements for the subdesign instantiation in the top-level design. For information about how to generate and use block abstractions, see [Using Hierarchical Models](#).

Design Compiler automatically compiles the hierarchical circuits without collapsing the hierarchy. After each module in the design is compiled, Design Compiler continues to optimize the circuit until the constraints are met. This process sometimes requires recompiling subdesigns on a critical path. When the performance goals are achieved or when no further improvement can be made, the compile process stops.

Hierarchical compilation is automatic when the design being compiled has multiple levels of hierarchy that are not marked `dont_touch`. Design Compiler preserves the hierarchy information and optimizes individual levels automatically, based on the constraints at the top level of hierarchy (`dont_touch` attributes placed on the top level of hierarchy are ignored).

The top-down compile strategy has these advantages:

- Provides a push-button approach
- Takes care of interblock dependencies automatically

Using the Top-Down Hierarchical Compile Strategy

Note:

If your top-level design contains one or more block abstractions, use the compile flow described in [Using Hierarchical Models](#).

To implement a top-down compile, perform the following steps:

1. Read in the entire design.
2. Apply attributes and constraints to the top level.

Attributes and constraints implement the design specification. You can assign local attributes and constraints to subdesigns, provided that those attributes and constraints are defined with respect to the top-level design.

3. Compile the design.

The constraints are applied by including the constraint file (defaults.con) shown in [Example 10-1](#). A top-down compile script for the TOP design is shown in [Example 10-2](#). The script contains comments that identify each of the steps.

Example 10-2 Top-Down Compile Script

```
# read in the entire design
read_verilog E.v
read_verilog D.v
read_verilog C.v
read_verilog B.v
read_verilog A.v
read_verilog TOP.v
current_design TOP
link

# apply constraints and attributes
source defaults.con

# compile the design
compile_ultra
```

See Also

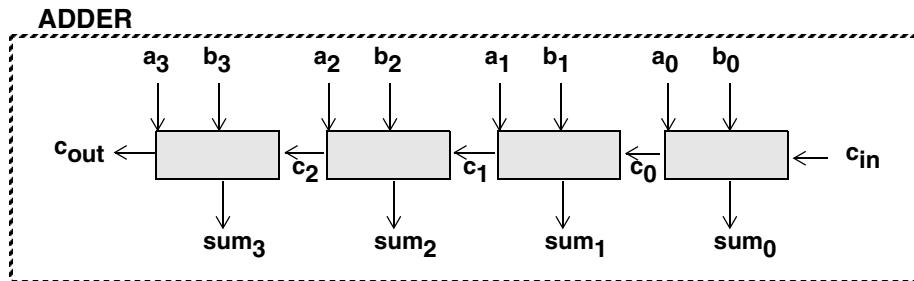
- [Using Attributes](#)
- [Defining the Design Environment](#)
- [Defining Design Constraints](#)

Top-Down Hierarchical Compile Strategy Example

This example of a top-down hierarchical compile uses a hierarchical, 4-bit adder design named ADDER. ADDER is composed of four 1-bit full adders with A, B, and CIN (carry in) inputs and SUM and COUT (carry out) outputs.

[Figure 10-2](#) shows a diagram of design ADDER.

Figure 10-2 Hierarchical Design



1. Read in as much of the design hierarchy as you need to use in this session. For example, enter

```
dc_shell> read_file -format ddc adder.ddc
Reading ddc file '/usr/design/adder.ddc'
Current design is 'ADDER'.
```

2. Run the `check_design -multiple_designs` command to determine whether any subdesigns (components) are referenced more than once (unresolved) or whether the hierarchy is recursive. The command reports all multiply instantiated designs along with instance names and associated attributes (`dont_touch`, `black_box`, and `ungroup`).

For example, enter

```
dc_shell> check_design -multiple_designs
Information: Design 'FULL_ADDER' is instantiated 4 times.
    Cell 'ADD0' in design 'ADDER'
    Cell 'ADD1' in design 'ADDER'
    Cell 'ADD2' in design 'ADDER'
    Cell 'ADD3' in design 'ADDER'
```

3. Resolve multiple instances.

For each cell referencing a subdesign that is not unique, you can let compile automatically uniquify the multiple instances, or, before running compile, you can do one of the following actions (you can use each action with one or more cells):

- Combine the cell into the surrounding circuitry (ungroup).
- Compile the cell separately, then use `set_dont_touch`.

You can manually force the tool to uniquify designs before compile by running the `uniquify` command, but this step contributes to longer runtime because the tool automatically “re-uniquifies” the designs when you compile the design. You cannot turn off the uniquify process.

4. Check the design again to verify that all multiple instances have been unqualified, or ungrouped or have the `dont_touch` attribute set. For example, enter

```
dc_shell> check_design -multiple_designs
```

5. Compile the design.

```
dc_shell> compile_ultra
```

See Also

- [Resolving Multiple Instances of a Design Reference](#)

Bottom-Up Compilation

Use the bottom-up compile strategy for medium-sized and large designs. In the bottom-up strategy, individual subdesigns are constrained and compiled separately and then incorporated into the top-level design. The top-level constraints are applied, and the design is checked for violations. Although it is possible that no violations are present, this outcome is unlikely because the interface settings between subdesigns usually are not sufficiently accurate at the start.

The bottom-up compile strategy provides these advantages:

- Compiles large designs by using the divide-and-conquer approach
- Requires less memory than top-down compile
- Allows time budgeting

The bottom-up compile strategy requires

- Iterating until the interfaces are stable
- Manual revision control

Bottom-up compilation is also known as the *compile-characterize-write_script-recompile* method. To improve the accuracy of the interblock constraints, you read in the top-level design and all compiled subdesigns and apply the `characterize` command to the individual cell instances of the subdesigns. Based on the more realistic environment provided by the compiled subdesigns, the `characterize` command captures environment and timing information for each cell instance and then replaces the existing attributes and constraints of each cell's referenced subdesign with the new values.

Using the improved interblock constraint, you recompile the characterized subdesigns and again check the top-level design for constraint violations. You should see improved results, but you might need to iterate the entire process several times to remove all significant violations.

After successful compilation, the designs are assigned the `dont_touch` attribute to prevent further changes to them during subsequent compile phases. Then, the compiled subdesigns are assembled to compose the designs of the next higher level of the hierarchy, which can also contain unmapped logic, and these designs are compiled. This compilation process is continued up through the hierarchy until the top-level design is synthesized. This method lets you compile large designs because Design Compiler does not need to load all the uncompiled subdesigns into memory at the same time. At each stage, however, you must estimate the interblock constraints, and typically you must iterate the compilations, improving these estimates, until all subdesign interfaces are stable.

When applying the bottom-up compile strategy, consider the following:

- The `read_file` command runs most quickly with the `.ddc` format. If you will not be modifying your RTL code after the first time you read (or elaborate) it, save the unmapped design to a `.ddc` file. This will save time when you reread the design.
- The `compile_ultra` command affects all subdesigns of the current design. If you want to optimize only the current design, you can remove or not include its subdesigns in your database, or you can place the `dont_touch` attribute on the subdesigns by using the `set_dont_touch` command.
- The subdesign constraints are not preserved after you perform a top-level compile. To ensure that you are using the correct constraints, always reapply the subdesign constraints before compiling or analyzing a subdesign.
- By default, `compile` modifies the original copy in the current design. This could be a problem when the same design is referenced from multiple modules, which are compiled separately in sequence. For example, the first `compile` could change the interface or the functionality of the design by boundary optimization. When this design is referenced from another module in the subsequent `compile`, the modified design is unqualified and used.

You can set the `compile_keep_original_for_external_references` variable to `true`, which enables `compile` to keep the original design when there is an external reference to the design. When the variable is set to `true`, the original design and its subdesigns are copied and preserved before doing any modifications during `compile` if there is an external reference to this design.

Typically, you require this variable only when you are doing a bottom-up compile without setting a `dont_touch` attribute on all the subdesigns, especially those with boundary optimizations turned on. If there is a `dont_touch` attribute on any of the instances of the design or in the design, this variable has no effect.

Using the Bottom-Up Hierarchical Compile Strategy

The bottom-up compile strategy requires these steps:

1. Develop both a default constraint file and subdesign-specific constraint files.

The default constraint file includes global constraints, such as the clock information and the drive and load estimates. The subdesign-specific constraint files reflect the time budget allocated to the subblocks.
2. Compile the subdesigns independently.
3. Read in the top-level design and any compiled subdesigns not already in memory.
4. Set the current design to the top-level design, link the design, and apply the top-level constraints.

If the design meets its constraints, you are finished. Otherwise, continue with the following steps.
5. Apply the `characterize` command to the cell instance with the worst violations.
6. Use `write_script` to save the characterized information for the cell.

You use this script to re-create the new attribute values when you are recompiling the cell's referenced subdesign.
7. Use `remove_design -all` to remove all designs from memory.
8. Read in the RTL design of the previously characterized cell.

Recompiling the RTL design instead of the cell's mapped design usually leads to better optimization.
9. Set `current_design` to the characterized cell's subdesign and recompile, using the saved script of characterization data.
10. Read in all other compiled subdesigns.
11. Link the current subdesign.
12. Choose another subdesign, and repeat steps 3 through 9 until you have recompiled all subdesigns, using their actual environments.

Note:

When performing a bottom-up compile, if the top-level design contains glue logic as well as the subblocks (subdesigns), you must also compile the top-level design. In this case, to prevent Design Compiler from recompiling the subblocks, you first apply the `set_dont_touch` command to each subdesign.

Bottom-Up Compile Script Example

[Example 10-3](#) shows a bottom-up compile script for the TOP design. The script contains comments that identify each of the steps in the bottom-up compile strategy. In the script it is assumed that block constraint files exist for each of the subblocks (subdesigns) in design TOP. The compile script also uses the default constraint file (defaults.con) shown in [Example 10-1](#).

Note:

This script shows only one pass through the bottom-up compile procedure. If the design requires further compilations, you repeat the procedure from the point where the top-level design, TOP.v, is read in.

Example 10-3 Bottom-Up Compile Script

```
set all_blocks {E D C B A}

# compile each subblock independently
foreach block $all_blocks {
    # read in block
    set block_source "$block.v"
    read_file -format verilog $block_source
    current_design $block
    link
    # apply global attributes and constraints
    source defaults.con
    # apply block attributes and constraints
    set block_script "$block.con"
    source $block_script
    # compile the block
    compile_ultra
}

# read in entire compiled design
read_file -format verilog TOP.v
current_design TOP
link
write -hierarchy -format ddc -output first_pass.ddc

# apply top-level constraints
source defaults.con
source top_level.con

# check for violations
report_constraint

# characterize all instances in the design
set all_instances {U1 U2 U2/U3 U2/U4 U2/U5}
characterize -constraint $all_instances

# save characterize information
foreach block $all_blocks {
```

```
current_design $block
set char_block_script "$block.wscr"
write_script > $char_block_script
}

# recompile each block
foreach block $all_blocks {

    # clear memory
    remove_design -all

    # read in previously characterized subblock
    set block_source "$block.v"
    read_file -format verilog $block_source

    # recompile subblock
    current_design $block
    link
    # apply global attributes and constraints
    source defaults.con
    # apply characterization constraints
    set char_block_script "$block.wscr"
    source $char_block_script
    # apply block attributes and constraints
    set block_script "$block.con"
    source $block_script
    # recompile the block
    compile_ultra
}
```

See Also

- [Using Hierarchical Models](#)
- [Performing a Bottom-up Hierarchical Compile](#)

Provides steps for performing a bottom-up compile in topographical mode

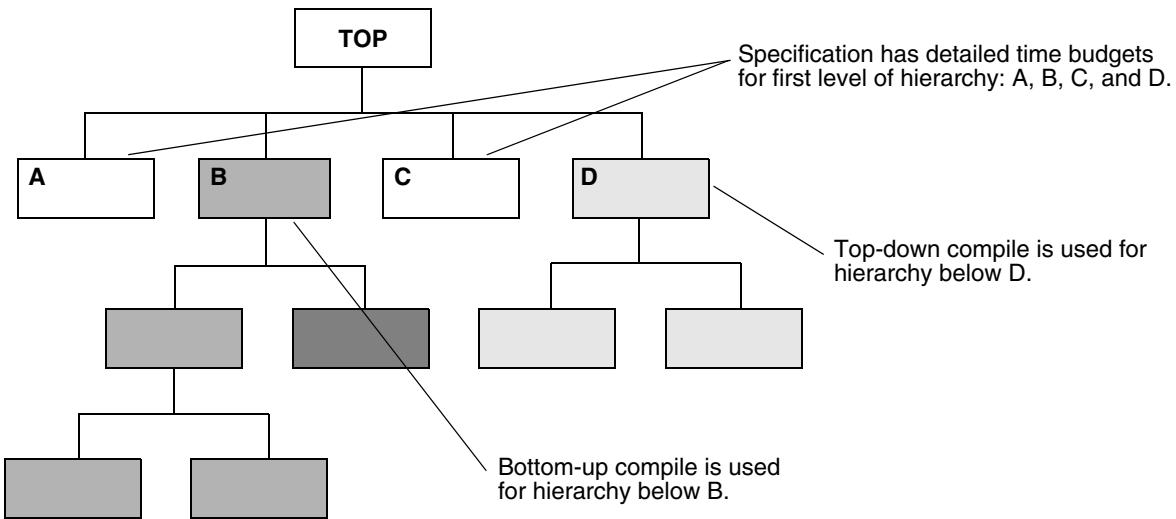
Mixed Compile

You can take advantage of the benefits of both the top-down and the bottom-up compile strategies by using both.

- Use the top-down compile strategy for small hierarchies of blocks.
- Use the bottom-up compile strategy to tie small hierarchies together into larger blocks.

[Figure 10-3](#) shows an example of the mixed compilation strategy.

Figure 10-3 Mixing Compilation Strategies



Performing a Top-Level Compile

The `-top` option of the `compile` and `compile_ultra` commands invokes the top-level optimization process. The top-level optimization capability fixes constraint violations occurring at the top level after the subblocks in a design are assembled. These violations might occur due to changes in the environment around the subblocks as a result of the optimizations that have been performed in the subblocks.

Top-level optimization fixes only violations of top-level nets because it is assumed that the subblocks have been compiled separately and are meeting timing. However, any design rule violations present in the design will be fixed regardless of where the violation occurs.

The `-top` option works with mapped designs only and runs significantly faster than an incremental compilation because of its emphasis on top-level nets. Additionally, it does not perform any incremental implementation selection of synthetic components, structuring, or area recovery on the design.

Using the `-top` Option With Other Compile Options

The `compile_ultra -top` option is incompatible with the following options:

- `-incremental`
- `-timing_high_effort_script`
- `-area_high_effort_script`

The `compile -top` option is incompatible with the following options:

- `-incremental_mapping`
- `-exact_map`
- `-no_map`
- `-area_effort`

Limiting Optimization to Paths Within a Specific Range

By default, the `-top` option aims to fix all design rule violations in the entire design but only the intermodule timing paths with violations. It does not address the intramodule timing paths. To direct Design Compiler to attempt to optimize the intermodule paths with timing delay violations within a specific range, set the `critical_range` attribute before you compile with the `-top` option.

Fixing Timing Violations For All Paths

By default, when the `-top` option is used, Design Compiler fixes all design rules but only those timing violations whose paths cross top-level hierarchical boundaries.

Setting the `compile_top_all_paths` environment variable to `true` causes the `-top` option to attempt to fix timing violations for all paths.

Compile Log

The compile log records the status of the compile run. Each optimization task has an introductory heading, followed by the actions taken while that task is performed. There are three tasks in which Design Compiler works to reduce the compile cost function described in [Compile Cost Function](#):

- Delay optimization (see [Delay Optimization](#))
- Design rule fixing (see [Design Rule Fixing](#))
- Area recovery (see [Area Recovery](#))

While completing these tasks, Design Compiler performs many trials to determine how to reduce the cost function. For this reason, these tasks are collectively known as the trials phase of optimization. The compile log displays reduction in costs as shown in

Example 10-4. You can customize the trials phase output by setting the `compile_log_format` variable.

Example 10-4 Compile Log

Beginning Delay Optimization Phase

ELAPSED TIME	AREA	WORST SLACK	TOTAL SLACK	DESIGN RULE COST	ENDPOINT
0:00:18	15003.8	0.00	0.0	33.8	
0:00:18	15003.8	0.00	0.0	33.8	
0:00:18	15003.8	0.00	0.0	33.8	
0:00:18	15003.8	0.00	0.0	33.8	
0:00:18	15003.8	0.00	0.0	33.8	
0:00:18	15003.8	0.00	0.0	33.8	
0:00:18	15003.8	0.00	0.0	33.8	
0:00:18	15003.8	0.00	0.0	33.8	

Beginning Design Rule Fixing (max_capacitance)

ELAPSED TIME	AREA	WORST SLACK	TOTAL SLACK	DESIGN RULE COST	ENDPOINT
0:00:18	15003.8	0.00	0.0	33.8	
0:00:19	15172.8	0.00	0.0	0.0	
0:00:20	15172.8	0.00	0.0	0.0	
0:00:20	15172.8	0.00	0.0	0.0	
0:00:20	15172.8	0.00	0.0	0.0	

Beginning Area-Recovery Phase (max_area 0)

ELAPSED TIME	AREA	WORST SLACK	TOTAL SLACK	DESIGN RULE COST	ENDPOINT
0:00:20	15172.8	0.00	0.0	0.0	

```

0:00:21    15085.7      0.00      0.0      0.0
0:00:21    15085.7      0.00      0.0      0.0
0:00:21    15085.7      0.00      0.0      0.0
0:00:21    15085.7      0.00      0.0      0.0

Loading db file '/remote/srm147/LS_IMAGES/D20061217/libraries/syn/
tc6a_cbacore.db'
Optimization Complete
-----
1

```

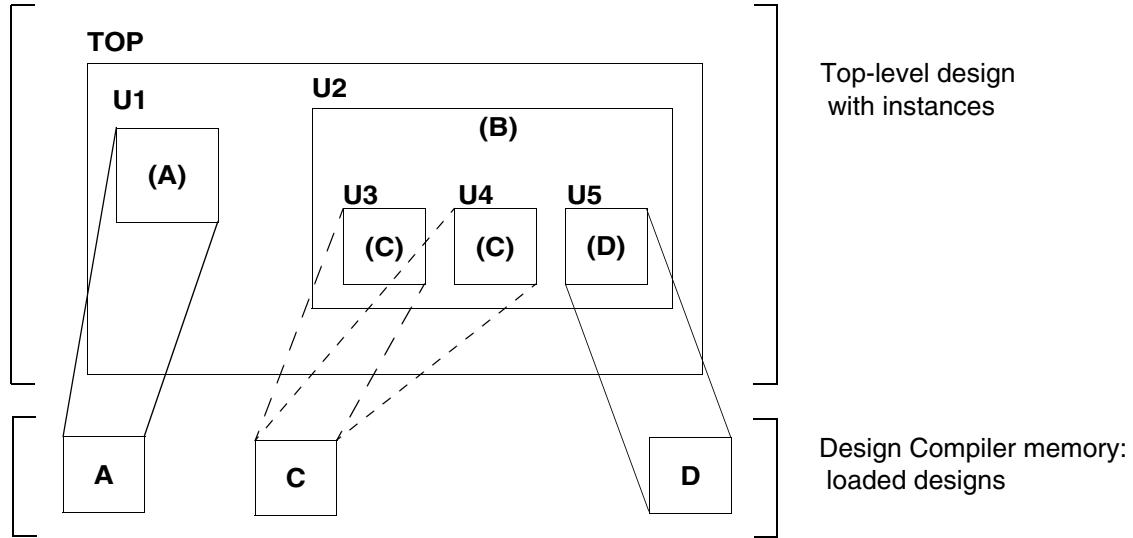
See Also

- [Analyzing Your Design During Optimization Using the Compile Log](#)

Resolving Multiple Instances of a Design Reference

In a hierarchical design, subdesigns are often referenced by more than one cell instance, that is, multiple references of the design can occur. For example, [Figure 10-4](#) shows the design TOP, in which design C is referenced twice (U2/U3 and U2/U4).

Figure 10-4 Multiple Instances of a Design Reference



To report information messages related to multiply-instantiated designs, use the `check_design -multiple_designs` command. The command lists all multiply instantiated

designs along with instance names and associated attributes (`dont_touch`, `black_box`, and `ungroup`). The following methods are available for handling designs with multiple instances:

- **The Uniquify Method**

The tool automatically uniquifies designs as part of the compile process. You can manually force the tool to uniquify designs before compile by running the `uniquify` command. However, this step contributes to longer runtimes because the tool automatically “re-uniquifies” the designs when compiling the design. You cannot turn off the uniquify process.

- **Compile-Once-Don’t-Touch Method**

This method uses the `set_dont_touch` command to preserve the compiled subdesign while the remaining designs are compiled.

- **Ungroup Method**

This method uses the `ungroup` command to remove the hierarchy.

The Uniquify Method

The `uniquify` process copies and renames any multiply referenced design so that each instance references a unique design. The process removes the original design from memory after it creates the new, unique designs. The original design and any collections that contain it or its objects are no longer accessible.

The tool automatically uniquifies designs as part of the compile process. The uniquification process can resolve multiple references throughout the hierarchy the current design (except those having a `dont_touch` attribute). After this process finishes, the tool can optimize each design copy based on the unique environment of its cell instance.

You can also create unique copies for specific references by using the `-reference` option of the `uniquify` command, or you can specify specific cells by using the `-cell` option. Design Compiler makes unique copies for cells specified with the `-reference` or the `-cells` option, even if they have a `dont_touch` attribute.

The `uniquify` command accepts instance objects—that is, cells at a lower level of hierarchy. When you use the `-cell` option with an instance object, the complete path to the instance is uniquified. For example, the following command uniquifies both instances `mid1` and `mid1/bot1`, assuming that `mid1` is not unique:

```
prompt> uniquify -cell mid1/bot1
```

Design Compiler uses the naming convention specified in the `uniqualify_naming_style` variable to generate the name for each copy of the subdesign. The default naming convention is `%s_%d`, which is defined as follows:

`%s`

The original name of the subdesign, or the name specified in the `-base_name` option.

`%d`

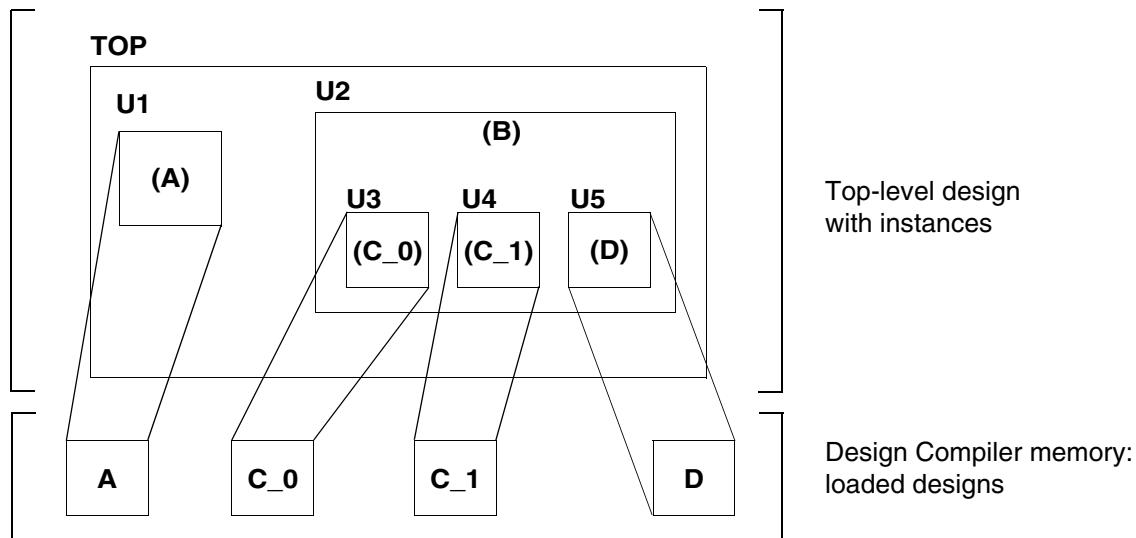
The smallest integer value that forms a unique subdesign name.

The following command sequence resolves the multiple instances of design C in design TOP shown in [Figure 10-4](#); it uses the automatic unqiufy method to create new designs `C_0` and `C_1` by copying design C and then replaces design C with the two copies in memory.

```
prompt> current_design top
prompt> compile_ultra
```

[Figure 10-5](#) shows the result of running this command sequence.

Figure 10-5 Uniquify Results



Compared with the compile-once-don't-touch method, the unqiufy method has the following characteristics:

- Requires more memory
- Takes longer to compile

Compile-Once-Don't-Touch Method

If the environments around the instances of a multiply referenced design are sufficiently similar, use the compile-once-don't-touch method. In this method, you compile the design, using the environment of one of its instances, and then you use the `set_dont_touch` command to preserve the subdesign during the remaining optimization.

To use the compile-once-don't-touch method to resolve multiple instances, follow these steps:

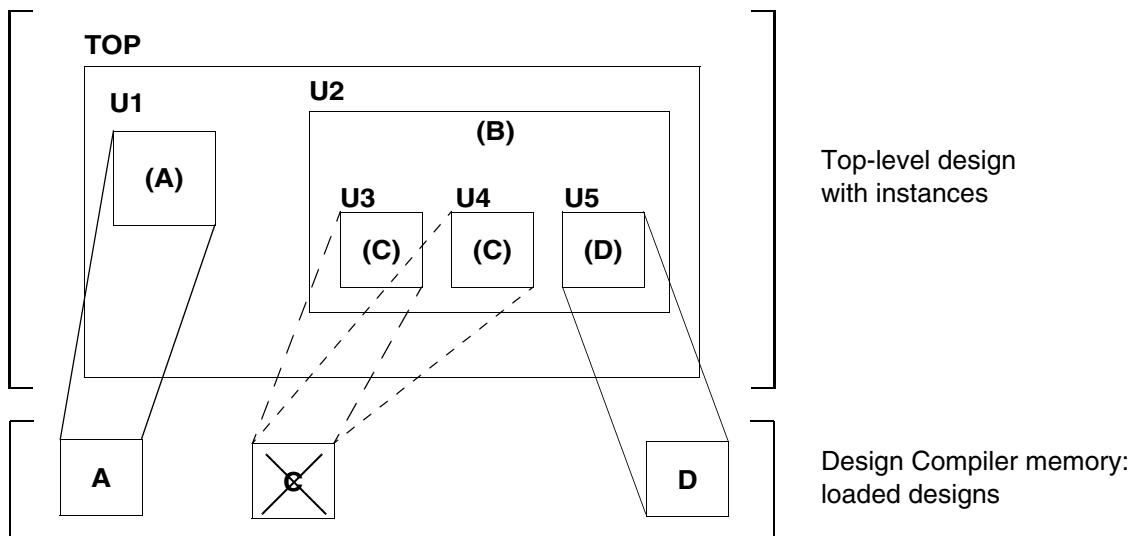
1. Characterize the subdesign's instance that has the worst-case environment.
2. Compile the referenced subdesign.
3. Use the `set_dont_touch` command to set the `dont_touch` attribute on all instances that reference the compiled subdesign.
4. Compile the entire design.

For example, the following command sequence resolves the multiple instances of design C in design TOP by using the compile-once-don't-touch method (assuming U2/U3 has the worst-case environment). In this case, no copies of the original subdesign are loaded into memory.

```
prompt> current_design top
prompt> characterize U2/U3
prompt> current_design C
prompt> compile_ultra
prompt> current_design top
prompt> set_dont_touch {U2/U3 u2/u4}
prompt> compile_ultra
```

[Figure 10-6](#) shows the result of running this command sequence. The X drawn over the C design, which has already been compiled, indicates that the `dont_touch` attribute has been set. This design is not modified when the top-level design is compiled.

Figure 10-6 Compile-Once-Don't-Touch Results



The compile-once-don't-touch method has the following advantages:

- Compiles the reference design one time
- Requires less memory than the uniquify method
- Takes less time to compile than the uniquify method

The principal disadvantage of the compile-once-don't-touch method is that the characterization might not apply well to all instances. Another disadvantage is that you cannot ungroup objects that have the `dont_touch` attribute.

See Also

- [Preserving Subdesigns](#)

Ungroup Method

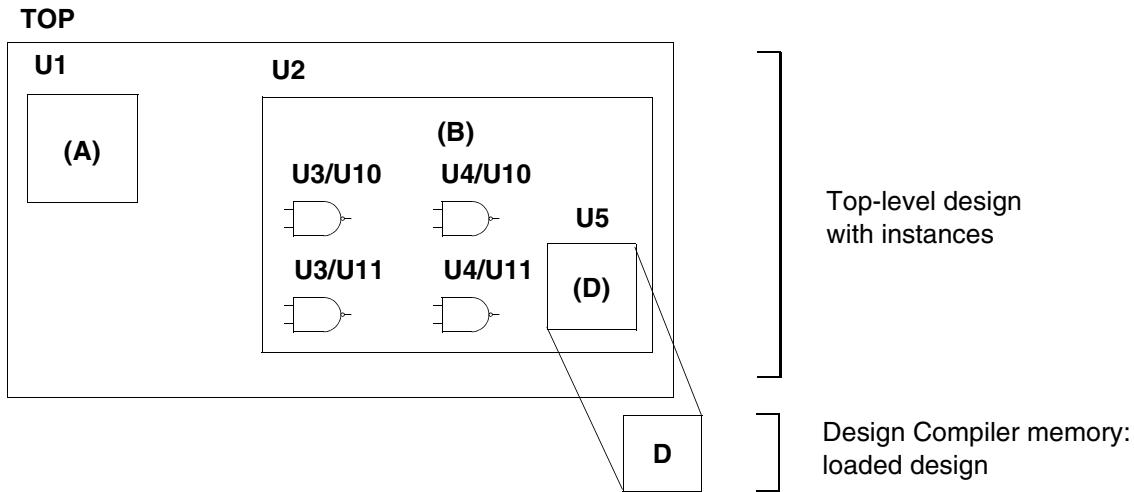
The ungroup method has the same effect as the uniquify method (it makes unique copies of the design), but in addition, it removes levels of hierarchy. This method uses the `ungroup` command to produce a flattened netlist. For details about the `ungroup` command, see [Removing Levels of Hierarchy](#).

After ungrouping the instances of a subdesign, you can recompile the top-level design. For example, the following command sequence uses the ungroup method to resolve the multiple instances of design C in design TOP:

```
prompt> current_design B
prompt> ungroup {U3 U4}
prompt> current_design top
prompt> compile_ultra
```

[Figure 10-7](#) shows the result of running this command sequence.

Figure 10-7 Ungroup Results



The ungroup method has the following characteristics:

- Requires more memory and takes longer to compile than the compile-once-don't-touch method
- Provides the best synthesis results

The obvious drawback in using the ungroup method is that it removes the user-defined design hierarchy.

See Also

- [Removing Levels of Hierarchy](#)

Test-Ready Compile

Test-ready compile reduces iterations and design time, by accounting for the impact of the scan implementation during the logic optimization process. The optimization cost functions consider the impact of the scan cells themselves and the additional loading due to the scan-chain routing. By accounting for the timing impact of scan design from the start of the synthesis process, test-ready compilation eliminates the need to recompile your design after scan insertion. Use the `-scan` option of the `compile_ultra` command or the `compile` command to enable test-ready compile. When you use this option, the tool replaces all sequential elements during optimization.

See Also

- [Performing a Test-Ready Compile](#)

11

Optimizing the Design

Optimization is the Design Compiler synthesis step that maps the design to an optimal combination of specific target logic library cells, based on the design's functional, speed, and area requirements. You use the `compile_ultra` command or the `compile` command to start the compile process, which synthesizes and optimizes the design. Design Compiler provides options that enable you to customize and control optimization. Several of the many factors affecting the optimization outcome are discussed in the following topics.

To learn about Design Compiler optimization processes and techniques, see the following topics:

- [Overview of the Optimization Process](#)
- [Optimization Phases](#)
- [Compile Cost Function](#)
- [Optimization Flow](#)
- [Optimization Techniques](#)

Overview of the Optimization Process

Optimization is the step in the synthesis process that attempts to implement a combination of library cells that meets the functional, speed, and area requirements of your design. Optimization transforms the design into a technology-specific circuit based on the attributes and constraints you place on the design.

Design Compiler performs the following levels of optimization in the following order:

1. [Architectural Optimization](#)
 2. [Logic-Level Optimization](#)
 3. [Gate-Level Optimization](#)
-

Architectural Optimization

Architectural optimization works on the HDL description. It includes such high-level synthesis tasks as

- Sharing common subexpressions
- Sharing resources
- Selecting DesignWare implementations (not available in DC Expert)
- Reordering operators
- Identifying arithmetic expressions for datapath synthesis (not available in DC Expert)

Except for DesignWare implementations, these high-level synthesis tasks occur only during the optimization of an unmapped design. DesignWare selection can recur after gate-level mapping.

High-level synthesis tasks are based on your constraints and your HDL coding style. After high-level optimization, circuit function is represented by GTECH library parts, that is, by a generic, technology-independent netlist.

See Also

- [Preparing for Synthesis](#)

Provides information about how your coding style affects architectural optimization

- [Resource Sharing](#)

Provides information about resource sharing, which reduces the amount of hardware needed to implement operators in your Verilog or VHDL description

- [DesignWare Libraries](#)

Provides information about using DesignWare components and developing additional DesignWare libraries using a DesignWare Developer license

Logic-Level Optimization

Logic-level optimization works on the GTECH netlist. It consists of the following two processes:

- Structuring

This process adds intermediate variables and logic structure to a design, which can result in reduced design area. Structuring is constraint based. It is best applied to noncritical timing paths.

During structuring, Design Compiler searches for subfunctions that can be factored out and evaluates these factors, based on the size of the factor and the number of times the factor appears in the design. Design Compiler turns the subfunctions that most reduce the logic into intermediate variables and factors them out of the design equations.

- Flattening

The goal of this process is to convert combinational logic paths of the design to a two-level, sum-of-products representation. Flattening is carried out independently of constraints. It is useful for speed optimization because it leads to just two levels of combinational logic.

During flattening, Design Compiler removes all intermediate variables, and therefore all its associated logic structure, from a design.

Gate-Level Optimization

Gate-level optimization works on the generic netlist created by logic synthesis to produce a technology-specific netlist. It includes the following processes:

- Mapping

This process uses gates (combinational and sequential) from the target libraries to generate a gate-level implementation of the design whose goal is to meet timing and area goals. You can use the various options of the `compile_ultra` or `compile` command to control the mapping algorithms used by Design Compiler. For more information, see [Sequential Mapping](#).

- Delay optimization

The process goal is to fix delay violations introduced in the mapping phase. Delay optimization does not fix design rule violations or meet area constraints. For more information, see [Delay Optimization](#).

- Design rule fixing

The process goal is to correct design rule violations by inserting buffers or resizing existing cells. Design Compiler tries to fix these violations without affecting timing and area results, but if necessary, it does violate the optimization constraints. For more information, see [Design Rule Fixing](#).

- Area optimization

The process goal is to meet area constraints after the mapping, delay optimization, and design rule fixing phases are completed. However, Design Compiler does not allow area recovery to introduce design rule or delay constraint violations as a means of meeting the area constraints. For more information, see [Area Recovery](#).

You can change the priority of the constraints by using the `set_cost_priority` command. Also, you can disable design rule fixing by specifying the `-no_design_rule` option when you run the `compile_ultra` command or `compile` command. However, if you use this option, your synthesized design might violate design rules.

See Also

- [Optimization Constraints](#)
- [Managing Constraint Priorities](#)

Optimization Phases

During optimization, Design Compiler attempts to meet the constraints you have set on the design. Design Compiler's optimization algorithms use costs to determine if a design change is an improvement. Design Compiler calculates two cost functions: one for design rule constraints and one for optimization constraints and accepts an optimization move if it decreases the cost of one component without increasing more-important costs. By default, the *design rule constraints* (transition, fanout, capacitance, and cell degradation) have a higher priority than the *optimization constraints* (delay and area).

To learn more about the types of optimizations Design Compiler performs, see the following topics:

- [Combinational Optimization](#)
- [Sequential Optimization](#)
- [Local Optimizations](#)

Combinational Optimization

The combinational optimization phase transforms the logic-level description of the combinational logic to a gate-level netlist.

Combinational optimization includes

- [Technology-Independent Optimization](#)

This optimization operates at the logic level. Design Compiler represents the gates as a set of Boolean logic equations.

- [Mapping](#)

During this process, Design Compiler selects components from the logic library to implement the logic structure.

- [Technology-Specific Optimization](#)

This optimization operates at the gate level.

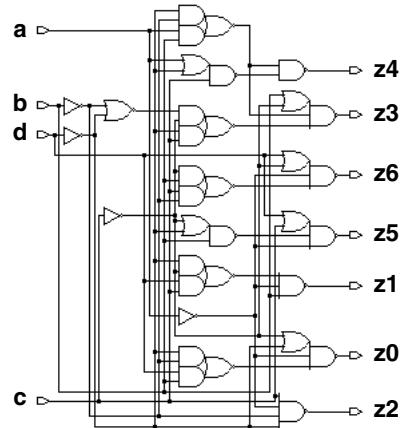
[Figure 11-1](#) shows the logic-level description of the combinational logic and the gate-level optimization for design LED.

Figure 11-1 Logic-Level and Gate-Level Optimization for Design LED

```

design_name LED
.inputnames a b c d
.outputnames z0 z1 z2 z3 z4 z5 z6
n1 = c' ;
n2 = d' ;
n3 = a' ;
n4 = ((n2' + n6') * (d' + b')) ;
n5 = ((n2' + n1') * (d' + c')) ;
n6 = b' ;
n7 = ((n2' + n6') * (a' + b')) ;
n8 = ((n12' + n1') * (n2' + c')) ;
n9 = ((a' * n2') + c') ;
n10 = ((n1' * n2') + b') ;
n11 = ((n1' + b') * (c' + n6')) ;
n12 = (n6' * n2') ;
z0 = ((n1' * n2') + n3' + n4') ;
z1 = (n5' + n3' + b') ;
z2 = (c' + n3' + n6' + n2') ;
z3 = ((b' * n1') + n7' + n8') ;
z4 = (n7' + n9') ;
z5 = ((d' * c') + n10' + n3') ;
z6 = ((d' * n1') + n3' + n11') ;

```



Total Area = 32

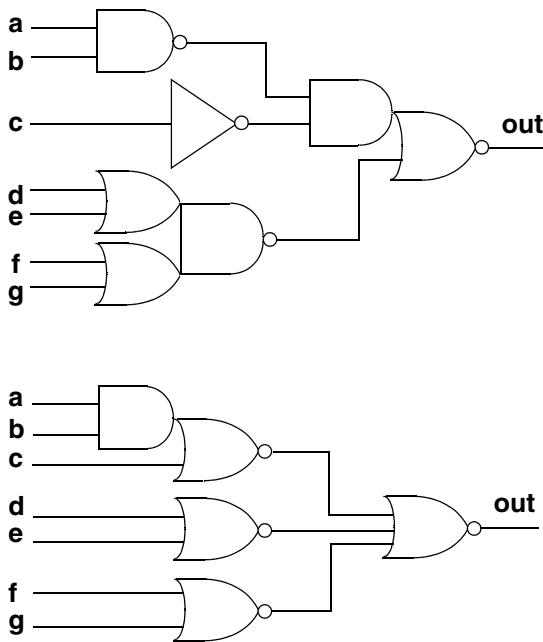
Technology-Independent Optimization

Technology-independent optimization applies algebraic and Boolean techniques to a set of logic equations. This optimization implements the logic equations to meet your timing and area goals.

Mapping

During mapping, Design Compiler selects components from the logic library to implement the logic structure. Design Compiler tries different logic combinations, using only components that approach the defined speed and area goals. [Figure 11-2](#) shows examples of mapped gates.

Figure 11-2 Mapped Gates



Technology-Specific Optimization

Technology-specific optimization synthesizes a gate-level design that attempts to meet your timing and area constraints.

Sequential Optimization

Sequential optimization includes the initial optimization phase, which maps sequential cells to cells in the library, and the final optimization phase, where Design Compiler optimizes timing-critical sequential cells (cells on the critical path):

- [Initial Sequential Optimization](#)
- [Final Sequential Optimization](#)

Initial Sequential Optimization

Initial sequential optimization maps sequential cells to cells in the target library. You can map to either standard sequential cells or scan-equivalent cells. Initial sequential optimization takes place during the first phase of gate-level optimization.

At this point in the optimization process, information about the delay through the combinational logic is incomplete. Design Compiler does not have enough information to select the optimum sequential cell. The tool can correct this lack of information later, in the final sequential optimization phase.

Final Sequential Optimization

Design Compiler has accurate values for all delays through the I/O pads and combinational logic before it enters the final sequential optimization phase. In this phase, Design Compiler optimizes timing-critical sequential cells (cells on the critical path).

The tool examines each sequential cell and its surrounding combinational logic to determine whether they might be replaced by more-optimal sequential cells from the target library in order to meet timing and area constraints.

Final sequential optimization can achieve the following:

- Improve design timing by choosing higher-performance sequential cells.
- Possibly reduce area and delay in the design if more optimal sequential cells exist in the library. The tool incorporates the combinational logic in the sequential cell, as shown in [Figure 11-3](#) and [Figure 11-4](#).
- Further improve area by remapping sequential elements.

Figure 11-3 Sequential Optimization

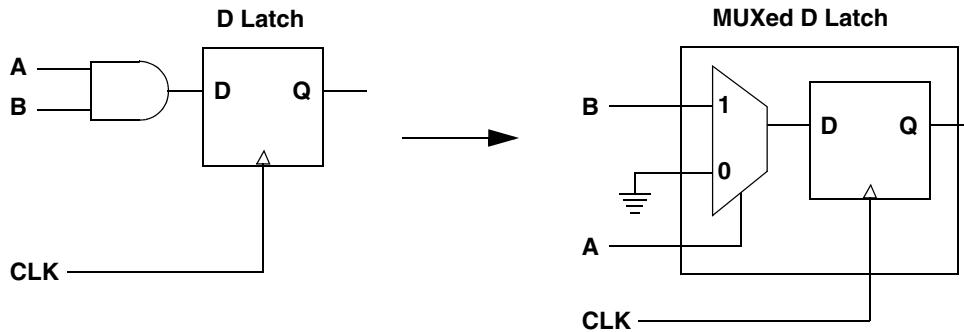
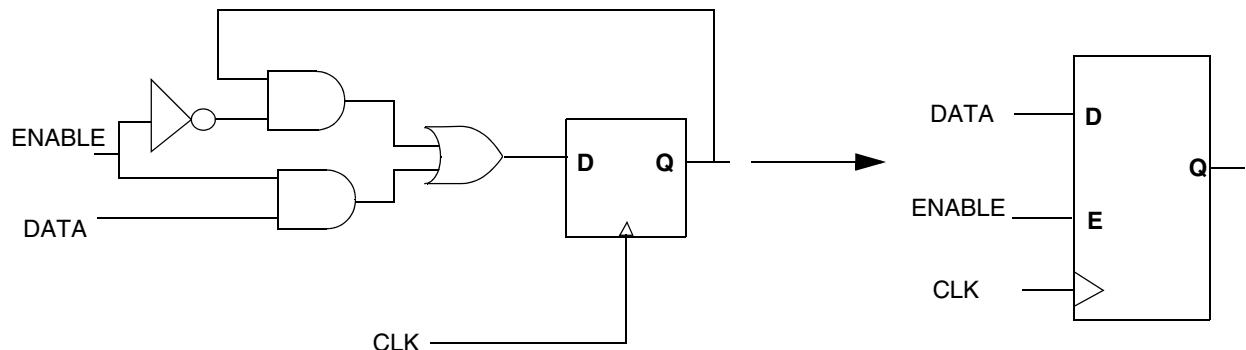
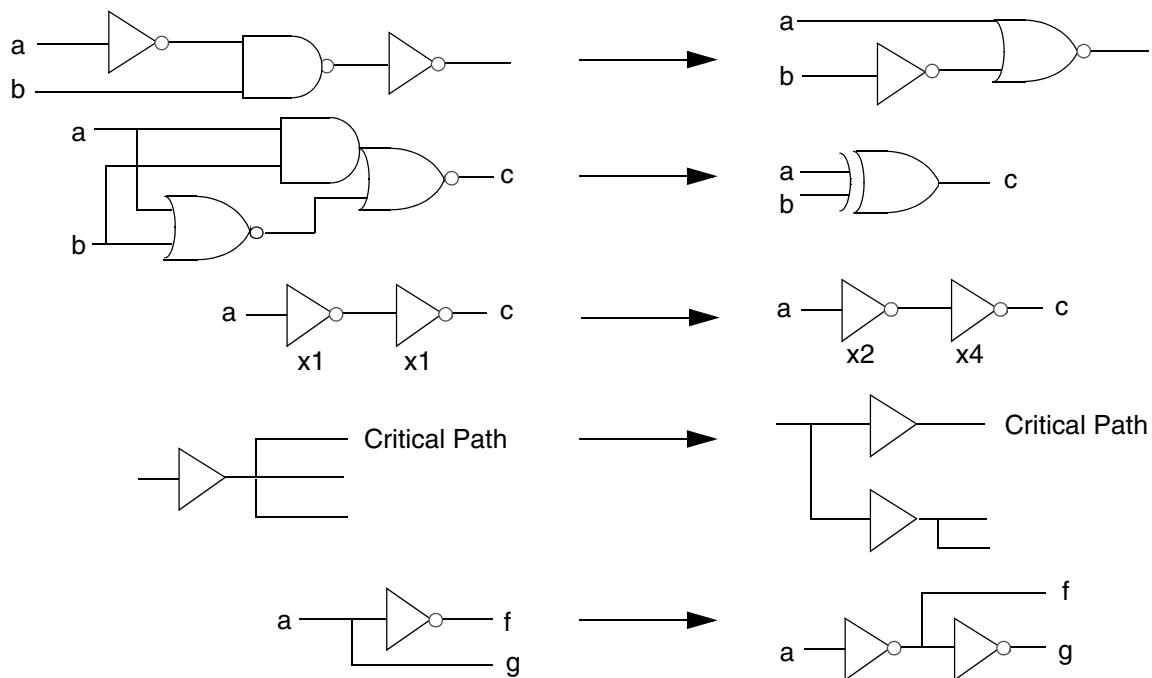


Figure 11-4 Sequential Optimization

Local Optimizations

The final step in gate-level optimization involves making local changes. Design Compiler makes incremental modifications to the design to adjust for timing or area.

[Figure 11-5](#) shows common local optimization steps. Design Compiler takes design rules into account during this phase. When two circuit solutions offer the same delay performance, Design Compiler implements the solution that has the lower design rule cost.

Figure 11-5 Local Optimization Steps

Compile Cost Function

During compile, Design Compiler's optimization algorithms use costs to determine if a design change is an improvement. Design Compiler calculates two cost functions, one for design rule constraints and one for optimization constraints, and accepts an optimization move if it decreases the cost of one component without increasing more-important costs. The tool reports the value of each cost function whenever a change is made to the design.

By default, Design Compiler prioritizes costs in the following order:

1. Design rule costs

- a. Connection class
- b. Multiple port nets
- c. Maximum transition time
- d. Maximum fanout
- e. Maximum capacitance
- f. Cell degradation

The goal of Design Compiler is to meet all constraints. However, by default, it gives precedence to design rule constraints because design rule constraints are functional requirements for designs. Using the default priority, Design Compiler fixes design rule violations even at the expense of violating your delay or area constraints.

2. Optimization costs

- a. Maximum delay
- b. Minimum delay
- c. Maximum power
- d. Maximum area

During the first phase of mapping, Design Compiler works to reduce the optimization cost function. Speed (timing) constraints automatically have a higher priority than area; therefore, an optimization move that improves maximum delay cost is always accepted. Optimization stops when all costs are zero or no further improvements can be made to the cost function.

The compile cost function considers only those components that are active in your design. Design Compiler evaluates each cost function component independently, in order of importance. When evaluating cost function components, Design Compiler considers only violators (positive difference between actual value and constraint) and works to reduce the cost function to zero.

You can change the priority of the maximum design rule costs and the delay costs by using the `set_cost_priority` command to specify the ordering. You must run the `set_cost_priority` command before compiling the design.

You can disable evaluation of the design rule cost function by using the `-no_design_rule` option or the optimization cost function by using the `-only_design_rule` option when you run the `compile` or `compile_ultra` command.

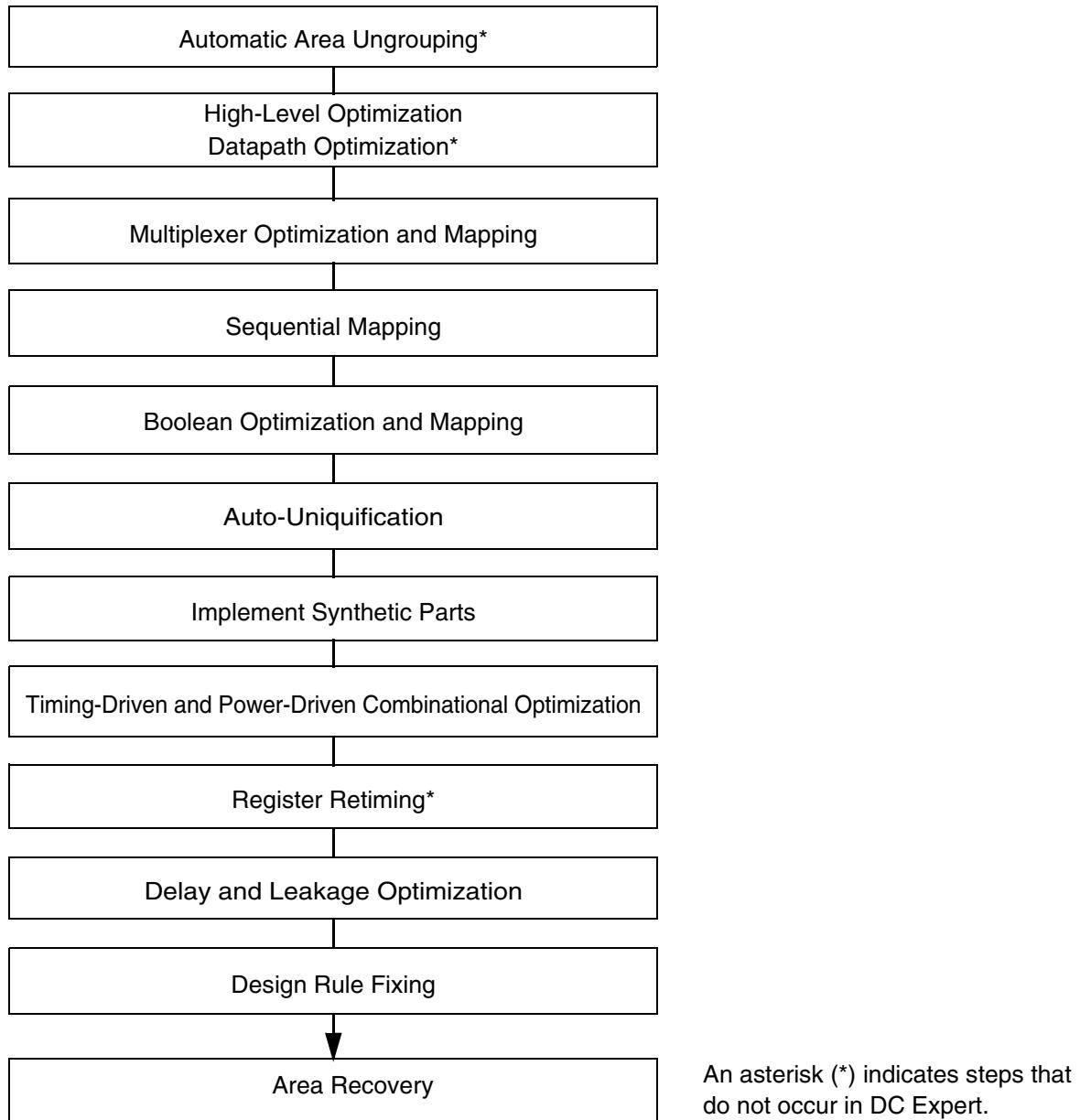
See Also

- [Managing Constraint Priorities](#)
- [Design Rule Constraints](#)
- [Optimization Constraints](#)

Optimization Flow

[Figure 11-6](#) shows the optimization flow. Steps that are not supported in the DC Expert flow are marked with an asterisk. The sections following the figure describe the steps in the flow.

Figure 11-6 Optimization Flow



Automatic Ungrouping

The `compile_ultra` command automatically ungroups logical hierarchies. Ungrouping merges subdesigns of a given level of the hierarchy into the parent cell or design. It removes hierarchical boundaries and allows Design Compiler to improve timing by reducing the levels of logic and to improve area by sharing logic. For information about the automatic ungrouping of hierarchies, see [Automatic Ungrouping](#).

You can also manually ungroup hierarchies by using the `ungroup` command or the `set_ungroup` command followed by `compile_ultra`. For information about manually ungrouping hierarchies, see [Removing Levels of Hierarchy](#).

High-Level Optimization and Datapath Optimization

During high-level optimization, resources are allocated and shared, depending on timing and area considerations. Resource sharing enables the tool to build one hardware component for multiple operations, which typically reduces the hardware required to implement your design. Additional optimizations, such as arithmetic optimization and the sharing of common subexpressions, are also performed. In addition, if you are using the DC Ultra tool, advanced datapath transformations are performed. For more information, see [High-Level Optimization and Datapath Optimization](#).

Multiplexer Mapping and Optimization

In this phase, Design Compiler maps combinational logic representing multiplexers in the HDL code directly to a single multiplexer (MUX) or a tree of multiplexer cells from the target logic library. Design Compiler reorders SELECT signals for better area and shares multiplexer trees. For more information, see [Multiplexer Mapping and Optimization](#).

Sequential Mapping

This phase consists of two steps: register inferencing by HDL Compiler and technology mapping by Design Compiler. The RTL is translated into a technology-independent representation called SEQGEN; the SEQGEN is then mapped to gates from the logic library. A SEQGEN is a generic sequential element that is used by Synopsys tools to represent registers and latches in a design. SEQGENs are created during elaboration and are mapped to flip-flops or latches during compile. For more information, see [Sequential Mapping](#).

Structuring and Mapping

In this phase, the tool optimizes unmapped unstructured logic and maps it to technology gates. Structuring is an optimization step that adds intermediate variables and logic structure to a design. During structuring, Design Compiler searches for subfunctions that can be factored out, then evaluates these factors based on the size of the factor and the number of times the factor appears in the design. The subfunctions that most reduce the logic are turned into intermediate variables and factored out of the design equations. Design Compiler offers timing-driven structuring to minimize delays and Boolean structuring to reduce area.

Automatic Uniquification

The uniquify process copies and renames any multiply referenced design so that each instance references a unique design. The process removes the original design from memory after it creates the new, unique designs. The original design and any collections that contain it or its objects are no longer accessible. The tool automatically uniquifies designs as part of the compile process. For more information about the uniquification process, see [The Uniquify Method](#).

Implementing Synthetic Parts

HDL operators (either built-in operators like + and * or HDL functions and procedures) are associated with synthetic parts, which are bound in turn to synthetic modules. Each synthetic module can have multiple architectural realizations, called implementations.

For example, when you use the HDL addition operator (+), HDL Compiler infers the need for an adder resource and puts an abstract representation of the addition operation in the netlist. During high-level optimization, the tool manipulates this abstract representation—called a synthetic operator—and applies optimizations such as arithmetic optimization or resource sharing.

During the high-level optimization phase, the tool used abstract representations for synthetic parts. During the implement synthetic parts phase, the tool maps synthetic modules to architectural representations (implementations). For more information about synthetic parts, see [Synthetic Operators](#) and the HDL Compiler and DesignWare documentation on SolvNet.

Timing-Driven Combinational Optimization

During this phase, Design Compiler performs optimization of combinational parts. This phase has two components: timing-driven structuring and incremental implementation selection. During timing-driven structuring, the tool restructures logic in the critical paths to improve delay cost. During incremental implementation selection, the tool explores alternative implementations for each synthetic operator. It evaluates and replaces synthetic implementations along the critical path to improve delay cost.

Register Retiming

Register retiming is available in DC Ultra only. Register retiming is a sequential optimization technique that moves registers through the combinational logic gates of a design to optimize timing and area. Register retiming adds an opportunity for improving circuit timing.

Design Compiler provides the following ways to perform register retiming:

- The `optimize_registers` command performs retiming of sequential cells (edge-triggered registers or level-sensitive latches) for pipelined designs. For more information about this retiming strategy, see [Pipelined-Logic Retiming](#).
 - The `compile_ultra` command supports the `-retime` option, which enables Design Compiler to automatically perform local retiming moves to improve worst negative slack (WNS). This capability, called adaptive retiming, optimizes an entire design. It works best with general non-pipelined logic. For more information about this retiming strategy, see [Adaptive Retiming](#).
-

Delay and Leakage Optimization

In this phase, Design Compiler attempts to fix existing delay violations by traversing the critical path. During delay optimization, Design Compiler reevaluates existing implementations to determine if they meet constraints. If constraints are not met, Design Compiler selects a different implementation. It applies local transformations such as upsizing, load isolation and splitting, and revisits mapping of sequential paths. Constraints affecting delay include clocks, input and output delays, external loads, input driving cells, operating conditions, and wire load tables. For more information, see [Delay Optimization](#).

During this stage of optimization, Design Compiler automatically performs leakage power optimization (except in DC Expert). During leakage power optimization, Design Compiler tries to reduce the overall leakage power in your design without affecting the performance. To do this, the optimization is performed on paths that are not timing-critical. When the target libraries are characterized for leakage power and contain cells characterized for multiple threshold voltages, Design Compiler uses the library cells with appropriate threshold voltages to reduce the leakage power of the design.

For more information about leakage power optimization for all Design Compiler tools, see [Leakage Power and Dynamic Power Optimization](#).

Design Rule Fixing

Design rules are provided in the vendor logic library to ensure that the product meets specifications and works as intended. In this phase, Design Compiler fixes any design rule violations. Whenever possible, Design Compiler fixes design rule violations by resizing gates across multiple logic levels—as opposed to adding buffers to the circuitry. For more information, see [Design Rule Fixing](#).

Area Optimization

During area optimization, Design Compiler attempts to minimize the number of gates in the design without degrading delay cost.

In DC Ultra (wire load mode and topographical mode), you can use the `compile_ultra` and `optimize_netlist -area` commands to provide significant area improvements. The `optimize_netlist -area` command performs monotonic gate-to-gate optimization to improve area without degrading timing or leakage.

You can use the `optimize_netlist -area` command as the final area recovery step at the end of the synthesis flow. You can also use the command on legacy netlists. To maximize area benefits, run synthesis using the `compile_ultra` command and then run the `optimize_netlist -area` command. For more information, see [Area Recovery](#).

Optimization Techniques

The following topics provide techniques you can use to control optimization and improve results:

- [Optimizing Once for Best- and Worst-Case Conditions](#)
- [Optimizing With Multiple Libraries](#)
- [Preserving Subdesigns](#)
- [Preserving the Clock Network After Clock Tree Synthesis](#)
- [Optimizing Datapaths](#)
- [Creating Path Groups](#)
- [Controlling Automatic Path Group Creation](#)

- [Isolating Input and Output Ports](#)
- [Fixing Heavily Loaded Nets](#)
- [Fixing Nets Connected to Multiple Ports](#)
- [Optimizing Buffer Trees](#)
- [Optimizing Multibit Registers](#)
- [Optimizing for Multiple Clocks Per Register](#)
- [Defining a Signal for Unattached Master Clocks](#)
- [Optimizing Total Negative Slack During Placement](#)

Optimizing Once for Best- and Worst-Case Conditions

You can constrain your design once for both minimum (best-case) and maximum (worst-case) optimization and timing analysis. The tool then optimizes and analyzes the timing in a single compile run.

You can constrain the design by using either a single logic library or multiple libraries. Whether you use one or multiple libraries, the general methodology is as follows:

1. Set up a logic library file or files. Make sure the files contain
 - Best- and worst-case operating conditions
 - Optimistic and pessimistic wire load models
 - Minimum and maximum timing delaysIf you use multiple libraries, see [Optimizing With Multiple Libraries](#).
2. Specify minimum and maximum constraints.
 - Environmental—including operating conditions and wire loads
 - Clock information—including clock skew and clock transition
 - Optimization—including input and output delays, drive, load, and resistance
 - Design rule—including transition time, fanout, and capacitance
3. Optimize the design for simultaneous minimum and maximum timing. To ensure that minimum delay constraints are optimized with respect to a particular clock, specify the `fix_hold` attribute for that clock, using the `set_fix_hold` command.
4. Report and analyze the paths showing constraint violations.

The following constraint-related, reporting, and back-annotation commands support both minimum and maximum optimization and timing analysis.

Constraint-Related Commands

```
set_min_library  
set_operating_conditions  
set_wire_load_model  
set_wire_load_mode  
set_wire_load_min_block_size  
set_wire_load_selection_group  
set_clock_uncertainty  
set_clock_transition  
set_drive  
set_load  
set_port_fanout_number  
set_resistance
```

The `set_min_library` command is described in the next section, [Optimizing With Multiple Libraries](#).

Reporting Commands

```
report_annotated_delay  
report_area  
report_attribute  
report_bus  
report_cache  
report_cell  
report_clock  
report_clusters  
report_compile_options  
report_constraint  
report_delay_calculation  
report_design  
report_design_lib  
report_fsm  
report_hierarchy  
report_internal_loads  
report_lib  
report_name_rules  
report_net  
report_path_group  
report_port  
report_power  
report_qor  
report_reference  
report_resources  
report_synlib  
report_timing  
report_timing_requirements
```

```
report_transitive_fanin
report_transitive_fanout
report_wire_load
```

Other commands, including `report_design`, `report_port`, and `report_wire_load`, generate reports on the minimum and maximum constraints on your design. You do not need to specify the `-min` option.

Optimizing With Multiple Libraries

The `set_min_library` command directs Design Compiler to use multiple technology libraries for minimum- and maximum-delay analyses in one optimization run. Thus, you can choose libraries that contain all of the following:

- Best- and worst-case operating conditions
- Optimistic and pessimistic wire load models
- Minimum and maximum timing delays

You can direct Design Compiler to analyze them simultaneously. To accomplish this analysis, use the `set_min_library` command to create a link between the data in the two libraries. Use the library file name (not the library name) with the `set_min_library` command.

The `set_min_library` command creates a minimum/maximum relationship between two library files. You specify a `max_library` to be used for maximum delay analysis and a `min_library` to be used for minimum delay analysis. Only `max_library` should be used for linking and as the target library.

When Design Compiler needs to compute a minimum delay value, it first analyzes the library cell in the `max_library`, then it looks to the `min_library` to determine if a match exists. If a library cell with the same name, the same pins, and the same timing arcs exists in the `min_library`, Design Compiler uses the timing information from the `min_library`. If the tool cannot find a matching cell in the `min_library`, it uses the cell in the `max_library`.

[Example 11-1](#) shows how you might use `set_min_library` with `set_operating_conditions` to control and report delay analysis. If you do not specify a minimum, Design Compiler uses the maximum condition for both minimum and maximum delay analysis. You cannot use the `-min` option without also using the `-max` option.

Example 11-1 Controlling and Reporting Delay Analysis

```
set link_library "LIB_WC_COM.db"
set target_library $link_library
set_min_library LIB_WC_COM.db -min_version LIB_BC_COM.db
set_operating_conditions -max WC_COM -min BC_COM
source minmax.cons
set_fix_hold clk
```

```
compile
report_timing -delay_type max
report_timing -delay_type min
```

Preserving Subdesigns

You can preserve a subdesign during optimization by using the `set_dont_touch` command. The command places the `dont_touch` attribute on cells, nets, references, and designs in the current design to prevent these objects from being modified or replaced during optimization.

To control the preservation of subdesigns during optimization, see

- [Preserving Cells, References, and Designs](#)
- [Preserving Nets](#)
- [Removing a dont_touch Setting](#)

Preserving Cells, References, and Designs

Use the `set_dont_touch` command on subdesigns that you do not want optimized with the rest of the design hierarchy. The `dont_touch` attribute does not prevent or disable timing through the design.

When you use the `set_dont_touch` command on cells, references, and designs in the current design, consider the following:

- Setting the `dont_touch` attribute on a hierarchical cell sets an implicit `dont_touch` on all cells below that cell.
- Setting the `dont_touch` attribute on a library cell sets an implicit `dont_touch` on all instances of that cell.
- Setting the `dont_touch` attribute on a reference sets an implicit `dont_touch` on all cells using that reference during subsequent optimizations of the design.
- Setting the `dont_touch` attribute on a design has an effect only when the design is instantiated within another design as a level of hierarchy. In this case, the `dont_touch` attribute on the design implies that all cells under that level of hierarchy are subject to the `dont_touch` attribute. Setting `dont_touch` on the top-level design has no effect because the top-level design is not instantiated within any other design.
- You cannot manually or automatically ungroup objects marked as `dont_touch`. That is, the `ungroup` command and the `compile -ungroup_all` and `-auto_ungroup` options have no effect on objects marked as `dont_touch`.

Preserving Nets

When you use the `set_dont_touch` command on nets in the current design, consider the following:

- By default, if you set a `dont_touch` attribute on a net, Design Compiler sets an implicit `dont_touch` only on mapped combinational cells that are connected to that net. If the net is connected only to unmapped cells, optimization might remove the net.

The `dont_touch` attribute is ignored on nets that have unmapped cells on them. During compilation, Design Compiler issues warnings for `dont_touch` nets connected to unmapped cells (generic logic).

- You can preserve specific nets throughout the compilation process by setting the `enable_keep_signal_dt_net` variable to `true` before using the `set_dont_touch` command on the net. When you do this, the `dont_touch` attribute on the net sets an implicit `size_only` attribute on logic connected to that net even if logic connected to it is unmapped and not combinational.

Warning:

Preserving specific nets throughout compilation can cause quality of results (QoR) degradation. If you preserve a net that is in the critical path, the QoR degradation can be severe. This net preservation functionality is intended to facilitate verification. It should not be used when you are trying to achieve final QoR goals.

Design Compiler issues warnings during compilation to indicate nets that are user-preserved.

In the following example, the `enable_keep_signal_dt_net` variable is enabled to preserve the `my_net` net throughout compilation.

1. Make sure the net to be preserved is present in the design and then use the `enable_keep_signal_dt_net` variable and the `set_dont_touch` command to preserve the net:

```
prompt> get_nets *my_net*
prompt> set enable_keep_signal_dt_net true
prompt> set_dont_touch [get_nets *my_net*] true
```

2. Run the `compile_ultra` command:

```
prompt> compile_ultra
```

Check for OPT-154 warning messages at the beginning of compilation to indicate that the net is preserved:

```
Warning: Preserving net 'my_net'. Expect QoR impact. (OPT-154)
```

3. Make sure the net is preserved after compilation:

```
prompt> get_nets *my_net*
```

Removing a dont_touch Setting

To remove the `dont_touch` attribute, use the `remove_attribute` command or the `set_dont_touch` command set to `false`.

Preserving the Clock Network After Clock Tree Synthesis

To preserve a clock network after clock tree synthesis, use the `set_dont_touch_network` command. This command sets a `dont_touch_network` attribute on a net group. When placed on a clock tree, the `dont_touch_network` attribute ensures that your clock network is preserved during subsequent optimizations.

To list the clock networks in the design, use the `report_transitive_fanout -clock_tree` command.

Starting at the specified source object, the `set_dont_touch_network` command propagates the `dont_touch_network` attribute throughout the hierarchy of the clock network. By default, the propagation stops at output ports, or at sequential components if setup and hold relationships exist. If you use the `-no_propagate` option, the propagation stops at any logical cell.

The propagation of the `dont_touch_network` attribute occurs only in a forward direction, starting from the specified source object and spreading to objects driven by the source. The propagation cannot go backward, even to electrically connected nets in the same net group. This method of propagation highlights an important difference between the `set_dont_touch_network` and the `set_ideal_network` commands: the `set_ideal_network` command propagates in both the forward and backward directions.

For example, if your design is represented by the simple circuit in [Figure 11-7](#), and you issue the following command:

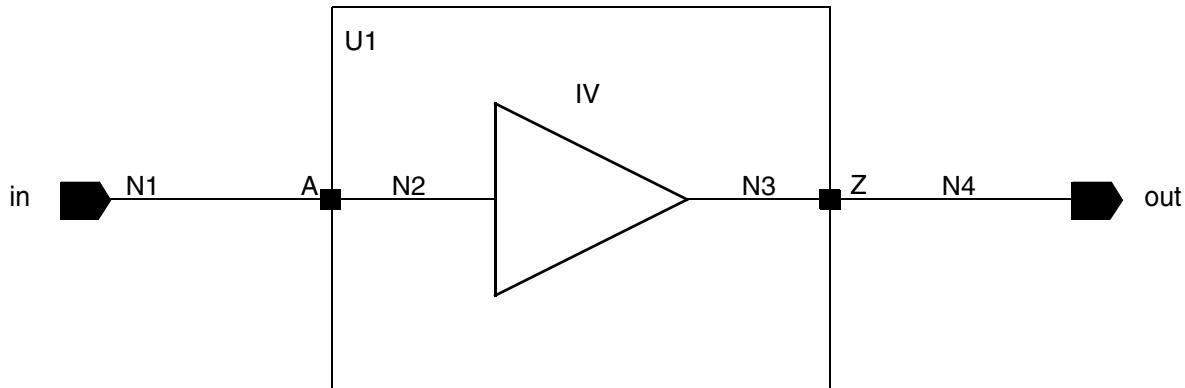
```
set_dont_touch_network U1/A
```

the `dont_touch_network` attribute is propagated from U1/A to U1/N2, U1/IV, U1/N3, and N4. If you issue the command

```
set_dont_touch_network U1/A -no_propagate
```

the `dont_touch_network` parameter is propagated from U1/A to U1/N2. It is not propagated past net U1/N2.

Figure 11-7 `set_dont_touch_network -no_propagate` option



Note:

Use the `set_auto_disable_drc_nets` command when you do not have an accurate representation of the clock tree and you want Design Compiler to treat the clock net as ideal. Typically, you would use this command when a clock tree has not yet been inserted in your design.

The `set_dont_touch_network` command cannot be used if the network has unmapped logic.

You can use the `get_attribute` command to check if an object has either the `dont_touch_network` or the `dont_touch_network_no_propagate` attribute.

See Also

- [Using Attributes](#)

Optimizing Datapaths

Datapath design is commonly used in applications that contain extensive data manipulation, such as 3-D, multimedia, and digital signal processing (DSP). Datapath optimization is comprised of two steps: *Datapath extraction*, which transforms arithmetic operators (for example, addition, subtraction, and multiplication) into datapath blocks, and *datapath implementation*, which uses a datapath generator to generate the best implementations for these extracted components. Datapath optimization is enabled by default when you use the `compile_ultra` command.

Datapath optimization provides the following benefits:

- Shares datapath operators
- Extracts the datapath
- Explores better solutions that might involve a different resource-sharing configuration
- Allows the tool to make better tradeoffs between resource sharing and datapath optimization

See Also

- [Datapath Extraction](#)
- [Datapath Implementation](#)

Creating Path Groups

By default, Design Compiler groups paths based on the clock controlling the endpoint (all paths not associated with a clock are in the default path group). If your design has complex clocking, complex timing requirements, or complex constraints, you can create path groups to focus Design Compiler on specific critical paths in your design.

Use the `group_path` command to create path groups. The `group_path` command allows you to

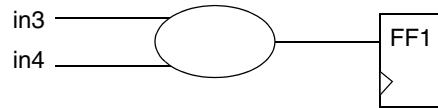
- Control the optimization of your design
- Optimize near-critical paths
- Optimize all paths

Controlling the Optimization of Your Design

You can control the optimization of your design by creating and prioritizing path groups, which affect only the maximum delay cost function. By default, Design Compiler works only on the worst violator in each group.

You set path group priorities by assigning weights to each group. The weight can be from 0.0 to 100.0. The default weight is 1.0. Do not use very small values (for example, 0.0001); smaller values can prevent the tool from implementing small improvements to the design during compilation.

For example, [Figure 11-8](#) shows a design that has multiple paths to flip-flop FF1.

Figure 11-8 Path Group Example

To indicate that the path from input in3 to FF1 is the highest-priority path, use the following command to create a high-priority path group:

```
prompt> group_path -name group3 -from in3 -to FF1/D -weight 2.5
```

Optimizing Near-Critical Paths

When you add a critical range to a path group, you change the maximum delay cost function from worst negative slack to critical negative slack. The critical range negative slack method considers all violators in each path group that are within a specified delay margin (referred to as the critical range) of the worst violator. Design Compiler optimizes all paths within the critical range.

For example, if the critical range is 2.0 ns and the worst violator has a delay of 10.0 ns, Design Compiler optimizes all paths that have a delay between 8.0 and 10.0 ns.

The critical range negative slack is the sum of all negative slack values within the critical range for each path group. When the critical range is large enough to include all violators, the critical negative slack is equal to the total negative slack.

Using the critical negative slack method, the equation for the maximum delay cost is

$$\sum_{i=1}^m \left(\left\langle \sum_{j=1}^n v_{ij} \right\rangle \times w_i \right)$$

m is the number of path groups.

n is the number of paths in the critical range in the path group.

v_{ij} is a violator within the critical range of the *i* th path group.

w_i is the weight assigned to the *i* th path group.

Design Compiler calculates the maximum delay violation for each path within the critical range as

```
max (0, (actual_path_delay - max_delay))
```

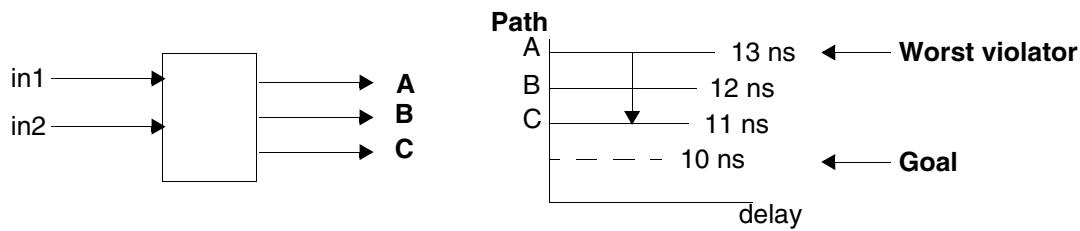
Specifying a critical range might increase runtime. To limit the runtime increase, use critical range only during the final implementation phase of the design, and use a reasonable critical range value. A guideline for the maximum critical range value is 10 percent of the clock period.

Use one of the following methods to specify the critical range:

- Use the `-critical_range` option of the `group_path` command.
- Use the `set_critical_range` command.

For example, [Figure 11-9](#) shows a design with three outputs, A, B, and C.

Figure 11-9 Critical Range Example



Assume that the clock period is 20 ns, the maximum delay on each of these outputs is 10 ns, and the path delays are as shown. By default, Design Compiler optimizes only the worst violator (the path to output A). To optimize all paths, set the critical delay to 3.0 ns. For example,

```
create_clock -period 20 clk
set_critical_range 3.0 $current_design
set_max_delay 10 {A B C}
group_path -name group1 -to {A B C}
```

Optimizing All Paths

You can optimize all paths by creating a path group for each endpoint in the design. Creating a path group for each endpoint enables total negative slack optimization but results in long compile runtimes.

Use the following script to create a path group for each endpoint:

```
set endpoints \
    [add_to_collection [all_outputs] \
    [all_registers -data_pins]]
foreach_in_collection endpt $endpoints {
    set pin [get_object_name $endpt]
    group_path -name $pin -to $pin
}
```

Controlling Automatic Path Group Creation

Design Compiler allows you to control the automatic creation of path groups in the design to improve quality of results (QoR). You can control whether the tool creates the path groups before optimization or only for blocks that fail timing after optimization.

- To create path groups before optimization, run the `create_auto_path_groups` command after reading the RTL, analyzing and elaborating, and reading the SDC constraints:

```
prompt> create_auto_path_groups -mode rtl
prompt> compile_ultra
prompt> remove_auto_path_groups
prompt> report_qor
```

This creates one path group per hierarchy within the design. If there are multiple levels of hierarchy, the tool creates path groups for each level.

The advantage of creating path groups at this stage is that the tool can perform all optimizations during the initial compile. However, the number of hierarchical levels within the design can be large, leading to a large number of path groups and causing excessive runtime.

You can prevent the creation of path groups for hierarchies that have a small number of registers by using the `-min_regs_per_hierarchy` option. This option specifies the minimum number of registers per hierarchy. No path group is created for hierarchies with a number of registers lower than this value.

- To create path groups after the design completes the initial compile, run the `create_auto_path_groups` command after the `compile_ultra` command:

```
prompt> compile_ultra
prompt> create_auto_path_groups -mode mapped
prompt> compile_ultra -incremental
prompt> remove_auto_path_groups
prompt> report_qor
```

This creates one path group per hierarchy only for those hierarchical blocks in the design that do not meet timing. If multiple levels of hierarchy do not meet timing, the tool creates path groups for each of those levels.

The tool creates fewer path groups when you use the `-mode mapped` option compared to the `-mode rtl` option because the tool assigns separate path groups only to those hierarchical blocks that fail timing. As a result, the runtime is reduced. In most cases, use the `create_auto_path_groups` command with the `-mode mapped` option after the initial compile.

The `create_auto_path_groups` command automatically creates specific path groups for I/Os, macros, and integrated clock-gating cells. To prevent the creation of specific path

groups, specify the `-exclude` option with any combination of the following values: `IO`, `macro`, and `ICG`. For example,

```
prompt> create_auto_path_groups -mode rtl -exclude {IO macro}
```

The `create_auto_path_groups` command saves user-created path groups before automatically creating path groups. To remove the path groups created by the `create_auto_path_groups` command, use the `remove_auto_path_groups` command. The `remove_auto_path_groups` command does not remove the user-created path groups.

Use the `remove_auto_path_groups` command in the following cases:

- After the initial or incremental compile to get a QoR report with only the user-created path groups.
- After the initial compile to remove the path groups created with the `create_auto_path_groups -mode rtl` command before creating new path groups with the `create_auto_path_groups -mode mapped` command.

Controlling Path Group Weights During High-Effort Area Optimization

You can ignore the path group weights during high-effort area optimization by setting the `optimize_area_ignore_path_group_weights` variable to `true`. When you enable the variable, the weights of all path groups are set to 1 when the `optimize_netlist -area` command performs monotonic gate-to-gate optimization to improve area. After area optimization, the weight on each path group is restored to the original user-specified value. By default, the variable is `false`.

Isolating Input and Output Ports

You can isolate input and output ports to improve the accuracy of timing models. To insert isolation logic at specified input or output ports, use the `set_isolate_ports` command.

Input ports are isolated in the following cases:

- When they drive one or more input pins of a cell having several input pins (as a result of boundary optimization)
- When they drive one or more input pins belonging to different cells having several input pins

Output ports are isolated in the following cases:

- To ensure that the cell driving an output port does not also drive some internal logic within the design

- To specify particular driver cells at the output ports

This is useful when you want each output port to be driven explicitly by its own driver (no sharing of output drivers by two or more ports) or when you want to compile the design in the context of the environment in which the design will be used.

The isolation logic can be a buffer or a pair of inverters. Either Design Compiler selects the buffer or inverter from the target library, or you specify a particular cell from the target library. However, a user-specified cell must be a buffer or an inverter. Otherwise, Design Compiler outputs an error message.

The inserted isolation logic has the `size_only` attribute assigned to it. Therefore, when you compile the design, only sizing optimization is allowed for this logic. If the isolation logic is composed of an inverter pair, the `size_only` attribute is assigned only to the second inverter, which allows flexibility in optimizing the first inverter.

It is important to understand that port isolation can be applied only to the input or output ports of the *current* design. Therefore, to apply port isolation to a subdesign of your top-level design, you must first make the subdesign the current design.

Port isolation is currently intended for use only during bottom-up compile. That is, isolating hierarchical instance pins of lower-level designs from the top-level design in a top-down compilation is not supported.

Also, in a bottom-up compilation, you cannot simply isolate the ports of a subblock that you have temporarily designated as the current design and then expect that isolation logic automatically to propagate upward when you compile the top-level design. To ensure that the isolation logic of a subblock remains while the top-level design is compiled, you must use the `propagate_constraints` command to propagate the constraints upward after the subblock ports are isolated and before you compile the top-level design.

Port isolation does not work if

- A `dont_touch` net is connected to the port
- The specified isolation cell is not in the target library
- The specified port is not an output or input port (inout ports and tristate ports are not supported)
- The specified type option (in the `set_isolate_ports` command) is not a buffer or an inverter

Examples

Issue the `set_isolate_ports` command before the `compile_ultra` command. The isolation logic is inserted before the design is compiled. To insert the isolation logic cell IVDAP on all output ports of the current design, enter the following command:

```
dc_shell> set_isolate_ports [all_outputs] -driver IVDAP
```

[Example 11-2](#) shows a sample script.

Example 11-2 Inserting Isolation Logic on Output Ports

```
set_app_var target_library lsi_10k.db
set_app_var link_library {* lsi_10k.db}
read_verilog ./t1.v
current_design test1
link
set_isolate_ports [all_outputs] -driver IVDAP
compile_ultra
```

The following script fragments shows you how to compile a subdesign with port isolation, followed by a top-level compile:

```
current_design test1
link
set_isolate_ports [all_outputs] -type buffer -force
compile_ultra
current_design top
propagate_constraints
compile_ultra
```

Removing and Reporting Port Isolation Cells

You can remove the port isolation attribute from designs by using the `remove_isolate_ports` command. The isolation cells are then removed during the next compile. You can also use the `remove_attribute` command.

To obtain a list of isolated input or output ports in a design, use the `report_isolate_ports` command. When you issue the `report_isolate_ports` command, you see a report similar to the following:

```
*****
Report : isolate_ports
Design : top
Version: ...
Date   : ...
*****

Port Name    Cell Name    Inst. Name    Type      Forced Insertion
-----
stp          IVDA         U35          buffer     yes
rhcp         IVP          U34          inverter   yes
```

```
hip_1      IVDA      U32      buffer      no
-----
```

The `report_constraint` and `report_compile_options` commands also provide information about the isolated input or output ports.

Other commands that support the port isolation feature are

- `reset_design`

This command removes the port isolation attribute, as well as other attributes from the design.

- `write_script`

This command writes any `set_isolate_ports` commands (along with the other `dc_shell` commands) into the script file.

Fixing Heavily Loaded Nets

Heavily loaded nets often become critical paths. To reduce the load on a net, you can use either of two approaches:

- If the large load resides in a single module and the module contains no hierarchy, fix the heavily loaded net by using the `balance_buffer` command. For example, enter

```
source constraints.con
compile_ultra
balance_buffer -from [get_pins buf1/Z]
```

Note:

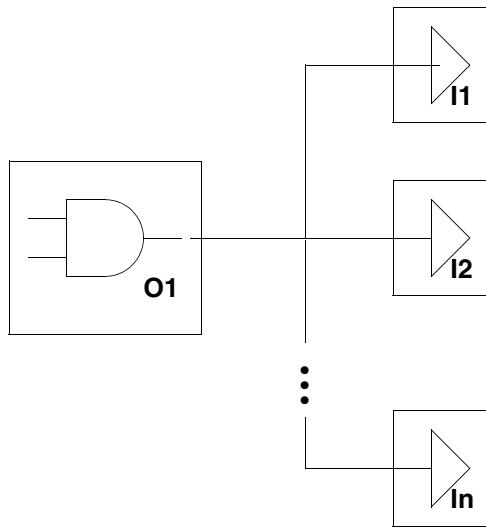
The `balance_buffer` command provides the best results when your library uses linear delay models. If your library uses nonlinear delay models, the second approach provides better results. In addition, the `balance_buffer` command is supported only in wire load mode, not in topographical mode.

- If the large loads reside across the hierarchy from several modules, apply design rules to fix the problem. For example,

```
source constraints.con
compile_ultra
set_max_capacitance 3.0
compile_ultra -only_design_rule
```

In rare cases, hierarchical structure might disable Design Compiler from fixing design rules.

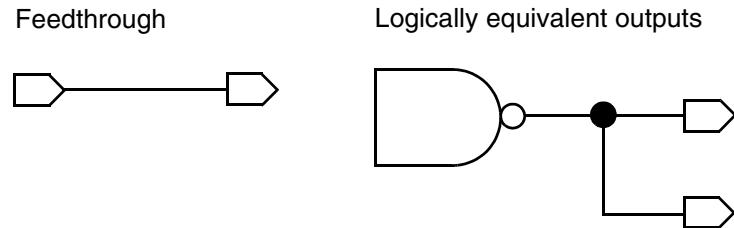
In the sample design shown in [Figure 11-10](#), net O1 is overloaded. To reduce the load, group as many of the loads (I1 through In) as possible in one level of hierarchy by using the `group` command or by changing the HDL. Then you can apply one of the approaches.

Figure 11-10 Heavily Loaded Net

Fixing Nets Connected to Multiple Ports

By default, Design Compiler does not buffer nets that are connected to multiple ports. As shown in [Figure 11-11](#), multiple-port connections are nets that connect:

- An input port to an output port (feedthrough) or
- Multiple output ports (logically equivalent outputs)

Figure 11-11 Multiple-Port Connection Types

To represent such nets, Design Compiler uses `assign` statements in the gate-level netlist. Back-end tools could potentially have problems with `assign` statements in the netlist.

To prevent multiple-port connections, use the `set_fix_multiple_port_nets` command to set the `fix_multiple_port_nets` attribute on a design.

Use the `set_fix_multiple_port_nets` command to control

- Feedthroughs

Specify the `-feedthroughs` option to insert buffers so that input ports are isolated from output ports. A feedthrough net occurs when an input port and output port are connected directly with no intervening logic.

- Multiple output ports

Specify the `-outputs` option to insert buffers so that no cell driver pin drives more than one output port.

- Constants

Specify the `-constants` option to duplicate constant logic so that no constant drives more than one output port. Alternatively, you can use the `-buffer_constants` option to buffer logic constants instead of duplicating them.

To exclude clock network nets from buffering during multiple-port nets fixing, use the `-exclude_clock_network` option.

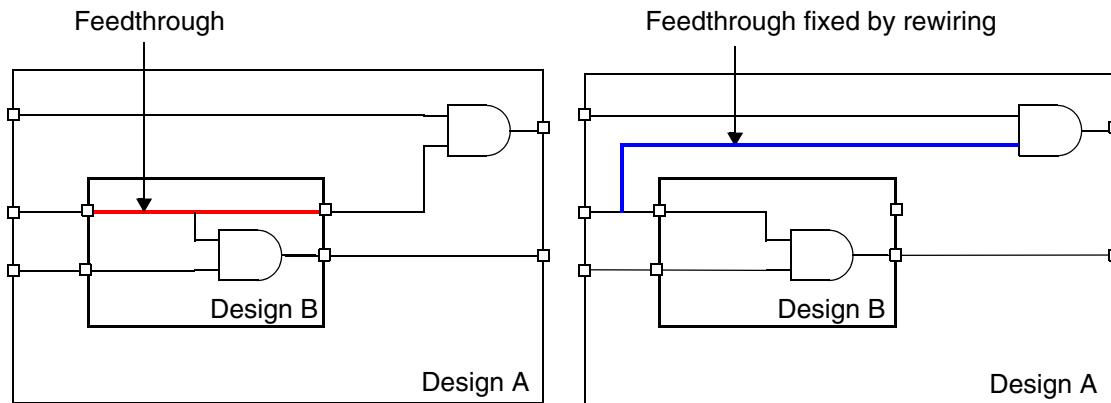
You can also fix multiple-port nets, feedthroughs, and constant-driven ports by rewiring the connections elsewhere in the hierarchy first. Any remaining feedthroughs, nets with logically equivalent outputs, and constants that could not be fixed by rewiring are fixed by buffering. To enable this feature, set the `compile_advanced_fix_multiple_port_nets` variable to `true`. It is set to `false` by default.

Note:

Design Compiler does not rewire across the hierarchy if boundary optimization is disabled.

The following figure shows a feedthrough in Design B that is fixed by rewiring up through the hierarchy as a result of the `compile_advanced_fix_multiple_port_nets` variable set to `true`. The feedthrough wire is moved up one level of hierarchy and reconnected to the net in Design A.

Figure 11-12 Feedthrough Fixed By Rewiring Up Through the Hierarchy



Optimizing Buffer Trees

Design Compiler allows you to build balanced buffer trees to fix design rule violations and improve timing delays caused by high-fanout nets. You can also generate a report for buffer trees and remove buffer trees.

Note:

The `balance_buffer` command is only supported in wire load mode. It is not supported in topographical mode.

For information about working with buffer trees, see the following topics:

- [Building Balanced Buffer Trees](#)
- [Reporting Buffer Trees](#)
- [Removing Buffer Trees](#)

Building Balanced Buffer Trees

You can build balanced buffer trees on user-specified nets and drivers of a mapped design by using the `balance_buffer` command. Use this command to fix design rule violations and improve timing delays caused by high fanout nets.

The `balance_buffer` command first removes any existing buffer tree on the specified net or driver and then builds a new buffer tree, free of design rule violations. The tree can span hierarchies downstream of the specified net or driver. However, the `balance_buffer` command does not change the hierarchical pin configuration to balance out loads across the hierarchies. Within each hierarchy, Design Compiler builds the buffer tree, taking into account the loads and drivers of the next downstream hierarchy.

The generated buffer tree consists of layered stages of buffers or inverters. Each stage uses the same buffer or inverter, and each gate of a given stage drives roughly the same load capacitance. The buffer and inverter cells are optimally chosen by Design Compiler from the logic library, unless you use the `-prefer` option to specify a particular cell from the logic library.

Technology libraries provide various kinds of delay models, including linear and nonlinear models. The `balance_buffer` command can use any delay model provided by the logic library to build the buffer tree.

When all loads have approximately the same value, balanced buffering creates an optimal buffer tree. When loads are not equal, balanced buffering still creates an optimal buffer tree with respect to the delay and design rule constraints, but the tree structure might not be balanced.

An input port driving a net to be buffered by `balance_buffer` must have its drive value set. If you omit the value, no buffer tree is created, because the default value implies that the input port has infinite drive.

Balanced buffering is constraint driven. It fixes design rules first, then optimizes for timing and area.

Note:

The `balance_buffer` command is not recommended for clock trees because the command does not take clock skew into account.

You can force buffer tree construction by using the `-force` option. However, if you do, the `balance_buffer` command might build buffer trees that worsen design cost. Note that if the design has no design rules or timing cost, buffer trees are not built even when the `-force` option is specified.

In the following example, a buffer tree is built first from the io port, and then two additional buffer trees are built to drive load1 and load2:

```
prompt> set_driving_cell -lib_cell IV io
prompt> balance_buffer -from io
prompt> balance_buffer -to {load1 load2}
```

To perform a functional verification or comparison between the initial mapped design and the mapped design after the `balance_buffer` command is run, use the Formality tool. For more information, see [Verifying Functional Equivalence](#).

Reporting Buffer Trees

You can generate a report for a buffer tree at the specified driver pins or driver nets by using the `report_buffer_tree` command. The command reports the full name of the driver pin, including the name of the cell, the name of the library cell, and the level of the driver, relative to the starting point. The starting point is defined as level 0 of the tree. For an example, see [Removing Buffer Trees](#).

Removing Buffer Trees

You can remove buffer trees at a specified driver pin, load pin, or driver net on a mapped design by using the `clean_buffer_tree` command.

To remove the entire buffer tree in the fanin up to a specified object, use the `-source_of` option with the `clean_buffer_tree` command. The object list must contain only output ports or input pins. You cannot use the `-source_of` option with the `-from`, `-to` or `-net` options.

The following example removes the entire buffer tree up to a particular pin.

1. Report the buffer tree from the b pin:

```
prompt> report_buffer_tree -from b
Driver (level 0): b (**in_port**)
Driver (level 1): abufa/Z
Driver (level 2): abufa_clk/Z
Driver (level 3): abufa_clk1/Z
Driver (level 4): abufa_clk2/Z
Load (level 5): reg1/CP
```

2. Remove the buffer tree up to the reg1/CP pin:

```
prompt> clean_buffer_tree -source_of reg1/CP
```

3. Report the buffer tree again from the b pin to view the modified buffer tree:

```
prompt> report_buffer_tree -from b
Driver (level 0): b (**in_port**)
Load (level 1): reg1/CP
```

Optimizing Multibit Registers

A multibit component is a group of cells with identical functionality. Two cells can have identical functionality even if they have different bit widths. Thus, a group of cells that includes one 3-bit register and one 5-bit register is a multibit component if the cells have identical functionality. A group of eight single-bit cells could also be considered a multibit component.

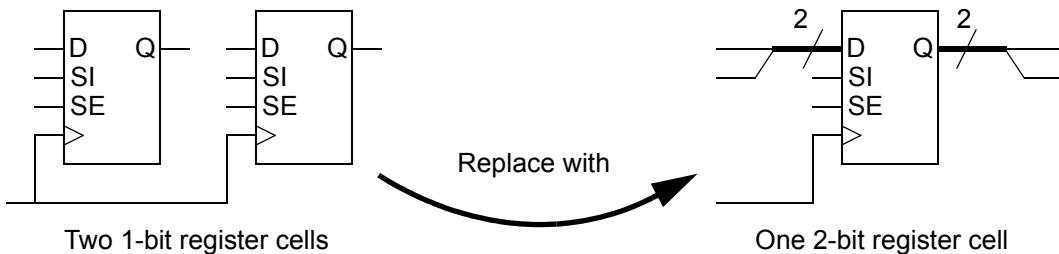
Design Compiler can replace single-bit register cells with multibit register cells if such cells are available in the logic library and physical library. Replacing single-bit cells with multibit cells reduces

- Clock tree power and the number of clock tree buffers
- Area due to shared transistors and optimized transistor-level layout
- The total clock tree net length

These benefits must be balanced against the loss of optimization flexibility in placing the register bits and routing the connections to them.

[Figure 11-13](#) shows how multiple single-bit registers can be replaced with a multibit register.

Figure 11-13 Replacing Multiple Single-Bit Register Cells With a Multibit Register Cell



The area of the 2-bit cell is less than that of two 1-bit cells due to transistor-level optimization of the cell layout, which might include shared logic, shared power supply connections, and a shared substrate well. The scan bits in the multibit register can be connected together in a chain as in this example, or each bit can have its own scan input and scan output.

Design Compiler supports the following methods for replacing single-bit register cells with multibit register cells:

- RTL bus inference flow (in wire load mode and topographical mode)

Design Compiler groups the register bits belonging to each bus (as defined in the RTL) into multibit components. You can also group bits manually by using the `create_multibit` command. The bits in each multibit component are targeted for implementation using multibit registers. The actual replacement of register bits occurs during execution of the `compile_ultra` command.

The following script shows a typical RTL bus inference flow for multibit register synthesis:

```
# Infer multibit registers for all buses during RTL reading
set hdlin_infer_multibit default_all

# Read RTL

# Manually assign bits to specific multibit registers
create_multibit -name my_multi_reg1 {reg_xy*}

# Choose timing-driven multibit register mapping
set_multibit_options -mode timing_driven

# Compile design with scan cell replacement and
# clock gating optimization
compile_ultra -scan -gate_clock
```

```
# Insert DFT scan logic
insert_dft

# Incremental compile to optimize scan chains
compile_ultra -incremental -scan
```

- Placement-based register banking flow (in Design Compiler Graphical)

Design Compiler Graphical groups single-bit register cells that are physically near each other into a register bank and replaces each register bank using one or more multibit register cells. This method works with both logically bussed signals and unrelated register bits that meet the banking requirements. The register bits assigned to a bank must use the same clock signal and the same control signals, such as preset and clear signals.

The following script example shows the placement-based multibit banking flow in Design Compiler Graphical:

```
# Read the RTL design
# Apply logic and physical constraints

# Enable placement-aware clock_gating
set_app_var power_cg_physically_aware_cg true

# Initial compile
compile_ultra -spg -scan -gate_clock
write_file -hierarchy -format ddc -output initial_compile.ddc

# Multibit register banking
identify_register_banks -input_map_file input.map \
    -output_file create_reg_bank.tcl \
    -register_group_file reg_group.txt \
    -common_net_pins {CP RN SN}
source create_reg_bank.tcl

# DFT scan insertion
set_scan_configuration ...
insert_dft

# Incremental compile
compile_ultra -scan -incremental -gate_clock -spg
write_scan_def -output mapped.scandef
check_scan_def
write_file -hierarchy -format ddc -output mapped.ddc
```

For more information about optimizing multibit registers, see the *Multibit Register Synthesis and Physical Implementation Application Note*.

Optimizing for Multiple Clocks Per Register

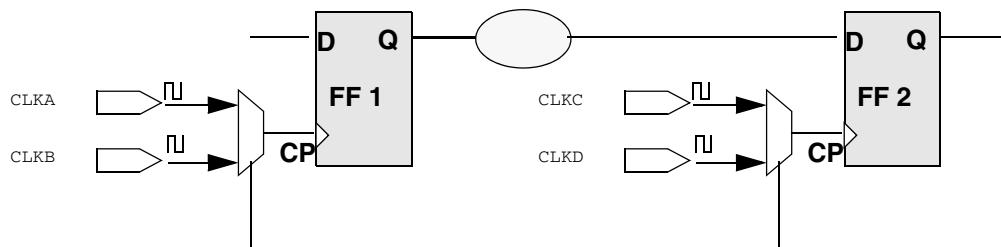
When a sequential device is driven by different clocks, you can either restrict the optimization to one clock at a time or allow multiple clocks to propagate to the sequential device.

To restrict the propagation of clocks so that exactly one reaches the register, you can use either of the following commands: `set_case_analysis` or `set_disable_timing`. You use these commands when you want to select a single active clock for timing analysis and optimization; this method requires multiple iterations for all clocks.

By default, the tool analyzes and optimizes the timing based on all clocks operating together, including interactions between different clocks, such as data launch by one clock and capture by another. To restrict operation to only one clock at a time, set the `timing_enable_multiple_clocks_per_reg` variable to `false`.

For example, consider the circuit in [Figure 11-14](#). By default, the tool considers the following clock interactions: CLKA to CLKC, CLKA to CLKD, CLKB to CLKC, and CLKB to CLKD.

Figure 11-14 Multiple Clocks per Register



To allow multiple clocks to propagate to a register:

1. Ensure that the `timing_enable_multiple_clocks_per_reg` variable is set to `true`.
2. Define multiple clocks at a pin or port by using the `-add` option with the `create_clock` command.

This option allows you to add clocks to the existing clocks; otherwise the existing clocks are overwritten.

If your design has generated clocks, use the following options with the `create_generated_clock` command:

- The `-add` option to specify multiple generated clocks on the same source or pin. Ideally, one generated clock must be specified for each clock that fans into the master pin.
- The `-master_clock` option to specify which clock is the master clock.

- The `-source` option to specify the clock source from which the clock is generated, that is, the pins or ports where the clock waveform is applied to the design.

Additionally, when you use the `-add` option along with the `create_clock` and `create_generated_clock` commands, you must use the `-name` option so that clocks that are defined on the same pin or port have unique names.

3. (Optional) Define a network latency for each clock signal that passes through a specific pin or port to a set of fanout registers.

When multiple clock signals traverse the same pins of a design, any register clock pins that are in the fanout of these clock signals are assigned the same network latency.

You can, however, set clock network latency on a pin or port with reference to a specific clock. To do so, use the `-clock` option with the `set_clock_latency` command. When computing the network latency along the path from a clock definition point to a register clock pin, Design Compiler uses the network latency value associated with the pin or port closest to the register, ignoring network latencies that do not reference the clock of interest.

Important:

For latch-based designs that have more than two clocks, it is strongly recommended that you set false paths between mutually exclusive clocks; otherwise you might observe longer runtime and higher memory usage. Unrelated clocks are those that do not interact with one another. For example, consider [Figure 11-14](#). By default, Design Compiler analyzes the interactions between all combinations of clocks. However, the logic enables only two possible interactions: CLKA to CLKC and CLKB to CLKD. Therefore, set false paths between the unrelated clocks as follows:

```
set_false_path -from CLKA -through FF1/CP -to CLKD
set_false_path -from CLKB -through FF1/CP -to CLKC
```

4. To report registers with multiple clock pins, use the `check_timing -multiple_clock` command and `report_clock` command.

The `check_timing` command generates a warning message if the `timing_enable_multiple_clocks_per_reg` variable is set to false and more than one clock signal reaches a register. The `report_clock` command provides you with the following information about a generated clock: the name of the master clock, the name of the master clock source pin, and the name of the generated clock pin.

Example

The following commands create two clocks on the same port and associate a network latency with each clock signal traversing a particular pin. The commands also set false paths between unrelated clocks.

```
create_clock -name CLKA -period 10 [get_ports CLK]
create_clock -name CLKB -period 8 -add [get_ports CLK]
set_clock_latency 1.16 -clock CLKA [get_pins b1/Z]
```

```
set_clock_latency 1.26 -clock CLKB [get_pins b1/Z]
set_input_delay .85 -clock CLKA [get_ports d]
set_input_delay .95 -clock CLKB -add_delay [get_ports d]
set_false_path -from CLKA -to CLKB
set_false_path -from CLKB -to CLKA
```

Defining a Signal for Unattached Master Clocks

Design Compiler can connect master clock pins to a specified signal when it translates or optimizes to flip-flops that have master- and slave-clock pins. In your HDL code, you describe the master-slave latch as a flip-flop by specifying only the slave clock. Specify the master clock as an input port but do not connect it. In addition, set the `clocked_on_also` attribute on the master clock port. Design Compiler then maps the logic to a master-slave cell in the library.

You use the `set_attribute` command to set the `signal_type` attribute to `clocked_on_also` on the master clock port. Design Compiler then maps the logic to a master-slave cell in the library. For multiple clock designs, you use the `-associated_clock` option to specify the associated slave clock.

Example 1

This example illustrates how you use the `set_attribute` command to describe a master-slave latch with a single master-slave clock pair.

```
module MSDFF (Q, D, MCLK, SCLK)
  input D, MCLK, SCLK;
  output Q;
  reg Q;

  //synopsys_dc_tcl_script_begin
  //set_attribute -type string MCLK signal_type
  //                clocked_on_also
  //set_attribute -type boolean MCLK level_sensitive true
  //synopsys_dc_tcl_script_end

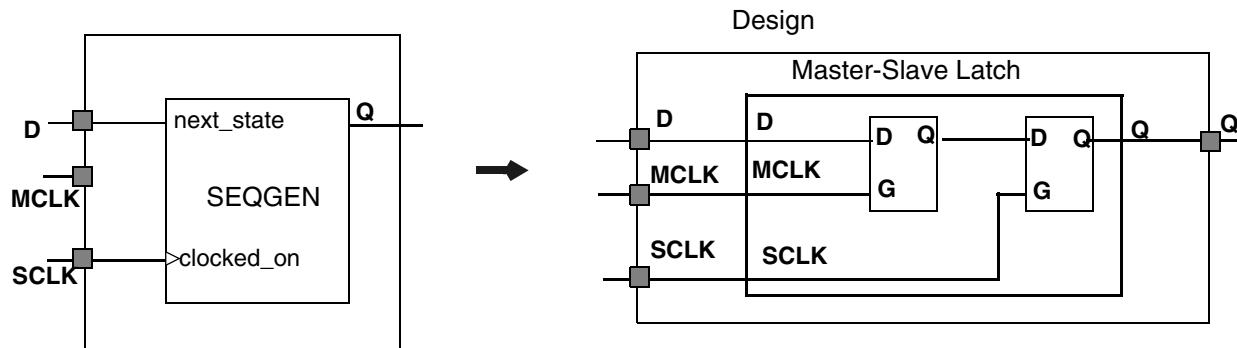
  always @ (posedge SCLK)
    Q <= DATA;
endmodule
```

Alternatively, instead of embedding the `set_attribute` command in the RTL, you can include the following commands in your script:

```
create_clock -period 10 [get_ports SCLK]
set_attribute -type string MCLK signal_type clocked_on_also
set_attribute -type boolean MCLK level_sensitive true
```

[Figure 11-15](#) shows the generic cell inferred by Design Compiler and the resultant master-slave latch after compile. As shown in the figure, unattached master clocks are connected to the MCLK port.

Figure 11-15 Generic Cell and Master-Slave Latch After Compilation



Example 2

This example illustrates how you use the `-associated_clock` option to specify the associated slave clock in multiple clock designs. Slave clock port SCK1 and master clock port MCK1 are paired; SCK2 and MCK2 are also paired. The `compile_ultra` or `compile` command automatically connects unconnected master clock pins of cells in the fanout of SCK1 to port MCK1.

```
prompt> set_attribute -type string MCK1 signal_type clocked_on_also
prompt> set_attribute -type boolean MCK1 level_sensitive true
prompt> set_attribute -type boolean MCK1 associated_clock SCK1
prompt> set_attribute -type string MCK2 signal_type clocked_on_also
prompt> set_attribute -type boolean MCK2 level_sensitive true
prompt> set_attribute -type boolean MCK2 associated_clock SCK2
```

See Also

- The HDL Compiler documentation
Provides information about describing master-slave latches

Optimizing Total Negative Slack During Placement

To control whether the tool optimizes the design for total negative slack or worst negative slack, set the following variables in Design Compiler topographical mode:

- `placer_tns_driven`

To optimize the design to reduce the total negative slack instead of the worst negative slack, set the `placer_tns_driven` variable to `true` before running the initial `compile_ultra` command. By default, the placer optimizes the worst negative slack.

- `placer_tns_driven_in_incremental_compile`

By default, the tool optimizes the total negative slack in the design only when the total negative slack exceeds a certain threshold. During incremental compile, to optimize the total negative slack even if the total negative slack is low, set the variable to `true`.

12

Optimizing Across Hierarchical Boundaries

Boundary optimization and automatic ungrouping are the strategies by which Design Compiler optimizes logic across subdesigns when the design has levels of hierarchy. Boundary optimization performs optimization across design hierarchies while retaining the levels of hierarchy and provides Design Compiler with opportunities to simplify the logic. Ungrouping removes levels of hierarchy and allows Design Compiler more freedom to share common terms in the design.

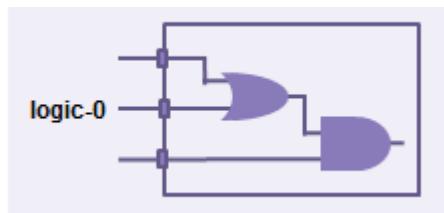
To learn how to control these optimization techniques, see the following topics:

- [Boundary Optimization](#)
- [Port Punching in Design Compiler Graphical](#)
- [Port Punching and Phase Inversion With Automatic High-Fanout Synthesis](#)
- [Other Optimizations That Affect Hierarchical Boundaries](#)
- [Automatic Ungrouping](#)

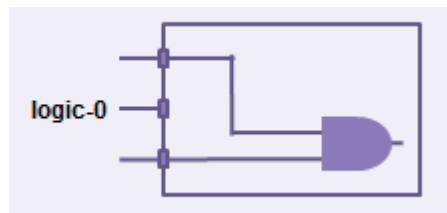
Boundary Optimization

The `compile_ultra` command performs the following types of boundary optimization by default:

- Propagation of constants across the hierarchy:

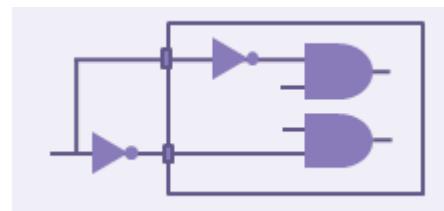


Without boundary optimization

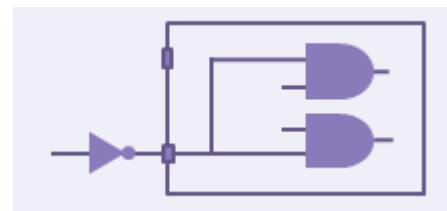


With boundary optimization

- Propagation of equal and opposite information across the hierarchy:

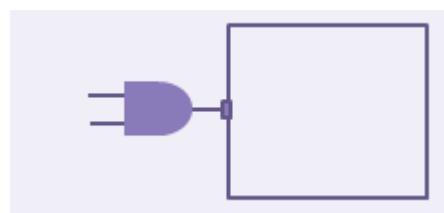


Without boundary optimization

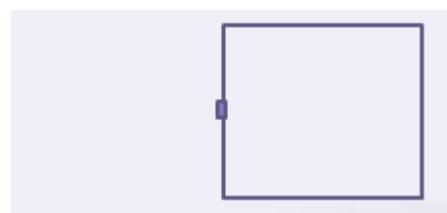


With boundary optimization

- Propagation of unconnected port information across the hierarchy:

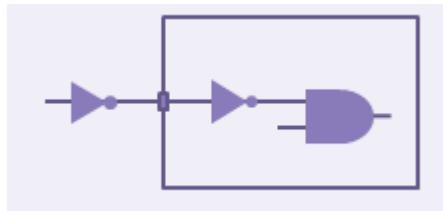


Without boundary optimization

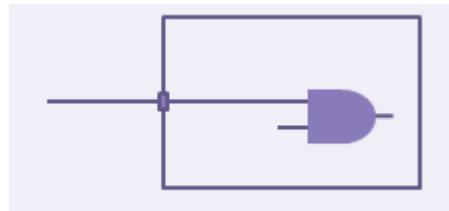


With boundary optimization

- Pushing of inverters across the hierarchy (phase inversion):



Without boundary optimization



With boundary optimization

To learn how to control boundary optimization, see the following topics:

- [Disabling Boundary Optimization Throughout the Design](#)
- [Disabling Boundary Optimization for a Specific Design](#)
- [Controlling Constant Propagation](#)
- [Controlling Phase Inversion](#)
- [Propagating Unconnected Registers and Unconnected Bits of Multibit Registers Across Hierarchies With Boundary Optimization Disabled](#)

Disabling Boundary Optimization Throughout the Design

To disable all types of boundary optimization for the entire design, use either of the following commands:

- `compile_ultra -no_boundary_optimization`

You do not need to use the `compile_preserve_subdesign_interfaces` variable to disable the downward and upward propagation of constant and equal and opposite information when you use the `-no_boundary_optimization` option with the `compile_ultra` command.

In addition, when you use the `compile_ultra` command with the `-no_boundary_optimization` option, the Design Compiler tool automatically sets the `-no_automunge` option, which disables automatic ungrouping for the entire design.

- `optimize_netlist -area -no_boundary_optimization`

The command disables boundary optimization across the design. However, any `set_boundary_optimization` command settings take higher priority and are honored.

Disabling Boundary Optimization for a Specific Design

To disable all types of boundary optimization on a list of objects (cells or designs), use the `set_boundary_optimization` command. When you run the command, specify the `false` argument for a specific cell or design to disable boundary optimization for that cell or design.

The following example shows how to use the `set_boundary_optimization` command to preserve hierarchical boundaries for a cell named U2:

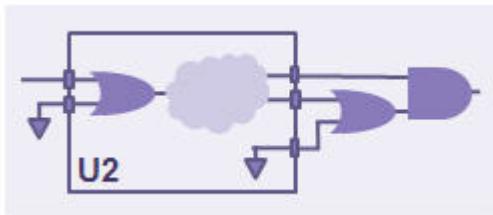
```
prompt> set_boundary_optimization [get_cells U2] false
```

When boundary optimization is disabled by the `set_boundary_optimization` command, only the downward propagation of constant and equal and opposite information is disabled. The information is not propagated from a design to its subdesign; however, the information is propagated from a subdesign to its parent design by default.

You can disable both downward and upward propagation of constant and equal and opposite information across the hierarchy by setting the `compile_preserve_subdesign_interfaces` variable to `true`. The default for this variable is `false`.

[Figure 12-1](#) shows a design before boundary optimization. [Figure 12-2](#) shows the same design after boundary optimization is performed by the `compile_ultra` command. In this example, the `boundary_optimization` attribute is set to `false` on cell U2 by the `set_boundary_optimization` command.

Figure 12-1 Design Before Boundary Optimization



The design on the left in [Figure 12-2](#) shows the default behavior, which disables only the downward propagation of constant and equal and opposite information. In the design on the right, both downward and upward propagation of constant and equal and opposite information is disabled by the `compile_preserve_subdesign_interfaces` variable set to `true`.

Figure 12-2 Design After Boundary Optimization With the `compile_ultra` Command



`compile_preserve_subdesign_interfaces` set to `false` (default) `compile_preserve_subdesign_interfaces` set to `true`

When you use the `set_boundary_optimization` command and specify the `false` argument, Design Compiler automatically prevents designs and cells from being ungrouped when you run the `compile_ultra` command.

Controlling Constant Propagation

Design Compiler automatically propagates constants, unconnected pins, and complement information when you use the `compile_ultra` command unless you disable boundary optimization. To learn how to control constant propagation when boundary optimization is disabled and how to disable constant propagation through specific hierarchical pins, see

- [Controlling Constant Propagation When Boundary Optimization is Disabled](#)
- [Disabling Constant Propagation Through Specific Hierarchical Pins](#)

Controlling Constant Propagation When Boundary Optimization is Disabled

Constant propagation is enabled by default even when boundary optimization is disabled. When the `compile_enable_constant_propagation_with_no_boundary_opt` variable is set to `true` (the default), constants are propagated across hierarchical boundaries even when boundary optimization is disabled. When the variable is set to `true`,

- The `compile_ultra -no_boundary_optimization` command performs constant propagation throughout the design.
- The `compile_ultra` command without the `-no_boundary_optimization` option performs constant propagation throughout the design, including the blocks that have the `boundary_optimization` attribute set to `false` on them by the `set_boundary_optimization` command.

To disable constant propagation along with other boundary optimizations across hierarchical boundaries, set the `compile_enable_constant_propagation_with_no_boundary_opt` variable to `false`.

If you disable boundary optimization of a subdesign to perform hierarchical formal verification, and you verify the subdesign and the rest of the design separately, allowing constant propagation on the subblock could cause verification problems. You need to set the `compile_enable_constant_propagation_with_no_boundary_opt` variable to `false` to disable constant propagation along with other boundary optimizations.

Disabling Constant Propagation Through Specific Hierarchical Pins

You can prevent the propagation of constants through specific hierarchical pins by using the `set_compile_directives` command with the `-constant_propagation` option set to `false` for a specific hierarchical pin or a list of hierarchical pins. When you do this, Design Compiler sets a `const_prop_off` attribute on the specified pins and disables constant propagation through them.

Controlling Phase Inversion

You can control phase inversion, which is the moving of inverters across hierarchical boundaries during boundary optimization, by using the `compile_disable_hierarchical_inverter_opt` variable.

To prevent inverters from moving across hierarchical boundaries during boundary optimization, set the `compile_disable_hierarchical_inverter_opt` variable to `true`. When the variable is set to `true`, Design Compiler does not move inverters across hierarchical boundaries even if doing so could improve the design. The variable is set to `false` by default, allowing phase inversion. This variable has no effect on the design when boundary optimization is disabled.

During phase inversion, Design Compiler adds a `_BAR` suffix to the port name. The `port_complement_naming_style` variable defines the convention for renaming ports complemented as a result of boundary optimization. However, using the `port_complement_naming_style` variable is not recommended because it is not supported in Formality.

Propagating Unconnected Registers and Unconnected Bits of Multibit Registers Across Hierarchies With Boundary Optimization Disabled

You can enable the following area-reduction enhancements when boundary optimization is disabled by setting the `compile_optimize_unloaded_seq_logic_with_no_bound_opt` variable to `true`. By default, the variable is `false`. When the variable is set to `true`, the tool

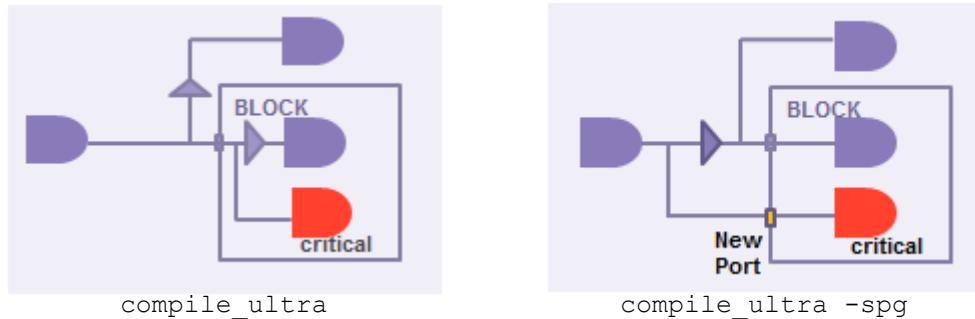
- Removes unconnected registers across hierarchies.
- Enables the propagation of unconnected logic through multibit registers, resulting in the optimization of unused bits and smaller-sized multibit cells.

Port Punching in Design Compiler Graphical

Design Compiler Graphical performs port punching for more flexible load balancing across the hierarchy and improved QoR when you use the `compile_ultra -spg` command.

[Figure 12-3](#) shows buffering across the hierarchy with the `compile_ultra` command (the design on the left) compared to buffering across the hierarchy with the `compile_ultra -spg` command (the design on the right).

Figure 12-3 Buffering Across the Hierarchy Comparison



To disable port punching on the entire design, use the `compile_ultra` command with the `-no_boundary_optimization` option. To disable port punching on specific instances, use the `set_boundary_optimization` command, and set it to `false` for the specific instances.

Port punching is disabled also when you disable port punching with automatic high-fanout synthesis by using the `set_ahfs_options` command with the `-enable_port_punching false` option.

Port Punching and Phase Inversion With Automatic High-Fanout Synthesis

Automatic high-fanout synthesis performs port punching and phase inversion in Design Compiler topographical mode. By default, automatic high-fanout synthesis inserts buffers or inverters across hierarchy boundaries to achieve better QoR.

Port punching in automatic high-fanout synthesis is disabled under the following conditions:

- When you disable boundary optimization by using the `compile_ultra` command with the `-no_boundary_optimization` option, Design Compiler automatically disables port punching and phase inversion for the entire design.
- When you use the `set_boundary_optimization` command and set it to `false` for specific instances, port punching and phase inversion is disabled for those instances.
- When you use the `set_ahfs_options` command with the `-enable_port_punching` option set to `false`, port punching is disabled during automatic high-fanout synthesis.

When you use the `set_ahfs_options` command with the `-preserve_boundary_phase` option set to `true`, inverters cannot move across hierarchical boundaries and the logical phase of the boundary is unchanged during automatic high-fanout synthesis.

Other Optimizations That Affect Hierarchical Boundaries

Design Compiler performs boundary optimization for nets connected to multiple ports and constant-driven ports when the advanced multiple-port net and constant-driven port fixing capability is enabled. For more information, see [Fixing Nets Connected to Multiple Ports](#).

Hierarchical clock gating also performs optimization across hierarchical boundaries. For information, see the *Power Compiler User Guide*.

Automatic Ungrouping

The `compile_ultra` command automatically ungroups certain logical hierarchies. Ungrouping merges subdesigns of a given level of the hierarchy into the parent cell or design. It removes hierarchical boundaries and allows Design Compiler to improve timing by reducing the levels of logic and to improve area by sharing logic.

You can also manually ungroup hierarchies by using the `ungroup` command or the `set_ungroup` command followed by the `compile_ultra` command. For information about manually ungrouping hierarchies, see [Removing Levels of Hierarchy](#).

To learn more about automatic ungrouping, see

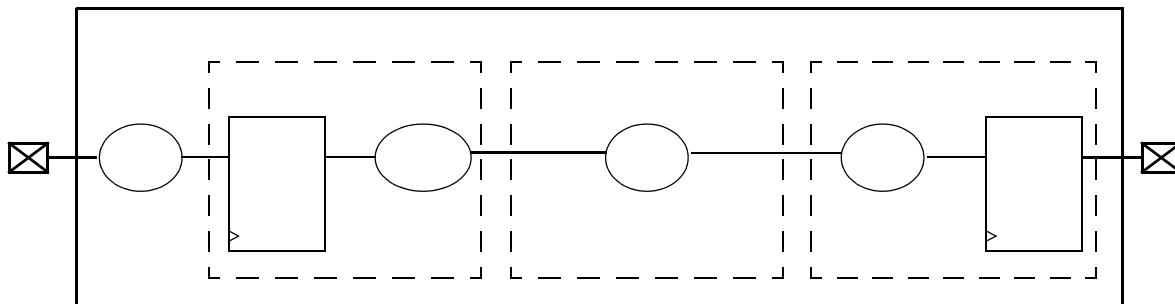
- [Automatic Ungrouping of Hierarchies](#)
- [Automatic Ungrouping of Designs With Timing Exceptions](#)
- [Exceptions to Automatic Ungrouping](#)

Automatic Ungrouping of Hierarchies

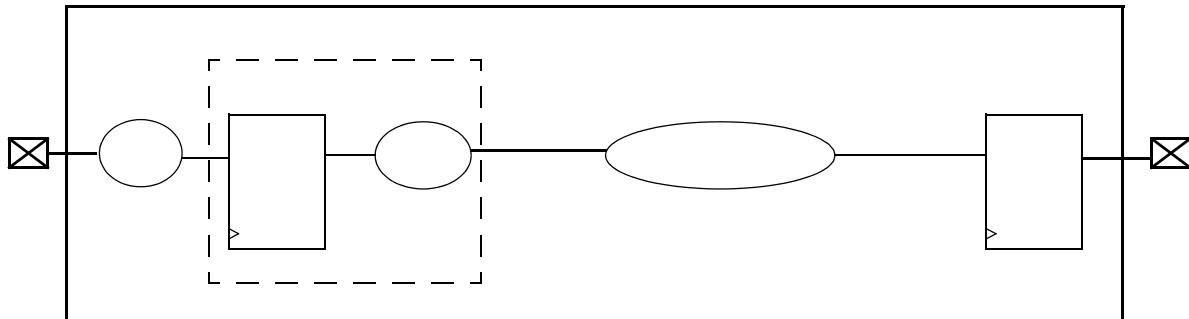
[Figure 12-4](#) shows the hierarchy before and after ungrouping.

Figure 12-4 Automatic Ungrouping of Hierarchies

Before Ungrouping



After Ungrouping



During optimization, Design Compiler performs the following types of automatic grouping:

- Area-based automatic ungrouping

Before initial mapping, the `compile_ultra` command performs area-based automatic ungrouping. The tool estimates the area for unmapped hierarchies and removes small subdesigns; the goal is to improve area and timing quality of results. Because the tool performs automatic ungrouping at an early stage, it has a better optimization context.

Additionally, datapath extraction is enabled across ungrouped hierarchies. These factors improve the timing and area quality of results.

- **Delay-based automatic ungrouping**

During delay optimization, the `compile_ultra` command performs delay-based automatic ungrouping. It ungroups hierarchies along the critical path and is used essentially for timing optimization.

- **QoR-based automatic ungrouping**

When you use the `compile_ultra` command with the `-spg` option, Design Compiler Graphical ungroups additional hierarchies to improve QoR.

To disable automatic ungrouping on a hierarchical design and its subdesigns, use the `-no_auungroup` option with the `compile_ultra` command. To disable automatic ungrouping on specific instances, use the `set_ungroup` command to set the `ungroup` attribute to `false` on specific designs, cells, or references.

See Also

- [Optimization Flow](#)

Shows how automatic ungrouping fits into the overall compile flow

Automatic Ungrouping of Designs With Timing Exceptions

When preserving timing constraints, Design Compiler reassigns the timing constraints to appropriate adjacent, persistent pins (that is, pins on the same net that remain after ungrouping). The constraints are moved forward or backward to other pins on the same net. Note that the constraints can be moved backward only if the pin driving the given hierarchical pin drives no other pin. Otherwise the constraints must be moved forward.

If the constraints are moved to a leaf cell, that cell is assigned a `size_only` attribute to preserve the constraints during a compile. Thus, the number of `size_only` cells can increase, which might limit the scope of the optimization process. To counter this effect, when both the forward and backward directions are possible, Design Compiler chooses the direction that helps limit the number of newly assigned `size_only` attributes to leaf cells.

You can disable this behavior by setting the `auto_ungroup_preserve_constraints` variable to `false`. The default is `true`.

Hierarchies are ungrouped when the following timing constraints are set on hierarchical pins:

- `set_false_path`
- `set_multicycle_path`

- `set_min_delay`
- `set_max_delay`
- `set_input_delay`
- `set_output_delay`
- `set_disable_timing`
- `set_rtl_load`
- `create_clock -period`

Exceptions to Automatic Ungrouping

Hierarchies are not automatically ungrouped in the following cases:

- The hierarchy has user-specified constraints such as `dont_touch`, `size_only`, or `set_ungroup` attributes.
- When boundary optimization is disabled, automatic ungrouping is also disabled.
- In non-topographical mode, the wire load model for the hierarchy is different from the wire load model of the parent hierarchy.

You can override this behavior by setting the `compile_auto_ungroup_override_wlm` variable to `true`. The default is `false`. When the variable is set to `true`, the ungrouped child cells of the hierarchy inherit the wire load model of the parent hierarchy. Consequently, the child cells might have a more pessimistic wire load model.

13

High-Level Optimization and Datapath Optimization

During high-level optimization, Design Compiler performs arithmetic simplifications and resource sharing. A resource is an arithmetic or comparison operator read in as part of an HDL design. A datapath block contains one or more resources that are grouped and optimized by a datapath generator. During the high-level optimization phases, resources are allocated and shared, depending on timing and area considerations. Resource sharing enables the tool to build one hardware component for multiple operations, which typically reduces the hardware required to implement your design.

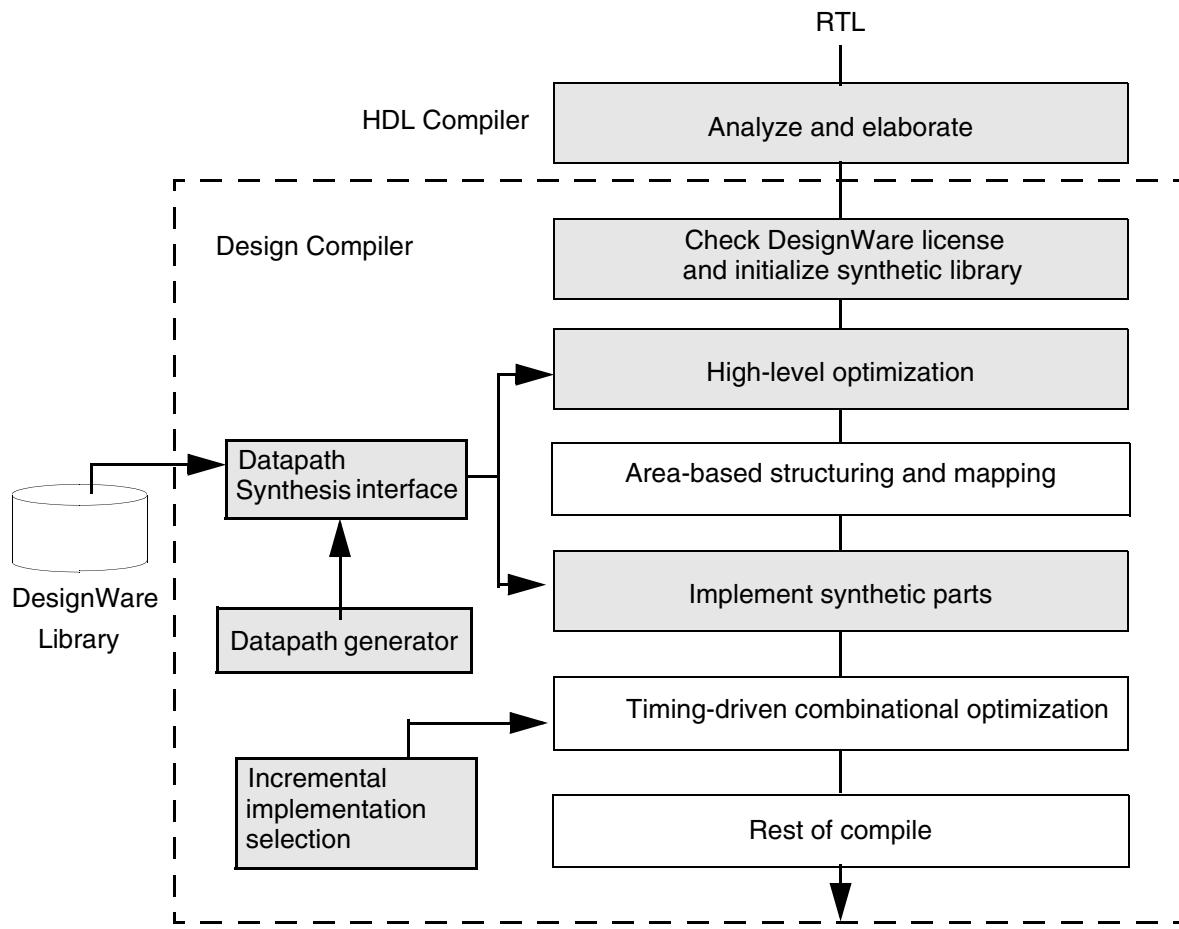
High-level optimizations are performed when you use the `compile` command or the `compile_ultra` command. The `compile_ultra` command explores additional optimization opportunities by performing automatic datapath extraction and advanced datapath transformations. To learn about these topics, see

- [Design Compiler Arithmetic Optimization](#)
- [Synthetic Operators](#)
- [High-Level Optimizations](#)
- [Datapath Optimization](#)
- [Analyzing Datapath Extraction](#)
- [Reporting Resources and Datapath Blocks](#)

Design Compiler Arithmetic Optimization

[Figure 13-1](#) shows how Design Compiler optimizes arithmetic components within the optimization flow. The shaded boxes pertain to the arithmetic optimization flow.

Figure 13-1 Optimization of Arithmetic Expressions



The steps in the Design Compiler arithmetic optimization flow are as follows:

1. When HDL Compiler elaborates a design, it maps HDL operators (either built-in operators like + and * or HDL functions and procedures) to synthetic (DesignWare) operators that appear in the generic netlist. For more information, see [Synthetic Operators](#).
2. Design Compiler checks for any required licenses and initializes the synthetic library. For more information, see [Checking DesignWare Licenses](#).

3. During high-level optimization, Design Compiler manipulates the synthetic operators and applies optimizations such as arithmetic simplifications and resource sharing. See [High-Level Optimizations](#). If you are using DC Ultra, Design Compiler performs automatic datapath extraction. For more information, see [Datapath Optimization](#).
 4. During the implement synthetic parts phase, the tool maps synthetic modules to architectural representations (implementations). For more information on synthetic parts, see the DesignWare documentation.
- Design Compiler uses the datapath generator to implement arithmetic components and generate the best implementations. In addition, if you are using DC Ultra, Design Compiler performs advanced datapath transformations on the extracted datapath blocks. For more information, see [Datapath Optimization](#).
5. During incremental implementation selection, Design Compiler explores alternative implementations for each arithmetic component. The tool evaluates and replaces synthetic implementations along the critical path to improve delay cost.

See Also

- [Optimization Flow](#)

Shows how high-level optimizations and datapath optimizations fit into the overall compile flow

Synthetic Operators

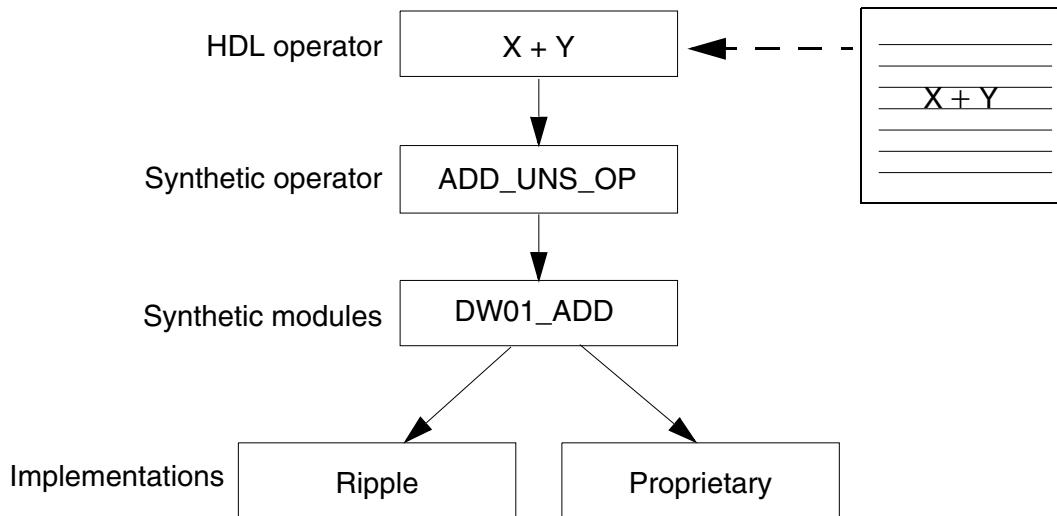
Synopsys provides a collection of intellectual property (IP), referred to as the DesignWare Building Block IP Library, to support the synthesis products. Building Block IP provides basic implementations of common arithmetic functions that can be referenced by HDL operators in your RTL source code.

The DesignWare library is built on a hierarchy of abstractions. HDL operators, either built-in operators, such as addition (+) and multiplication (*), or HDL functions and procedures, are associated with synthetic operators, which are bound in turn to synthetic modules. Each synthetic module can have multiple architectural realizations, called implementations. For example, when you use the HDL addition operator in a design description, HDL Compiler infers the need for an adder resource and puts an abstract representation of the addition operation into your circuit netlist. See [Figure 13-2](#).

During high-level optimization, Design Compiler manipulates these synthetic operators and applies optimizations such as arithmetic transformations and resource sharing.

To display information about the standard synthetic library that is included with a Design Compiler license, use the `report_synlib` command:

```
prompt> report_synlib standard.sldb
```

Figure 13-2 DesignWare Overview**See Also**

- The DesignWare documentation
Provides information about DesignWare synthetic operators, modules, and libraries

High-Level Optimizations

During high-level optimization, Design Compiler applies techniques such as tree delay minimization and arithmetic simplifications; it also performs resource sharing.

Tree Delay Minimization and Arithmetic Simplifications

During tree delay minimization, Design Compiler arranges the inputs to arithmetic trees. For example, the expression $a + b + c + d$ describes three levels of cascaded addition operations. The tool can rearrange this expression to $(a + b) + (c + d)$, which might result in faster logic (only two levels of cascaded operations).

Additional simplification is available with the `compile_ultra` command; some examples are as follows:

- Sink cancellation
The expression $(a + b - a)$ is simplified to b .
- Constant folding
The expression $(a * 3 * 5)$ is transformed to $(a * 15)$

Resource Sharing

Resource sharing reduces the amount of hardware needed to implement operators such as addition (+) in your Verilog or VHDL description. Without this feature, each operation is built with separate hardware. For example, every + operator builds an adder. This repetition of hardware increases the area of a design.

There are two basic types of resource sharing: Common subexpression elimination and sharing mutually exclusive operations.

To learn about resource sharing, see

- [Common Subexpression Elimination](#)
- [Sharing Mutually Exclusive Operations](#)
- [Enhanced Resource Sharing](#)

Common Subexpression Elimination

This type shares redundant computations in a design. To understand common subexpression elimination, consider [Example 13-1](#).

Example 13-1 Original RTL

```
X = A > B;  
Y = A > B && C;
```

This code contains two comparators and a logical add. In common subexpression elimination, the common subexpression, A > B, is grouped and the code is transformed to [Example 13-2](#).

Example 13-2 Expression A > B Shared

```
Temp = A > B;  
X = Temp;  
Y = Temp && C;
```

This transformation reduces the number of comparators from two to one.

Both HDL Compiler and Design Compiler perform common subexpression elimination. However, HDL Compiler does not share the +, *, and – operators by default because it might reduce the sharing options available to Design Compiler during compile. Design Compiler shares these operators and all other operators by default during timing-driven optimization.

The following operators are shared by default when Design Compiler does common subexpression elimination:

- Relational (=, <, >, <=, >=, !=)
- Shifting (<<, >>, <<<, >>>)

- Arithmetic (+, -, *, /, **, %)

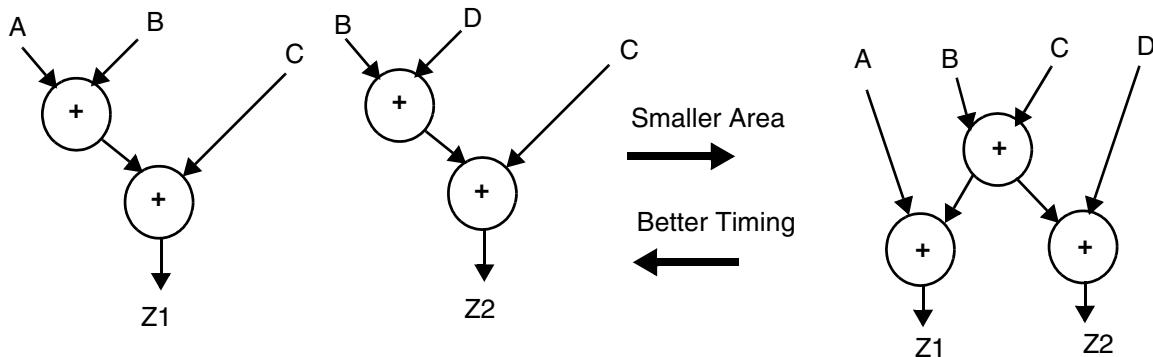
These are selectors that drive arithmetic operators.

Additionally, if you use the `compile_ultra` command, the tool can identify common subexpressions automatically; you do not need to use parentheses or write them in the same order. For example, the expressions

$(A + B + C)$ and $(B + A + D)$, $A + B$ and $B + A$ are recognized as a common subexpression.

Furthermore, the tool can either share common subexpressions or reverse the sharing depending on constraints. Consider the following expressions: $Z1 \leq A + B + C$, $Z2 \leq B + C + D$, and arrival time is $A < B < D < C$; [Figure 13-3](#) shows how the tool might reverse the sharing of common subexpressions depending on constraints. The tool determines whether to share or reverse the sharing of operators during a later phase—that is, during timing-driven optimization.

Figure 13-3 Sharing and Unsharing of Arithmetic Subexpressions



Sharing Mutually Exclusive Operations

This type of resource sharing shares operators in a single process when there is no execution path that reaches both operators from the start of the block to the end of the block—that is, operations that cannot be performed simultaneously are shared. To understand this type of resource sharing, consider [Example 13-3](#).

Example 13-3 Sharing Two + Operators

```
module resources(A,B,C,SEL);
  input A,B,C,D;
  input SEL;
  output [1:0] Z;

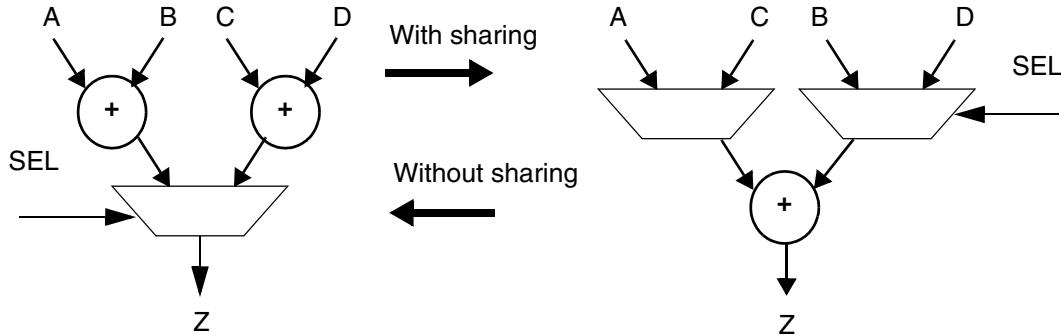
  reg [1:0] Z;

  always @ (A or B or C or D or SEL)
begin
  if(SEL)
    Z = B + A;
  else
    Z = C + D;
end
endmodule
```

[Example 13-3](#) shows code that adds either $A + B$ or $D + C$; what is added depends on whether the condition SEL is true.

Without resource sharing, the tool builds two adders and one MUX, and with resource sharing, the tool uses only one adder to build the design, as shown in [Figure 13-4](#).

Figure 13-4 Design With and Without Sharing



Similar to common subexpression sharing, the tool determines whether to share or reverse sharing depending on timing constraints. For example, when the arrival of the SEL signal is late and sharing the adder worsens timing quality of results (QoR), Design Compiler does not share the adder.

Enhanced Resource Sharing

You can perform enhanced resource sharing by using the `compile_enhanced_resource_sharing` variable. It is useful to enable the variable for designs that contain numerous arithmetic operators. If you set the variable to `true`, Design Compiler identifies more opportunities for sharing in the design, such as common

subexpression sharing of nonidentical comparators and sharing of mutually exclusive operators that do not drive the same selector directly. By default, the `compile_enhanced_resource_sharing` variable is set to `false`.

Datapath Optimization

Datapath design is commonly used in applications that contain extensive data manipulation, such as 3-D, multimedia, and digital signal processing (DSP). DC Ultra enables datapath optimization by default when you use the `compile_ultra` command. Datapath optimization is comprised of two steps: *Datapath extraction*, which transforms arithmetic operators (for example, addition, subtraction, and multiplication) into datapath blocks, and *datapath implementation*, which uses a datapath generator to generate the best implementations for these extracted components.

Note:

Datapath optimization requires a DC Ultra license and a DesignWare license. The DesignWare license is pulled when you run the `compile_ultra` command.

During datapath optimization, the tool does the following:

- Shares (or reverses the sharing) of datapath operators
- Uses the carry-save arithmetic technique
- Extracts the datapath
- Performs high-level arithmetic optimization on the extracted datapath
- Explores better solutions that might involve a different resource-sharing configuration
- Makes better tradeoffs between resource sharing and datapath optimization

For details about the datapath optimization steps, see

- [Datapath Extraction](#)
- [Datapath Implementation](#)
- [Advanced Datapath Transformations](#)

Datapath Extraction

Datapath extraction transforms arithmetic operators (for example, addition, subtraction, and multiplication) into datapath blocks to be implemented by a datapath generator. This transformation improves the quality of results (QoR) by utilizing the carry save arithmetic technique.

Carry save arithmetic does not fully propagate carries but instead stores results in an intermediate form. The carry-save adders are faster than the conventional carry-propagate adders because the carry-save adder delay is independent of bit-width. These adders use significantly less area than carry-propagate adders because they do not use full adders for the carry.

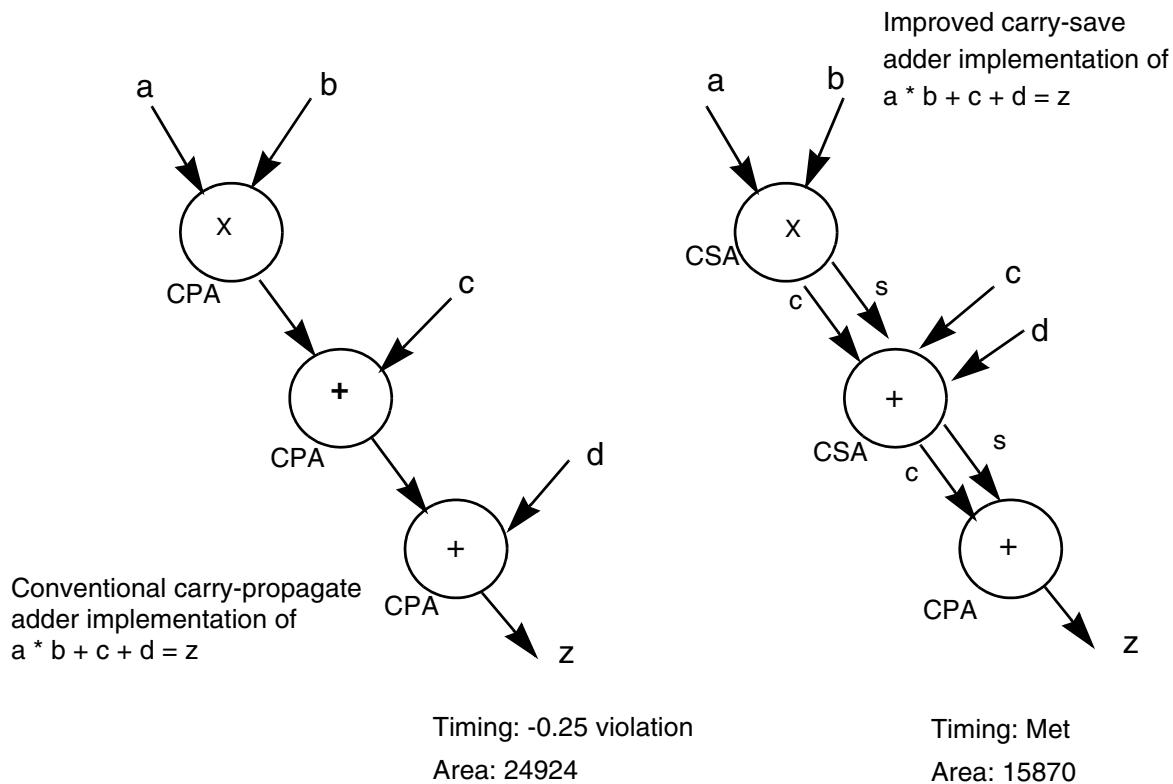
[Example 13-4](#) shows the code for the expression $a * b + c + d = z$. [Figure 13-5](#) shows that the conventional implementation of the expression $a * b + c + d = z$ would use three carry-propagate adders (CPAs); whereas, the carry save technique requires only one carry-propagate adder and two carry-save adders (CSAs). [Figure 13-5](#) also shows the timing and area numbers for both implementations.

Example 13-4 Code Example

```
module dp (a,b,c,d,e);
  input [15:0] a,b;
  input [31:0] c,d;
  output [31:0] e;

  assign e = ( a * b )+ c + d;
endmodule
```

Figure 13-5 Conventional Carry-Propagate Adder and Faster, Smaller Carry-Save Adder



When DC Ultra datapath optimization is used to compile design dp in [Example 13-4](#), the following improvements are realized:

- 8.3 percent timing improvement, compared to DC Expert results
- 36 percent area improvement, compared to DC Expert results

The DC Ultra datapath solution supports extraction of the following components:

- Arithmetic operators that can be merged into one CSA tree
- Operators extracted as part of a datapath: *, +, -, >, <, <=, >=, ==, !=, and MUXes
- Variable shift operators (<<, >>, <<<, >>> for Verilog and sll, srl, sla, sra, rol, ror for VHDL)
- Operations with bit truncation

The datapath flow can extract these components only if they are directly connected to each other—that is, no nonarithmetic logic between components. Keep the following points in mind:

- Extraction of mixed signed and unsigned operators is allowed only for adder trees
- Instantiated DesignWare components cannot be extracted

Datapath Implementation

All the synthetic operators and the extracted datapath elements are implemented by the DesignWare datapath generator in DC Ultra.

The DesignWare datapath generator uses the smart generation technology to perform the following tasks:

- Implement datapath blocks using context-driven optimizations
- Revisit high-level optimization decisions for a given timing context
- Consider logic library characteristics

For further fine tuning, the control of the smart generation strategies is available by using the Design Compiler `set_dp_smartgen_options` command.

See Also

- The DesignWare documentation

Provides more information about the datapath smart generation strategies

Advanced Datapath Transformations

DC Ultra performs advanced datapath transformations on the extracted datapath blocks, such as the following:

- Sum-of-products to Product-of-Sums
The expression $(A^*C + B^*C)$ is transformed to $(A + B) * C$.
- Comparator sharing
Expressions such as $A > B$, $A < B$, $A \leq B$ are transformed to use single subtractors with multiple comparison outputs.
- Optimization of parallel constant multipliers
- Operand reordering
The tool can rearrange operands of multipliers or comparators to produce different quality of results (QoR).
- Explore trade-offs between common subexpression elimination (CSE) sharing and mutually exclusive operations (MUTEX) sharing. Design Compiler can undo the CSE sharing in the GTECH netlist generated by HDL Compiler to facilitate the optimum combination of CSE and MUTEX sharing.

Analyzing Datapath Extraction

To get the best QoR results from DC Ultra datapath optimization, it is important that the biggest possible datapath blocks are extracted from the RTL code. To examine how the datapath blocks will be extracted from your RTL before you run the `compile_ultra` command, use the `analyze_datapath_extraction` command. The command analyzes the arithmetic contents of the design and provides feedback so you can improve the RTL code as needed. For information about coding styles to help improve datapath extraction, see [SolvNet article 015771, “Coding Guidelines for Datapath Synthesis.”](#)

When the `analyze_datapath_extraction` command detects an arithmetic operator that is not extracted, it reports the information about what is blocking the datapath extraction, including the file name of the RTL and the line number of the operator it is inferred from. The following example shows an analysis report about fanout truncation that breaks the datapath block:

```
prompt> analyze_datapath_extraction
Information: Checking out the license 'DesignWare'. (SEC-104)
Analyzing datapath extraction ...
Information: Operator associated with resources 'add_440 (test.v:446)' in
design 'test' breaks the datapath block because there is leakage due to
truncation on its fanout. (HDL-120)
```

To generate an HTML report instead of a text-based report, specify the `-html_file_name` option with the `analyze_datapath_extraction` command. The HTML report provides direct links to the RTL file. [Figure 13-6](#) shows an excerpt from an HTML report. You can open the `test.v` file shown in the figure by clicking the `test.v` link. The number after the `test.v` file name refers to the line number in the file.

Figure 13-6 Datapath Report in HTML Format

Datapath Report for DP_OP_11J1_124_796	
Cell	Contained Operations
DP_OP_11J1_124_796	mult_5 (test.v:5); mult_5_2 (test.v:5); add_6 (test.v:6);

In the GUI, you can cross-probe from the datapath extraction report to the RTL file in the RTL browser to inspect the RTL and determine if any changes are needed. For more information about cross-probing, see the Design Vision Help.

To disable the automatic ungrouping of hierarchies during analysis, use the `-no_auoutgroup` option. This ensures that all hierarchies are preserved unless otherwise specified. The `-no_auoutgroup` option behavior is the same in the `analyze_datapath_extraction` command as it is in the `compile_ultra` command. If you plan to specify the `-no_auoutgroup` option with the `compile_ultra` command, you should also specify it with the `analyze_datapath_extraction` command.

To filter messages in the datapath extraction report, use the `-filter_msg` option. Messages with severity levels less than or equal to the specified level are filtered out. For example, the following command filters out the `analyze_datapath_extraction` command warnings that have a priority of medium or lower, and only warnings that have a high priority are displayed:

```
prompt> analyze_datapath_extraction -filter_msg medium
```

To sort the warning messages for each subdesign from high-to-low priority, use the `-sort_msg` option. Warnings that point to datapath leakage and unsigned subtraction, such as the HDL-120 and HDL-132 warnings, have a higher priority. This option also provides a high-level summary of the high-priority warnings issued for each subdesign.

As the following example shows, the tool provides a high-level summary of leakage warnings followed by the actual warnings for each subdesign:

```
prompt> analyze_datapath_extraction -sort_msg
*****
Leakage detections on design DW_mult_add
*****
Msg type      |    Msg count    | Max lkg width | Avg. lkg width
-----|-----|-----|-----
HDL-120      3          35          19
-----
***Sorted leakage messages...
```

For more information about datapath leakage, including examples showing how leakage occurs, see [SolvNet article 015771, “Coding Guidelines for Datapath Synthesis,”](#) the `analyze_datapath_extraction` man page, and the HDL-120 and HDL-132 message man pages.

To specify the maximum number of leakage detection messages (HDL-120 and HDL-132) to display for each design, use the `-max_msgs` option. The `-sort_msg` option is enabled automatically when you use the `-max_msgs` option. This ensures that only the top leakage messages are reported.

For large designs, analyzing one or more subdesigns can be useful. You can perform hierarchical analysis on specified designs by using the `get_designs` command with the `analyze_datapath_extraction` command:

```
prompt> analyze_datapath_extraction [get_designs design_list]
```

The `analyze_datapath_extraction` command generates separate reports for each specified design.

After the analysis, a report shows the contained resources in the design. This report helps to find the resources in the design source code. Note that the datapath names in this report might differ from the datapath names in the report generated using the `report_resources` command after compilation. However, the names of the contained resources will be the same.

For information about interpreting the resource report, see [Reporting Resources and Datapath Blocks](#).

Reporting Resources and Datapath Blocks

To generate a report that lists the resources and datapath blocks used in a design during compilation, run the `report_resources` command. By default, the command returns a report for the current design; however, you can specify a particular design or a list of designs by using the `design_list` argument.

To understand the `report_resources` report, consider the code in [Example 13-5](#).

Example 13-5 RTL for Datapath Module

```
module datapath (a, b, c, d, sel, z1, z2);
  input [7:0] a, b, c, d;
  input sel;
  output [15:0] z1, z2;
  wire [15:0] prod = sel? a * b : a * c;
  assign z1 = prod + d;
  assign z2 = c * d;
endmodule
```

When this code is compiled, the `report_resources` command generates the report shown in [Example 13-6](#).

Example 13-6 Datapath Report for Design Datapath Generated by the `report_resources` Command

```
*****
Report : resources
Design : test
...
*****
Resource Report for this hierarchy in file /usr/.../test.v
=====
| Cell           | Module          | Parameters | Contained Operations |
=====
| mult_x_1       | DW_mult_uns     | a_width=8   | mult_7 (test.v:7)    |
|                 |                 | b_width=8   |                         |
| DP_OP_11J1_124_796 |               |             |                         |
|                 | DP_OP_11J1_124_796 |             |                         |
=====

Datapath Report for DP_OP_11J1_124_796
=====
| Cell           | Contained Operations |
=====
| DP_OP_11J1_124_796 | mult_5 (test.v:5) mult_5_2 (test.v:5) |
|                   | add_6 (test.v:6)   |
=====

=====
|      | Data      |           |           |
| Var | Type     | Class     | Width    | Expression   |
=====
| I1  | PI       | Unsigned  | 8        |             |
| I2  | PI       | Unsigned  | 8        |             |
| I3  | PI       | Unsigned  | 8        |             |
| T5  | IFO      | Unsigned  | 16       | I1 * I2 (test.v:5) |
| O1  | PO       | Unsigned  | 16       | T5 + I3 (test.v:6) |
=====

Implementation Report
=====
|           | Current          | Set          |
| Cell     | Module          | Implementation | Implementation |
=====
| mult_x_1 | DW_mult_uns     | apparch (area) |             |
| DP_OP_11J1_124_796 | DP_OP_11J1_124_796 | str (area)   |             |
=====

No multiplexors to report
1
```

The `report_resources` report consists of the following reports:

- **Resource report**

The Resource report shows arithmetic operators that are mapped to individual DesignWare components. The report example shows that a `mult_x_7_1` multiplier cell is mapped to a `DW_mult_uns` unsigned DesignWare multiplier.

- **Datapath report**

The Datapath report shows arithmetic operators that are merged into a single datapath block by datapath extraction. The report example shows that the operators are merged into a single datapath cell, `DP_OP_11J1_124_796`. The contained operations show the list of operations that are contained in each cell. Note that the `_5` suffix in the operation names represents the line number in the RTL code in [Example 13-5](#). If more than one operator appears in a line, it is reflected in the suffix. Two operators appear in line 5 in [Example 13-5](#); therefore, the second multiplier is identified in the report with a `_5_2` suffix.

The Datapath report table shows the input (PI) and output (PO) ports of the datapath cell. It also shows the datapath expression with the intermediate fanout (IFO) and its data class. Note that the expression in this report is just a functional representation of each datapath block. The data class shows whether the signal should be zero extended (unsigned) or sign extended (signed) in computation when the signal is used for subsequent operations. The report does not describe the actual internal implementation of the block.

- **Implementation report**

The Implementation report shows the implementation of each DesignWare block. The report includes the implementation name and the optimization mode that is used to implement each DesignWare cell. The implementation report is generated for both detected DesignWare cells and ungrouped DesignWare cells.

- **Datapath Extraction report**

The Datapath Extraction report shows arithmetic operators that block datapath extraction. When the `report_resources` command detects operators that block datapath extraction, the tool reports the following messages in the Datapath Extraction report, depending on the issue:

- HDL-120: Datapath leakage blocks the datapath extraction
- HDL-125: Instantiated DesignWare component cannot be extracted
- HDL-132: Mixed-signal data type blocks the datapath abstraction

For example,

```
Datapath Extraction Report
Information: The output of subtractor associated with resources
'sub_388 (test.v:388)' is treated as signed signal. (HDL-132)
```

To generate an HTML report instead of a text-based report, specify the `-html_file_name` option with the `report_resources` command. The HTML report provides direct links to the RTL file. For example, you can open the test.v file shown in [Figure 13-7](#) by clicking the test.v link. The number after the test.v file name refers to the line number in the file.

In the GUI, you can cross-probe from the design resources report to the RTL file in the RTL browser to inspect the RTL and determine if any changes are needed. Similarly, you can cross-probe from the RTL file to the design resources report. For more information about cross-probing, see the Design Vision Help.

In [Figure 13-7](#), the Resource report shows that the `mult_x_1` cell is inferred from line 7 in the test.v file. The Datapath report shows that the `DP_OP_11J1_124_796` datapath cell has operators inferred from lines 5 and 6 in the test.v file.

Figure 13-7 Resource and Datapath Report in HTML Format

Resource Report for this hierarchy in file			
Cell	Module	Parameters	Contained Operations
<code>mult_x_1</code>	<code>DW_mult_uns</code>	<code>a_width=8; b_width=8; mult_7 (test.v:7);</code>	
<code>DP_OP_11J1_124_796</code>	<code>DP_OP_11J1_124_796</code>		

Datapath Report for DP_OP_11J1_124_796	
Cell	Contained Operations
<code>DP_OP_11J1_124_796</code>	<code>mult_5 (test.v:5); mult_5_2 (test.v:5); add_6 (test.v:6);</code>

Var	Type	Data Class	Width	Expression
I1	PI	Unsigned	8	
I2	PI	Unsigned	8	
I3	PI	Unsigned	8	
T5	IFO	Unsigned	16	<code>I1 * I2 (test.v:5)</code>
O1	PO	Unsigned	16	<code>T5 + I3 (test.v:6)</code>

Implementation Report				
Cell	Module	Current Implementation	Set Implementation	
<code>mult_x_1</code>	<code>DW_mult_uns</code>	<code>apparch (area)</code>		
<code>DP_OP_11J1_124_796</code>	<code>DP_OP_11J1_124_796</code>	<code>str (area)</code>		

14

Multiplexer Mapping and Optimization

Design Compiler can map combinational logic representing multiplexers in the HDL code directly to a single multiplexer (MUX) or a tree of multiplexer cells from the target logic library. Before you read further, see [Optimization Flow](#) to understand how multiplexer mapping and optimization fit into the overall compile flow.

Multiplexers are commonly modeled with if and case statements. To implement this logic, HDL Compiler uses SELECT_OP cells, which Design Compiler maps to combinational logic or multiplexers in the logic library. If you want Design Compiler to preferentially map multiplexing logic to multiplexers—or multiplexer trees—in your logic library, you must infer MUX_OP cells.

The MUX_OP cell should be inferred when you want Design Compiler to build a multiplexer tree structure for the case statement blocks in your HDL. MUXs can be implemented efficiently (in speed and area) in the library. This type of structure can provide advantages in circuit performance and savings in wiring area, compared to implementation constructed from random logic. This feature is supported only with the use of the case statement in VHDL or Verilog code.

During compilation, Design Compiler replaces each MUX_OP cell with the best multiplexer tree implementation of the N-input M-output multiplexer. The implementation depends on the constraints given for the design and the arrival times of the selector inputs.

To learn about multiplexer mapping and optimization, see

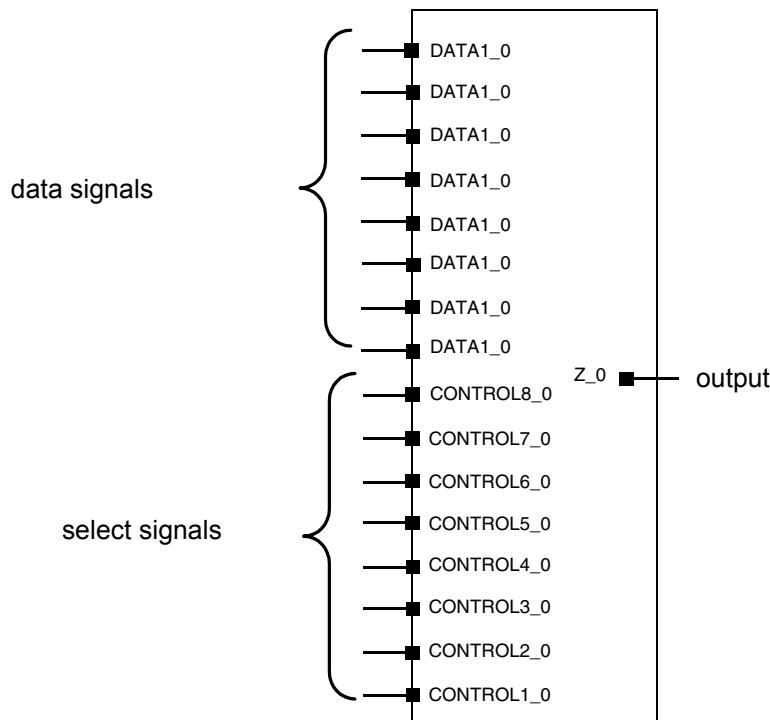
- [Inferring SELECT_OPs](#)
- [Inferring MUX_OPs](#)

- Library Cell Requirements for Multiplexer Optimization
- Mapping Multiplexers on Asynchronous Signal Lines
- Mapping to One-Hot Multiplexers

Inferring SELECT_OPs

By default, HDL Compiler uses SELECT_OP components to implement conditional operations implied by if and case statements. An example of a SELECT_OP cell implementation for an 8-bit data signal is shown in [Figure 14-1](#).

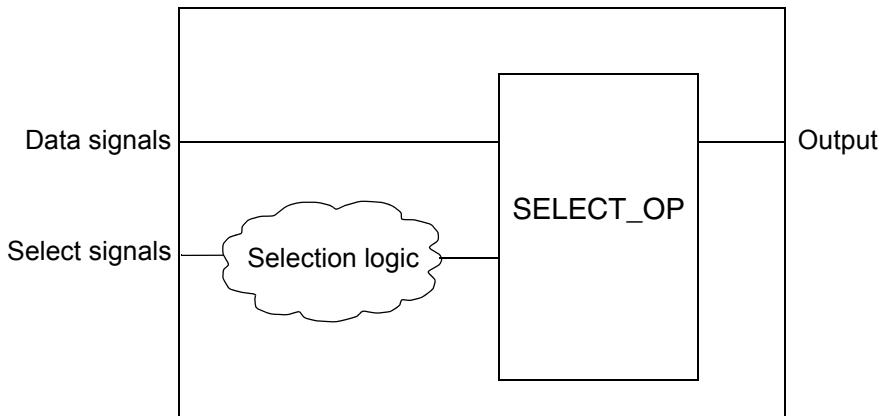
Figure 14-1 SELECT_OP Implementation for an 8-bit Data Signal



For an 8-bit data signal, 8 selection bits are needed.
This is called a one-hot implementation.

SELECT_OPs behave like one-hot multiplexers; the control lines are mutually exclusive, and each control input allows the data on the corresponding data input to pass to the output of the cell. To determine which data signal is chosen, HDL Compiler generates selection logic, as shown in [Figure 14-2](#).

Figure 14-2 Verilog Output—SELECT_OP and Selection Logic



Depending on the design constraints, Design Compiler implements the SELECT_OP with either combinational logic or multiplexer cells from the logic library. For more information on SELECT_OP inference, see the HDL Compiler documentation.

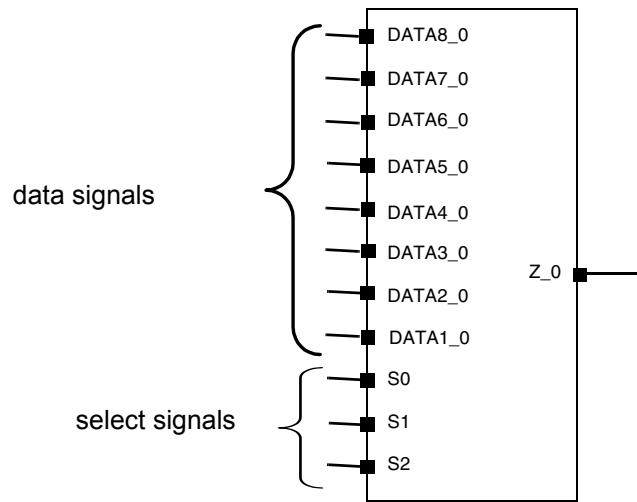
Inferring MUX_OPs

If you want Design Compiler to preferentially map multiplexing logic in your RTL to multiplexers—or multiplexer trees—in your logic library, you need to infer MUX_OP cells. These cells are hierarchical generic cells optimized to use the minimum number of select signals. They are typically faster than the SELECT_OP cell, which uses a one-hot implementation. Although MUX_OP cells improve design speed, they also might increase area. During optimization, Design Compiler preferentially maps MUX_OP cells to multiplexers—or multiplexer trees—from the logic library, unless the area costs are prohibitive, in which case combinational logic is used.

You can embed an attribute or directive in the HDL code or use variables to tell HDL Compiler which part of the HDL description to implement as a single multiplexer or a tree of multiplexers. When the HDL is read in, a generic cell called MUX_OP cell represents the multiplexer functionality. During optimization, Design Compiler maps the logic inside the MUX_OP cell to an implementation, using multiplexer cells from the library.

The MUX_OP cell is a generic representation of an N:1 multiplexer with M output bits. When VHDL or Verilog code is read in, the resulting design contains a MUX_OP cell for every case block inside a process that contains the `infer_mux` directive. A MUX_OP cell also is inferred for each signal (including bused signals) assigned inside the same case block. The signals in the HDL that compute the selector are connected to the select inputs of the MUX_OP cell. [Figure 14-3](#) shows a generic MUX_OP cell for an 8-bit data signal.

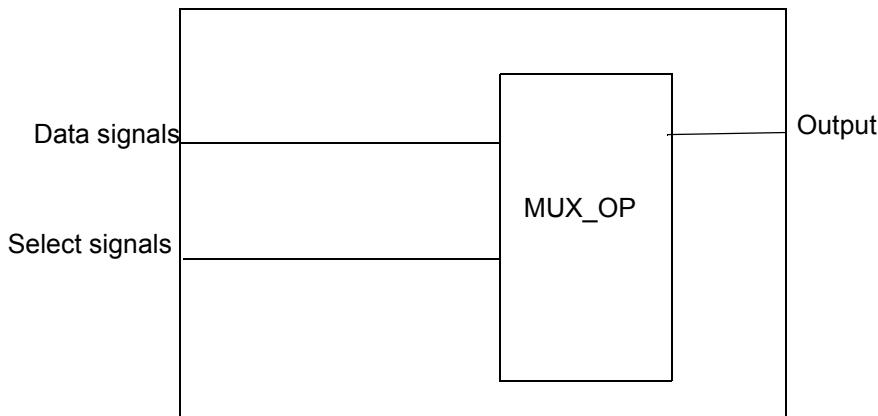
Figure 14-3 MUX_OP Generic Cell for an 8-bit Data Signal



For an 8-bit word, only 3 selection bits are needed.

The MUX_OP cell contains internal selection logic to determine which data signal is chosen; HDL Compiler does not need to generate any selection logic, as shown in [Figure 14-4](#).

Figure 14-4 HDL Compiler Output—MUX_OP Generic Cell for 8-Bit Data



The naming convention for MUX_OP cells in a design is

`_MUX_OP_N_S_M`

Argument	Description
N	The number of data inputs.
S	The number of select inputs.
M	The width of the output signal.

Because the MUX_OP cell is implemented as a level of hierarchy, certain types of logic sharing that might otherwise take place are no longer possible. In some cases, this can lead to a design that is less suitable than one having no MUX_OP cells.

The MUX_OP feature enables you to direct Design Compiler to generate a MUX tree structure for a given case statement when you already know that a MUX tree is the best representation for that statement. Design Compiler optimizes the tree based on constraints and yields the best-possible MUX tree.

To generate MUX_OP cells for a specific case or if statement, use the `infer_mux` directive in the HDL description as shown in [Example 14-1](#). For more information on this directive and inference limitations, see the HDL Compiler documentation.

Example 14-1 Using the infer_mux directive in the HDL Description

```
always@(SEL) begin
  case (SEL) // synopsys infer_mux
    2'b00: DOUT <= DIN[0];
    2'b01: DOUT <= DIN[1];
    2'b10: DOUT <= DIN[2];
    2'b11: DOUT <= DIN[3];
  endcase
```

Observe the following when you use MUX_OP cells:

- Do not set the HDL Compiler variable `hdlin_infer_mux` to all when you compile. Setting this variable to all results in implementing MUX_OP cells for every case statement in your HDL. MUX_OP cells should be inferred for case statements that can benefit most from a multiplexer tree structure.
- Incompletely specified case statements can result in MUX_OP cells with unused inputs or a partially collapsed multiplexer tree structure. In such cases, the difference between using a MUX_OP cell and implementing the multiplexer with random logic might not be significant.
- In general, most gains are realized when you use a MUX_OP cell for fully specified case statements that implement large multiplexer logic. If your design uses mostly small (2:1 or 4:1) multiplexers, you might get better results by not using MUX_OP cells.

- To ensure that MUX_OP cells are mapped to MUX technology cells, you must apply a `size_only` attribute to the cells to prevent logic decomposition during the optimization steps. You can set the `size_only` attribute on each MUX_OP manually or allow HDL Compiler to set it automatically when reading the RTL. You can control the automatic behavior by using the `hdlin_mux_size_only` variable. For more information about the `hdlin_mux_size_only` variable, see the HDL Compiler documentation.

See Also

- [Mapping to One-Hot Multiplexers](#)

Provides information about recognizing and mapping to one-hot multiplexers

Library Cell Requirements for Multiplexer Optimization

The multiplexer optimization requires the presence of at least a 2:1 multiplexer cell in the logic library. The inputs or outputs of this cell can be inverted. If a 2:1 multiplexer primitive cell does not exist in the library, you see the following warning message:

Warning: Target library does not contain any 2-1 multiplexer.
(OPT-853)

An implementation of the MUX_OP cell from the target library is created, but it might not be the best implementation possible. All multiplexer cells in the target library can be used to construct the implementation of the MUX_OP cell except

- Enabled multiplexer cells
- Bused output multiplexer
- Multiplexers larger than 32 : 1

For Design Compiler to make the best use of the multiplexer cells available in your logic library, recompile the library or obtain a library compiled with version V3.4a or later from your ASIC vendor.

Mapping Multiplexers on Asynchronous Signal Lines

Design Compiler can map MUX_OP cells and two-input SELECT_OP cells on asynchronous set and reset signals and clock signals to MUX cells, reducing glitches on asynchronous register pins. By default, the `compile` or `compile_ultra` commands preserve multiplexers in the fanin cone of asynchronous register pins. Specifically, if Design Compiler finds multiplexing logic (any MUX_OP or two-input SELECT_OP cell) in the fanin of only asynchronous register pins (clock or asynchronous set and reset pins), the

multiplexing logic is mapped to a multiplexer and a `size_only` attribute is applied to the multiplexer. To turn this capability off, set the `compile_enable_async_mux_mapping` variable to `false`.

You do not need to set the `size_only` attribute, either manually or with the `hdlin_mux_size_only` variable. Design Compiler sets the `size_only` attribute automatically.

If the MUX_OP or SELECT_OP cell is specified as `dont_touch`, Design Compiler does not map the cell to a multiplexer.

Mapping to One-Hot Multiplexers

Design Compiler also supports inference and mapping of one-hot multiplexers as described in the following topics:

- [Inferring One-Hot Multiplexers](#)
- [Library Requirements for One-Hot Multiplexers](#)
- [Optimization of One-Hot Multiplexers](#)

Inferring One-Hot Multiplexers

A one-hot multiplexer is a library cell that behaves functionally as an AND/OR gate such as an AO22 or AO222. The difference is that in case of a one-hot MUX, there are as many control inputs as data inputs and the function of the cell ANDs each control input with the corresponding data input. For example, a 4-to-1 one-hot MUX has the following function:

$$Z = (D_0 \& C_0) \mid (D_1 \& C_1) \mid (D_2 \& C_2) \mid (D_3 \& C_3)$$

One-hot MUXes are generally implemented using passgates, which makes them very fast and allows their speed to be largely independent of the number of data bits being multiplexed. However, this implementation requires that exactly one control input be active at a time. If no control inputs are active, the output remains floating. If more than one control input is active, there could be an internal drive fight.

Design Compiler allows you to control one-hot MUX inference and mapping. Because of the restriction on the control inputs of a one-hot MUX (that is, one control input is active at all times), Design Compiler cannot automatically make use of these gates. The tool cannot verify that the control inputs behave as required. Therefore, it does not automatically map to these cells. Instead, it maps only to cells that you specify can be mapped. Also, after the tool maps these cells, they cannot be unmapped. The cells can only be sized.

[Example 14-2](#) and [Example 14-3](#) show the coding styles that are supported.

Example 14-2 Supported Coding Style Example

```
case (1'b1) //synopsis full_case parallel_case infer_onehot_mux
sel1 : out = in1;
sel2 : out = in2;
sel3 : out = in3;
```

Example 14-3 Supported Coding Style Example

```
case({sel3, sel2, sel1}) //synopsis full_case parallel_case
infer_onehot_mux
001: out = in1;
010: out = in2;
100: out = in3;
```

Note:

The `parallel_case` and `full_case` directives are required. The `infer_onehot_mux` directive is supported only in Verilog and SystemVerilog.

See Also

- The HDL Compiler documentation

Provides information about coding styles and directives

Library Requirements for One-Hot Multiplexers

Design Compiler can recognize and map to a one-hot MUX cell in the target library only if the one-hot MUX cell meets with all of the following requirements:

- It is a single-output cell.
- Its inputs can be divided into two disjoint sets of the same size as follows:
 $C=\{C_1, C_2, \dots, C_n\}$ and $D=\{D_1, D_2, \dots, D_n\}$
where n is greater than 1 and is the size of the set. Actual names of the inputs can be different from the connotation shown.
- The `contention_condition` attribute must be set on the cell. The value of the attribute is a combinational function, FC, of inputs in set C that defines prohibited combinations of inputs as shown in the following examples (where the size n of the set is 3):

$FC = C_0' \& C_1' \& C_2' \mid C_0 \& C_1 \mid C_0 \& C_2 \mid C_1 \& C_2$

or

$FC = (C_0 \& C_1' \& C_2' \mid C_0' \& C_1 \& C_2' \mid C_0' \& C_1' \& C_2)$

- The cell must have a combinational function FO defined on the output with respect to all its inputs. This function FO must logically define, together with the contention condition, a base function F^* that is the sum of n product terms, where the i th term contains all the inputs in C , with C_i high and all others low and exclusively one input in D . Examples of the defined function are as follows (for $n = 3$):

$$F^* = C_0 \cdot C_1' \cdot C_2' \cdot D_0 + C_0' \cdot C_1 \cdot C_2' \cdot D_1 + C_0' \cdot C_1' \cdot C_2 \cdot D_2'$$

or

$$F^* = C_0 \cdot C_1' \cdot C_2' \cdot D_0' + C_0' \cdot C_1 \cdot C_2' \cdot D_1' + C_0' \cdot \bar{C}_1' \cdot C_2 \cdot \bar{D}_2'$$

The function FO itself can take many forms, as long as it satisfies the following condition:

$$FO \cdot FC' == F^*$$

That is, when FO is restricted by FC' , it should be equivalent to F^* . The term $FO = F^*$ is acceptable; other examples are as follows (for $n = 3$):

$$FO = (D_0 \cdot C_0) + (D_1 \cdot C_1) + (D_2 \cdot C_2)$$

or

$$FO = (D_0' \cdot C_0) + (D_1' \cdot C_1) + (D_2' \cdot C_2)$$

Note that when FO is restricted by FC , inverting all inputs in D is equivalent to inverting the output; however inverting only a subset of D would yield an incompatible function. Although Design Compiler supports any form of FO that satisfies the condition $FO \cdot FC' == F^*$, it is recommended that you use a simple form (such as those described previously or F^*).

An example of a properly specified cell is as follows. For more information on cell definition, see the Library Compiler documentation.

Example 14-4 One-hot MUX Cell Definition

```
cell(OHMUX2) {
  ...
  contention_condition : "(C0 C1 + C0' C1')";
  ...
  pin(D0) {
    direction : input;
  ...
  }
  pin(D1) {
    direction : input;
  ...
  }
  pin(C0) {
    direction : input;
  }
}
```

```
... ...
}
pin(C1) {
direction : input;
...
}
pin(Z) {
direction : output;
function : "(C0 D0 + C1 D1)";
...
}
```

Optimization of One-Hot Multiplexers

Because a one-hot MUX implementation requires that exactly one control input be active at a time, composition is not supported. Composition of larger cells from smaller ones requires extra logic to ensure that exactly one control input is active at any time, which is inconsistent with the intention of the use of one-hot MUXes; in addition, the implementation of composition also depends on the actual electronic structure of the library cells.

However, if Design Compiler does not find an exact match in the library, it generates a warning message and maps an RTL MUX to a larger MUX—that is, more inputs—from the library and tie the additional inputs to ground. If the RTL one-hot MUX cannot fit into the largest one-hot MUX cell from the target library, the tool does not perform one-hot MUX mapping and issues a warning message. The MUX in this case is mapped the normal way.

15

Sequential Mapping

Sequential mapping phase consists of two steps: register inferencing and technology mapping. Synopsys uses the term *register* for both edge-triggered registers and level-sensitive latches. Before you read further, see [Optimization Flow](#) to understand how sequential mapping fits into the overall compile flow.

Register inferencing is the process by which the RTL description of a register is translated into a technology-independent representation called a SEQGEN. SEQGENs are created during elaboration and are usually mapped to flip-flops during compile. Technology mapping is the process by which a SEQGEN is mapped to gates from a specified target logic library. It is performed when you use the `compile_ultra` or `compile` command.

To learn about register inferencing and technology mapping, see

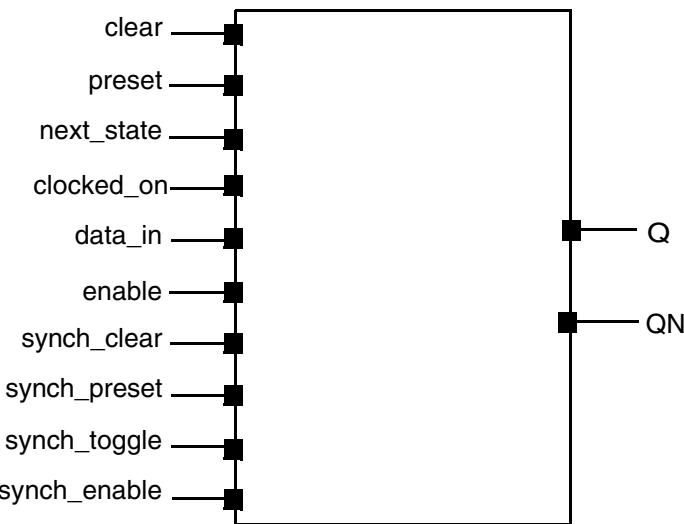
- [Register Inference](#)
- [Directing Register Mapping](#)
- [Specifying the Default Flip-Flop or Latch](#)
- [Reporting Register Types](#)
- [Unmapped Registers in a Compiled Design](#)
- [Automatically Removing Unnecessary Registers](#)
- [Merging Equal and Opposite Registers](#)
- [Inverting the Output Phase of Sequential Elements](#)

- [Mapping to Falling-Edge Flip-Flops](#)
- [Resizing Black Box Registers](#)
- [Preventing the Exchange of the Clock and Clock Enable Pin Connections](#)
- [Mapping to Registers With Synchronous Reset or Preset Pins](#)
- [Performing a Test-Ready Compile](#)
- [Using Register Replication](#)

Register Inference

When HDL Compiler reads in a Verilog or VHDL RTL description of the design, it translates the design into a technology-independent representation (GTECH). In GTECH, both registers and latches are represented by a SEQGEN cell, which is a technology-independent model of a sequential element as shown in [Figure 15-1](#). SEQGEN cells have all the possible control and data pins that can be present on a sequential element.

Figure 15-1 Generic SEQGEN Cell



[Table 15-1](#) lists the pins of a SEQGEN cell. Only a subset of these pins is used depending on the type of cell that is inferred. Unused pins are tied to zero.

Table 15-1 Pins of a SEQGEN Cell

Direction	Pin name	Function name and description	Cell type
Input	clear	Asynchronous clear (level-sensitive reset)	flip-flop or latch
	preset	Asynchronous preset (level-sensitive set)	flip-flop or latch
	enable	Asynchronous load (level-sensitive load of data_in)	flip-flop ¹ or latch
	data_in	Asynchronous data (loaded when enable==1)	flip-flop ¹ or latch

Table 15-1 Pins of a SEQGEN Cell (Continued)

Direction	Pin name	Function name and description	Cell type
	clocked_on	Synchronous clock (rising edge-triggered clock)	flip-flop
	next_state	Synchronous data (loaded at clocked_on edge)	flip-flop
	synch_enable	Synchronous load (load next_state at clocked_on edge)	flip-flop
	synch_clear	Synchronous clear (reset at clocked_on edge)	flip-flop
	synch_preset	Synchronous preset (set at clocked_on edge)	flip-flop
	synch_toggle	Synchronous toggle (toggle at clocked_on edge)	flip-flop
Output	Q	non-inverting output	flip-flop or latch
	QN	inverting output	flip-flop or latch

1. For register cells, the enable and data_in signals can specify synchronous reset or preset functionality when the `hdlin_ff_always_async_set_reset` variable is set to false.

Incorrect register inferencing results in incorrect technology mapping. One way to examine the type of register inferred is to examine the register inference reports in HDL Compiler. Set the following variable in HDL Compiler to generate additional information on inferred registers:

```
prompt> set_app_var hdlin_reporting_level verbose
```

[Example 15-1](#) shows an HDL Compiler inference report for a D flip-flop with a synchronous preset control.

Example 15-1 Inference Report

```
=====
| Register Name | Type      | Width   | Bus | MB | AR | AS | SR | SS | ST |
=====
| Q_reg        | Flip-flop | 1       | N  | N  | N  | N  | N  | Y  | N  |
=====
Sequential Cell (Q_reg)
Cell Type: Flip-Flop
Multibit Attribute: N
Clock: CLK
```

```
Async Clear: 0
Async Set: 0
Async Load: 0
Sync Clear: 0
Sync Set: SET
Sync Toggle: 0
Sync Load: 1
```

The inference report can help troubleshoot issues by indicating the type of register inferred (latch or flip-flop, single or multibit) and the control signals inferred for that register. The report shows the names of the nets connecting the pins on the SEQGEN elements in GTECH. In most cases, these control pins are tied to zero (inactive). If you are attempting to infer a synchronous or asynchronous reset or preset and it does not correctly appear in the inference report, check for issues in the specification of the register at the RTL level.

Correct inferencing of synchronous and asynchronous reset or preset control signals in the RTL results in connections to the synch_preset, synch_clear, preset, and clear pins on the SEQGEN GTECH cell. During technology mapping, Design Compiler uses the SEQGEN as the starting point for mapping. The sequential mapper checks the connections to the pins and the information present in the library cell descriptions when it maps to a logic library register.

Incorrect register inferencing or incomplete library information can produce unexpected results in mapping to the reset or preset pins of the register.

Design Compiler does not infer synchronous resets by default. To indicate to the tool which signals should be treated as synchronous resets, use the `sync_set_reset` Synopsys compiler directive in Verilog source files or the corresponding `sync_set_reset` Synopsys attribute in VHDL source files. HDL Compiler then connects these signals to the synch_clear and synch_preset pins on the SEQGEN in order to communicate to the mapper that these are the synchronous control signals and they should be kept as close to the register as possible. For information about inference of these signals, see the HDL Compiler documentation. For information about how Design Compiler maps these signals, see “[Mapping to Registers With Synchronous Reset or Preset Pins](#)” on page 15-17.

The correct mapping of asynchronous reset or preset registers requires that these registers be correctly described in the RTL and that corresponding cells exist in the logic library. As long as you follow the recommended coding guidelines for coding asynchronous registers, no special compiler directives are needed in order to infer asynchronous set or reset signals. For more information on inference of these signals, see the HDL Compiler documentation.

Directing Register Mapping

You can control register mapping in the following ways:

- Use the `set_register_type` command.

For more information, see [Specifying the Default Flip-Flop or Latch](#).

- Use exact mapping with the `-exact_map` option.

Use the `-exact_map` option with the `compile_ultra` or `compile` command to restrict the mapping to sequential cells with simple behavior (synchronous set and reset, synchronous toggle, synchronous enable, asynchronous set and reset, and asynchronous load and data). When you use the `-exact_map` option, sequential mapping does not try to encapsulate combinational logic originally outside the generic sequential element (SEQGEN) into the sequential cell.

- Use test-ready compile with the `compile_ultra -scan` or `compile -scan` command.

For more information, see [Performing a Test-Ready Compile](#).

Specifying the Default Flip-Flop or Latch

The `set_register_type` command specifies the default flip-flop or latch library cell type for some or all registers in the current design or current instance. A flip-flop type is represented by an example flip-flop; any flip-flop that has the same sequential characteristics as the specified flip-flop is considered to be of that type.

Use the `set_register_type` command to direct Design Compiler to infer a particular flip-flop or latch.

Note:

This mechanism for directing register types is not needed if you write the HDL to directly infer the correct register type.

You can specify a latch, a flip-flop, or both.

To set the default flip-flop type to FFX and the default latch type to LTCHZ, enter

```
prompt> set_register_type -flip_flop FFX -latch LTCHZ
```

Specifying a Constant Value for Disabled Registers

If your design has registers that are never enabled, the tool assumes that the disabled register is a constant (either 0 or 1) register when it performs optimization. You can specify a constant value for a disabled register as an initial state by using the `set_uninitialized_register_value` command. The constant value can either be 0 or 1. After you specify the constant value, as shown in the following example, the tool optimizes the register and the logic connected to the register using the value you specified.

```
dc_shell> set_uninitialized_register_value u1/reg[0] -value 0
```

In this example, the command sets the value of the u1/reg[0] register to 0. The tool uses this value to optimize the register and the logic connected to the register.

If you use the `set_uninitialized_register_value` command on a register whose enable input is not disabled, the tool ignores the command and does not issue any warning message.

Reporting Register Types

To learn how to report the current default register type specification for the design and for cells, see

- [Reporting the Register Type Specifications for the Design](#)
 - [Reporting the Register Type Specifications for Cells](#)
-

Reporting the Register Type Specifications for the Design

To list the current default register type specifications, use the `report_design` command:

```
prompt> report_design
*****
Report : design
Design : DESIGN
Version: ...
Date   : ...
*****
...
Flip-Flop Types:
    Default: FFX, FFXHP, FFXLP
```

Reporting the Register Type Specifications for Cells

To lists the current register type specifications for cells, use the `report_cell` and `all_registers` commands.

The following example shows a report:

```
*****
Report : cell
Design : reg_type
Version: ...
Date   : ...
*****
Attributes:
```

```

n - noncombinational
...
Cell      Reference      Library      Area      Attributes
-----
ffa       FFY           MY_LIB       9.00      1,n
ffb       FFY           MY_LIB       9.00      1,n
ffc       FFY           MY_LIB       9.00      1,n
ffd       FFY           MY_LIB       9.00      1,n
-----
Total 4 cells                      36.00

Flip-Flop Types:
 1 - Exact type FFY

```

Unmapped Registers in a Compiled Design

When Design Compiler fails to find a match for a register in the available target logic library, it issues the following warning in the compile log:

```
Warning: Target library contains no replacement for register
'Q_reg' (**FFGEN**). (TRANS-4)
```

In addition, the compiled gate-level netlist has the following type of cell:

```
\**FFGEN** Q_reg ( .next_state(D), .clocked_on(CLK),
.force_00(1'b0), .force_01(N0), .force_10(1'b0),
.force_11(1'b0), .Q(Q) );
```

****FFGEN**** is a model of the register functionality, similar to the SEQGEN. It is not a cell that you find in any logic library. Check the names of the cells mapped to ****FFGEN****; these are the cells for which Design Compiler could not find a match in the logic library. This behavior usually occurs when asynchronous registers or latches are inferred in the RTL but a cell with the corresponding asynchronous functionality is not available for mapping in the specified target libraries.

Check the failing registers to see the types of registers that you are attempting to infer:

- Are you inferring positive-edge or negative-edge clocked registers?
- What sets of asynchronous control pins are being inferred (reset only, preset only, both reset and preset)?
- Are you performing power gating on these cells (using retention registers)?
- Are you using a test-ready compile (`compile_ultra -scan` or `compile -scan`)?

Make sure that your target library has cells with all the features that you are attempting to use for register mapping. Verify that you have not disabled the use of the required cells with

the `set_dont_use` command. Use the following command to check the `set_dont_use` settings:

```
prompt> write_environment -environment_only
```

Automatically Removing Unnecessary Registers

During sequential mapping, Design Compiler can save area significantly by automatically detecting and removing unconnected registers and constant registers:

- [Removing Unconnected Registers](#)
- [Eliminating Constant Registers](#)

Removing Unconnected Registers

During optimization, Design Compiler deletes registers having outputs that do not drive any loads. The combinational logic cone associated with the input of the register can also be deleted if the cell is not used elsewhere in the design. Register outputs can become unconnected due to redundancy in the circuit or as a result of constant propagation. In some designs where the registers have been instantiated, the outputs might already be unconnected.

You can preserve such registers if you need to maintain consistency between the compiled design and the HDL source or for other design reasons, such as in the case of instantiated cells. To direct Design Compiler to preserve the registers, set the

`compile_delete_unloaded_sequential_cells` variable to `false` before you compile.

The default is `true`. When this variable is set to `false`, a warning message appears during compilation, indicating the presence of unloaded registers:

```
Warning: In design 'design_name', there are sequential cells  
not connected to any load. (OPT-109)
```

```
Information: Use the 'check_design' command for more  
information about warnings. (LINT-99)
```

You can use the `check_design` command after `compile` to identify cell instances that have unconnected outputs.

In cases where a feedback loop exists with no path from any section of the loop to a primary output, no warning appears because the output of the register is theoretically connected.

When the `compile_delete_unloaded_sequential_cells` variable is set to `true`, such cells are optimized away. Setting the value to `false` retains the cells and the logic cone associated with the inputs to the cells.

Eliminating Constant Registers

Certain registers in a design might never change their state because they have constant values on one or more input pins. These constant values can either be directly at the input or result from the optimization of fanin logic that eventually leads to a constant input of the register. Eliminating such registers can improve area significantly.

During compile, Design Compiler performs constant propagation to automatically find and replace such sequential elements with a constant.

[Table 15-2](#) lists cases in which a sequential element can be eliminated.

Table 15-2 Cases for Which Sequential Elements Are Eliminated

Type of register	Data	Preset	Reset
Simple, constant data	1 or 0		
Preset and constant data 1	1	X	
Constant preset	X	1	
Reset and constant data 0	0		X
Constant reset	X		1
Preset, reset, and constant data 1; reset always inactive	1	X	0
Preset, reset, and constant data 0; preset always inactive	0	0	X

Controlling Constant Propagation Optimization

Constant propagation optimization is controlled by the following:

- Constant register removal is enabled by default when you run the `compile_ultra` command and the `compile_seqmap_propagate_constants` variable is set to `true` (the default).

When the variable setting is `true`, the Design Compiler tool performs constant propagation, and issues an informational message in the log file when it removes a constant register in the design:

Information: The register '`Z_reg`' is a constant and will be removed. (OPT-1206)

Constant propagation occurs for both unmapped and mapped designs when the `compile_seqmap_propagate_constants` variable is set to `true`. However, the best

results are obtained when constant propagation is enabled during the initial sequential mapping from an unmapped design.

Constant register removal takes the value of DFT control signals into account to identify constant registers in non-scan mode. For example, if you apply the following constraints before running the `compile_ultra` command, the tool assumes that the value of the `test_mode` and `test_se` signals are 0 when it identifies the constant registers:

```
dc_shell> set_dft_signal -view spec -type TestMode -port test_mode \
                     -active_state 1

dc_shell> set_dft_signal -view spec -type ScanEnable -port test_se \
                     -active_state 1
```

Checking the value of the DFT control signals allows Design Compiler to remove the registers that have constant input only in non-scan mode. Design Compiler generates `guide_scan_input` command guidance in the .svf file to store the DFT signal value information for these registers that have constant input only in non-scan mode. The registers that are already scan stitched are not removed. This constant register removal functionality is not performed during incremental compile.

You can disable the constant register removal algorithm in the `compile_ultra` command by setting the `compile_seqmap_propagate_high_effort` variable to `false`. You can disable constant register removal completely by setting the `compile_seqmap_propagate_constants` variable to `false`. Both variables are `true` by default.

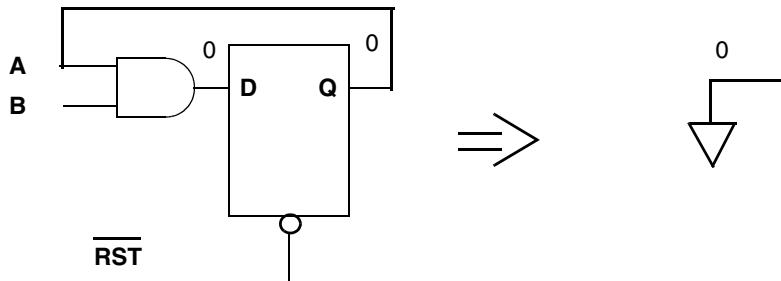
- Constant register removal is enabled by default when you run the `optimize_netlist -area` command. You can disable constant register removal in the `optimize_netlist -area` command by setting either of the `compile_seqmap_propagate_constants` or `compile_seqmap_propagate_high_effort` variable to `false`. Both variables are `true` by default.

Constant Propagation Optimization for Complex Conditions

Design Compiler removes sequential elements for which the logic leading to a constant value is particularly complex. That is, it attempts to identify more complex conditions leading to a register input. It analyzes each sequential element to determine its reset state. If a known reset state can be determined, Design Compiler checks whether the sequential element can switch state after it has reached its known reset state. If the sequential element cannot escape its reset state, Design Compiler replaces it with a constant equivalent to its reset state.

[Figure 15-2](#) illustrates such a case. A feedback path leads from the Q pin of the register back to the D input; the register cannot escape its reset state.

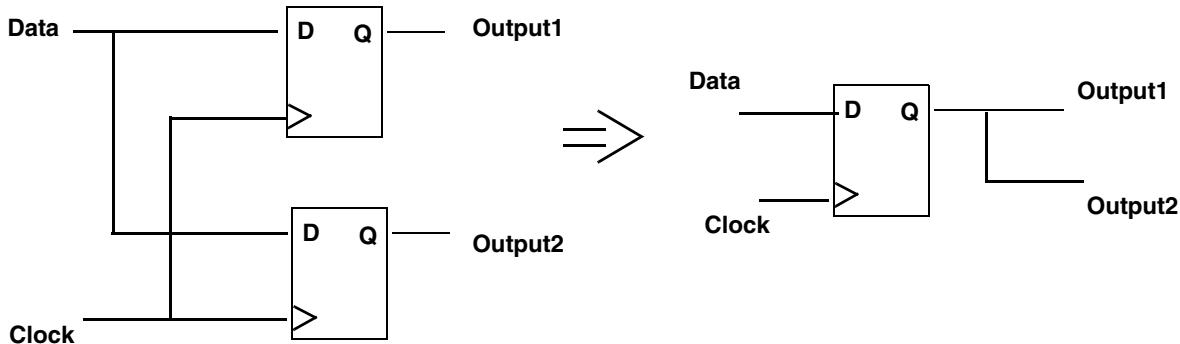
Figure 15-2 Constant Output Sequential Element That Cannot Escape Its Reset State



Merging Equal and Opposite Registers

By default, Design Compiler identifies and merges equal and opposite registers. Two registers that are equal in all states that can be reached from the reset state can be replaced with a single register driving both sets of loads. The same principle applies to registers that are opposite to each other. In addition to removing the registers, this type of optimization enables equal and opposite information to propagate beyond the registers so that subsequent logic can be optimized in the context of equal and opposite relationships. Equal and opposite register merging is shown in Figure 15-3.

Figure 15-3 Merging Equal and Opposite Registers



Design Compiler issues the following message when it merges registers:

Information: In design 'test', the register 'u2/op_reg' is removed because it is merged to 'u2/op1_reg'. (OPT-1215)

To disable register merging on specific cells or blocks, set the `set_register_merging` command to `false` on those cells or blocks. If you want to prevent register merging on all registers in your design, set the `compile_enable_register_merging` variable to `false`. If

you set the `compile_enable_register_merging` variable to `false`, you cannot enable any register merging, that is, the `compile_enable_register_merging` variable setting takes precedence over the `set_register_merging` command.

To make sure that the `register_merging` attribute is set on a cell or design, use the `report_attribute` command. For example,

```
prompt> report_attribute u1/op_reg
```

Design	Object	Type	Attribute Name	Value
test	u1/op_reg	cell	register_merging	false
test	u1/op_reg	cell	is_a_generic_seq	true
test	u1/op_reg	cell	ff_edge_sense	

See Also

- [Boundary Optimization](#)

Provides information about the propagation of equal and opposite information across the hierarchy

Inverting the Output Phase of Sequential Elements

Sequential phase inversion (allowing sequential elements to have their output phase inverted) is enabled by default when you use the `compile_ultra` command. In certain cases, you might be inferring a register that has one type of asynchronous control but your library has the opposite type of pin. For example, you might be inferring an asynchronous set but your library has only registers with asynchronous clear pins.

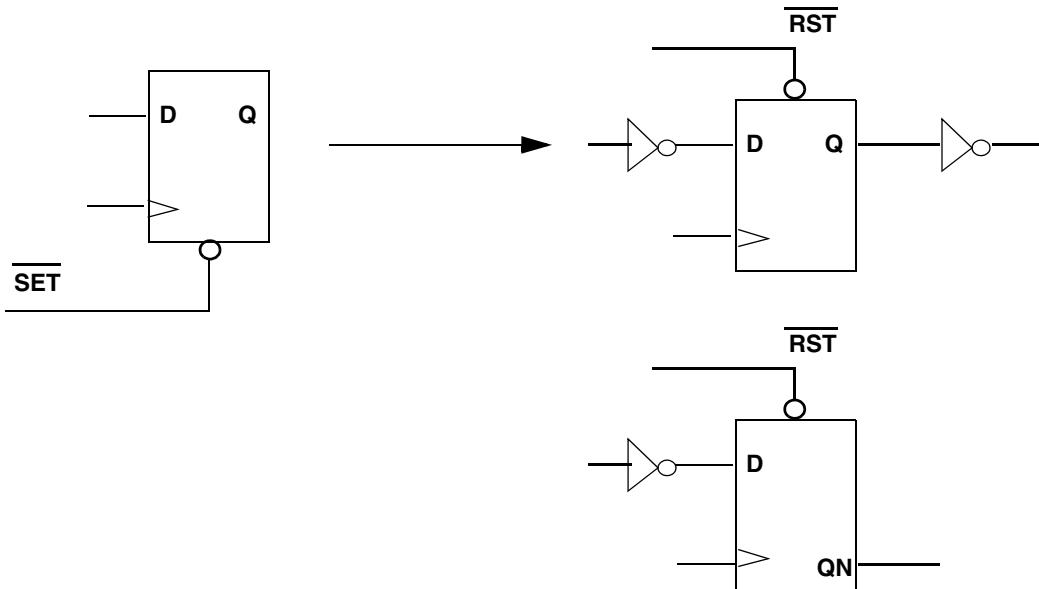
During mapping, Design Compiler issues the following warning message:

```
Warning: Target library contains no replacement for register  
'Q_reg' (**FFGEN**). (TRANS-4)
```

In such cases, Design Compiler maps to the opposite type of register and inverts all the data inputs and outputs.

[Figure 15-4](#) shows an example of output inversion transformations.

Figure 15-4 Output Inversion Transformations



Information about inverted registers is written to the verification guidance .svf file. The following informational message appears in the log file to remind you that you must include the .svf file in Formality when verifying designs with output inversion:

Information: Sequential output inversion is enabled. SVF file must be used for formal verification. (OPT-1208)

You can disable sequential output inversion during synthesis by using either of the following commands:

- `compile_ultra -no_seq_output_inversion`
- `optimize_netlist -area -no_seq_output_inversion`

Mapping to Falling-Edge Flip-Flops

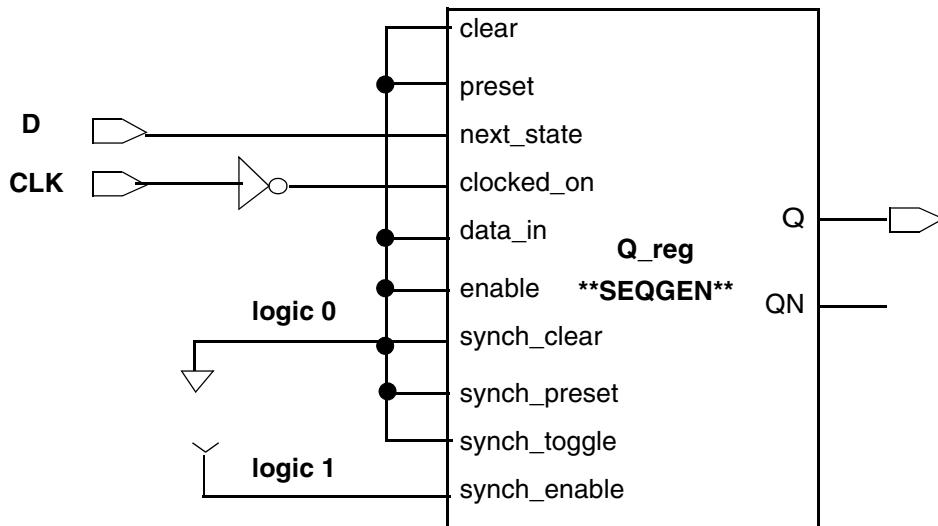
Falling-edge flip-flops are inferred by HDL Compiler by referencing the falling edge of the clock in the process describing the register. The resulting SEQGEN shows an inversion of the `clocked_on` pin as shown in [Figure 15-5](#).

HDL Compiler creates the SEQGEN shown in [Figure 15-5](#) for the following Verilog design:

```
module dff_inv_clk (D,CLK,Q);
  input D,CLK;
  output reg Q;
```

```
always @ (negedge clock)
  Q = D;
endmodule
```

Figure 15-5 SEQGEN Created by HDL Compiler



Note:

The inverter on the clock net in the GTECH is only used by the tool to represent an inverted clock for mapping the registers. It does not imply that an inverter is put on the clock net.

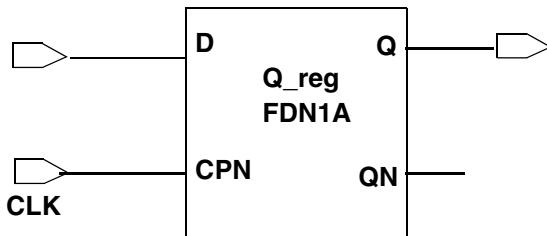
[Example 15-2](#) shows the inference report.

[Example 15-2 Inference Report](#)

```
=====
| Register Name | Type      | Width | Bus | MB | AR | AS | SR | SS | ST |
=====
| Q_reg         | Flip-flop | 1    | N  | N  | N  | N  | N  | N  | N  |
=====
Sequential Cell (Q_reg)
  Cell Type: Flip-Flop
  Multibit Attribute: N
  Clock: CLK'
  Async Clear: 0
  Async Set: 0
  Async Load: 0
  Sync Clear: 0
  Sync Set: 0
  Sync Toggle: 0
  Sync Load: 1
```

HDL Compiler sets attributes so that the register is mapped to a flip-flop with an inverted clock input as shown in [Figure 15-6](#). You do not have to define any constraints to enable mapping to falling-edge flip-flops. Given a library with both positive edge and negative edge-triggered flip-flops, Design Compiler creates the optimized negative edge-triggered design as shown in [Figure 15-6](#).

Figure 15-6 Register Mapped To Logic Library



Resizing Black Box Registers

Design Compiler supports the `user_function_class` attribute for cells that cannot be functionally modeled for synthesis. Design Compiler treats black box cells with the same `user_function_class` attribute and the same number of pins as functionally equivalent. The tool can resize such cells by using other cells with the same `user_function_class` attribute value, as long as timing arcs to the output pins of the cells are provided in the target logic library. This capability works for both combinational and sequential library cells.

If this attribute has not already been set on your black box registers in the library source (.lib) file, you can set it by using the `set_attribute` command. During register-sizing optimizations, registers with the `attribute` setting can be exchanged with other registers having the same `user_function_class` attribute. For example,

```
set_attribute [get_lib_cells my_lib/DFFX*] \
              user_function_class DFFX -type string
```

```
Information: Attribute 'user_function_class' is set on 4
objects. (UID-186)
my_lib/DFFX1 my_lib/DFFX2 my_lib/DFFX4 my_lib/DFFX8
```

Note that by default, Design Compiler does not resize integrated clock-gating cells. Such cells have the `clock_gating_integrated_cell` attribute set on the library cell and can only be sized by Power Compiler. Use the `identify_clock_gating` command to identify existing integrated clock-gating cells in the design.

Preventing the Exchange of the Clock and Clock Enable Pin Connections

Some libraries have flip-flops with both clock and clock enable pins. Although Design Compiler can successfully map to both of these pins, incomplete information in the library can lead to situations in which the clock and clock enable pins are exchanged during mapping.

In the library description of the flip-flop (.lib file), all pins included in the `clocked_on` attribute are treated as clocks by default. The `pin_func_type` attribute alone is not sufficient to distinguish a clock enable pin from a clock pin. You must also set the `clock` attribute to false on clock enable pins to prevent these pins from being treated as clocks by the sequential mapper.

Clock nets are distinguished from clock enable nets in the design based on clock constraints propagated to these nets. The following simplified library example shows how you can distinguish the clock enable pin (CE in the example) from the clock pin (CLK in the example) by specifying the `pin_func_type` attribute as `clock_enable` and also by setting the `clock` attribute to false on this pin.

```
cell (DFFCE) {
  ff (IQ, IQN) {
    next_state : "D" ;
    clocked_on : "CLK & CE" ;
  }
  pin (CLK) {
    direction : input;
    clock : true;
  }
  pin (CE) {
    direction : input;
    pin_func_type : clock_enable;
    clock : false ;
  }
  ...
}
```

Mapping to Registers With Synchronous Reset or Preset Pins

Before you read further, see [Register Inference](#) for information on inference of synchronous reset or preset pins.

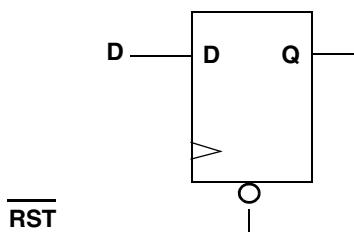
By default, Design Compiler does not map to synchronous reset or preset flip-flops or preserve the synchronous reset or preset logic close to the flip-flop. However, you can enable this behavior by setting the `compile_seqmap_honor_sync_set_reset` variable to `true`. When the variable is enabled, flip-flops with synchronous resets or presets are mapped in one of the following ways:

- If your library has registers with synchronous reset (or preset) pins, the reset (or preset) net is connected to the reset (or preset) pin of a register with a dedicated reset pin.
- If your library does not have any registers with synchronous reset (or preset) pins, the tool adds extra logic to the data input to generate the reset (or preset) condition on a register without a reset (or preset) pin. In these cases, Design Compiler attempts to map the logic as close as possible to the data pin to minimize X-propagation problems that lead to synthesis/simulation mismatches.

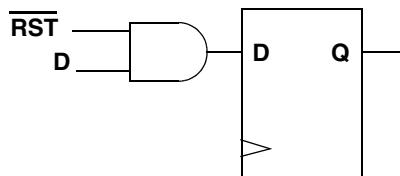
[Figure 15-7](#) shows examples of mapping to registers with and without a synchronous reset pin.

Figure 15-7 Mapping To Registers With and Without Synchronous Reset Pins

Synchronous Reset Using a Reset Pin



Synchronous Reset Using a Gate on the Data Pin



If your library does have registers with synchronous reset (or preset) pins, it is still important to use the `sync_set_reset` directive in your RTL so that Design Compiler can distinguish the reset (or preset) signals from other data signals and connect the reset signal as close to the register as possible.

Note:

Synchronous reset and preset signals are not inferred for level-sensitive latches. A synchronous reset or preset coding style on a latch always results in combinational logic on the data signal even if the library has latches with synchronous reset or preset pins.

Even if your library does contain registers with synchronous reset or preset pins, these registers are used for mapping. However, you must ensure that the synchronous pins in the library have the `nextstate_type` attribute correctly defined for each of these pins to correctly connect the pins.

The `nextstate_type` attribute is predefined for use in the .lib file as follows:

```
nextstate_type : data | preset | clear | load | scan_in |
scan_enable;
```

The `nextstate_type` attribute is used by the sequential mapper in order to distinguish between the different types of synchronous input pins. Without this information, the synchronous data, reset, and clear pins are all treated as synchronous data pins and may be exchanged during mapping.

[Table 15-3](#) describes the attribute values and their corresponding short integer value.

Table 15-3 Values Used for the `nextstate_type` Attribute

Enum value used in the .lib file	Short type value in Design Compiler	How the pin is identified
data	0	Synchronous data pin (the default)
preset	1	Synchronous preset pin
clear	2	Synchronous reset pin
load	3	Synchronous load pin
scan_in	4	Synchronous scan in pin
scan_enable	5	Synchronous scan enable pin

[Example 15-3](#) shows how the `nextstate_type` attribute (in **bold** type) is specified in the library for a synchronous reset register. Other necessary attributes have been omitted for the sake of clarity.

Example 15-3 Specifying the `nextstate_type` Attribute

```
cell (DFFSRST) {
  ff (IQ, IQN) {
    next_state : "RN D" ;
    clocked_on : "CLK" ;
  }
  pin (CLK) {
    direction : input ;
    clock : true ;
  }
  pin (D) {
    direction : input ;
    nextstate_type : data;
  }
  pin (RN) {
    direction : input ;
    nextstate_type : clear;
  }
}
```

```

pin (Q) {
  direction : output ;
  function : "IQ" ;
}
} /* end of cell DFFSRST

```

Note how the `nextstate_type` attribute has been added to the pin groups for the data and synchronous reset pins. Pins included in the `next_state` attribute have a default `nextstate_type` of data unless otherwise specified. This setting might result in the swapping of the data and synchronous reset or preset pins if the `nextstate_type` attribute is not assigned.

You should therefore check to ensure that the `nextstate_type` attribute has been correctly added to the synchronous input pins of registers with synchronous reset or preset pins in your library.

If you are compiling the .lib file using Library Compiler, pay special attention to the following types of warnings for your synchronous set or reset registers:

```

Warning: Line 5905, The 'DFFSRST' cell is missing the
"nextstate_type" attribute for some input pin(s) specified
in 'next_state' of its ff/ff_bank group. (LIBG-243)

```

You can examine the .lib file if it is available. Otherwise you can check the library by querying for the attribute in dc_shell. The following example shows that the attributes have been correctly defined in the library.

```

prompt> read_db mytechlib.db
prompt> get_attribute [get_lib_pin MYTECHLIB/DFFSRST/D] nextstate_type
0

prompt> get_attribute [get_lib_pin MYTECHLIB/DFFSRST/RN] nextstate_type
2

```

The following example shows that the attributes are not correctly defined in the library:

```

prompt> get_attribute [get_lib_pin MYTECHLIB/DFFSRST/D] nextstate_type
Warning: Attribute 'nextstate_type' does not exist on port
'D'. (UID-101)

```

Ideally, you should add the attributes to the library source .lib file. Otherwise, you can add the attributes to the library pins in Design Compiler prior to using the library for synthesis. Note that you use the short type integer value in Design Compiler to add the missing attributes. See [Table 15-3](#) for the appropriate value to use for each input pin type.

```

prompt> set_attribute [get_lib_pins MYTECHLIB/DFFSRST/D] \
    nextstate_type 0 -type short

prompt> set_attribute [get_lib_pins MYTECHLIB/DFFSRST/RN] \
    nextstate_type 2 -type short

```

After updating the library in memory by using these commands, you can save an updated version of the library by using the `write_lib` command as follows:

```
prompt> write_lib -format db MYTECHLIB -output mytechlib.fixed.db
```

For more information, see the Library Compiler documentation.

If the reset or preset registers are not being correctly mapped, check the following:

- Have you used the `sync_set_reset` compiler directive in your RTL to identify the set or reset?
- Does the register inference report from show that synchronous set or reset has been correctly inferred?
- Does the library have the `nextstate_type` attribute defined for the synchronous input pins?

Performing a Test-Ready Compile

Test-ready compile integrates logic optimization and scan replacement. During the first synthesis pass of each HDL design or module, test-ready compile maps all sequential cells directly to scan cells. The optimization cost function considers the impact of the scan cells themselves and the additional loading due to the scan chain routing.

For a test-ready compile, you specify a scan style by using the `test_default_scan_style` or the `set_scan_configuration -style` command. Include the `-scan` option to the `compile_ultra` command or `compile` command when compiling the design. This section describes the following topics:

- [Overview of Test-Ready Compile](#)
- [Scan Replacement](#)
- [Selecting a Scan Style](#)
- [Mapping to Libraries Containing Only Scan Registers](#)
- [Mapping to the Dedicated Scan-Out Pin](#)
- [Automatic Identification of Shift Registers](#)

See Also

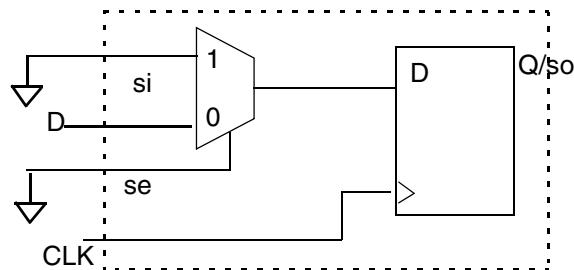
- [Test Synthesis in Topographical Mode](#)

Overview of Test-Ready Compile

During test-ready compile, the tool replaces regular flip-flops with flip-flops that contain logic for testability. [Figure 15-8](#) shows an example of how a D flip-flop is replaced with a scan register during test-ready compile. This type of architecture, a multiplexed flip-flop, incorporates a two-input MUX at the input of the D flip-flop. The select line of the MUX enables two modes—functional mode (normal data input) or test mode (scanned data input). In this example, the scan-in pin is *si*, the scan-enable pin is *se*, and the scan-out pin, *so*, is shared with the functional output pin, Q.

Other architectures are supported: clocked scan, level-sensitive scan design (LSSD), and auxiliary-clock LSSD. The scan style dictates the appropriate scan cells to insert during optimization. You can change the default scan style by using the `test_default_scan_style` or the `set_scan_configuration -style` command. For more information, see the DFT Compiler documentation.

Figure 15-8 Example of a Scan Register Used During Test-Ready Compile



Note:

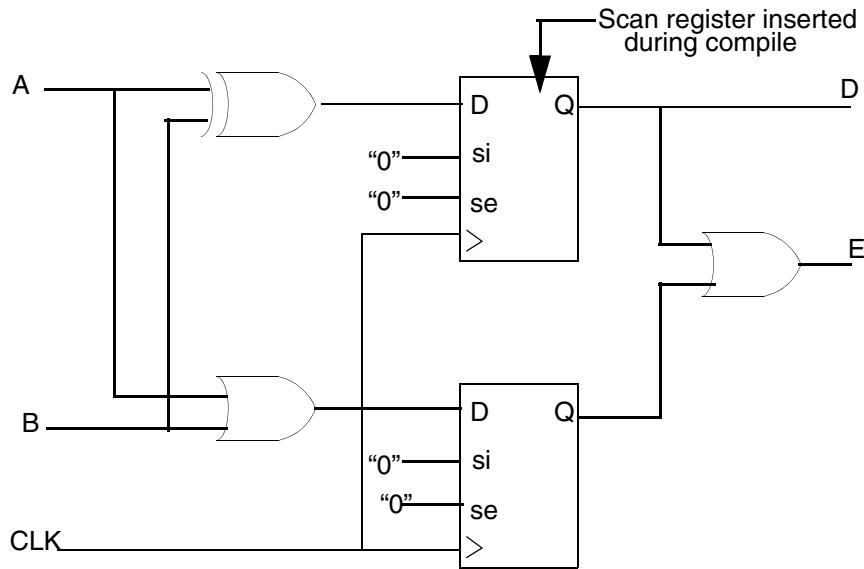
Scan connections are removed and modeled internally with pin or net capacitance. This results in improved QoR for both test-ready and scan-stitched designs. It also leads to better optimization opportunities, such as optimal cell sizing.

When you use test-ready compile, Design Compiler does the following:

- Maps all sequential cells directly to scan registers in your library
- Ties the test control pins to the appropriate state to enable functional mode

[Figure 15-9](#) shows the result of a test-ready compile.

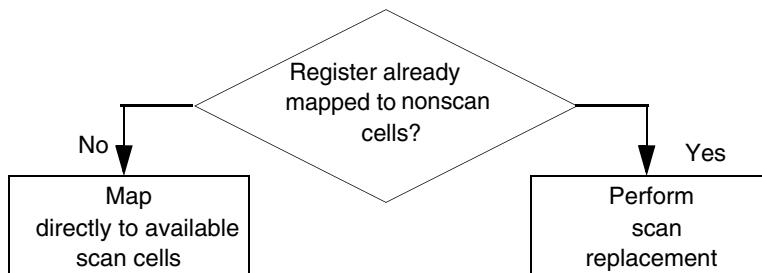
Figure 15-9 Result of Test-Ready Compile



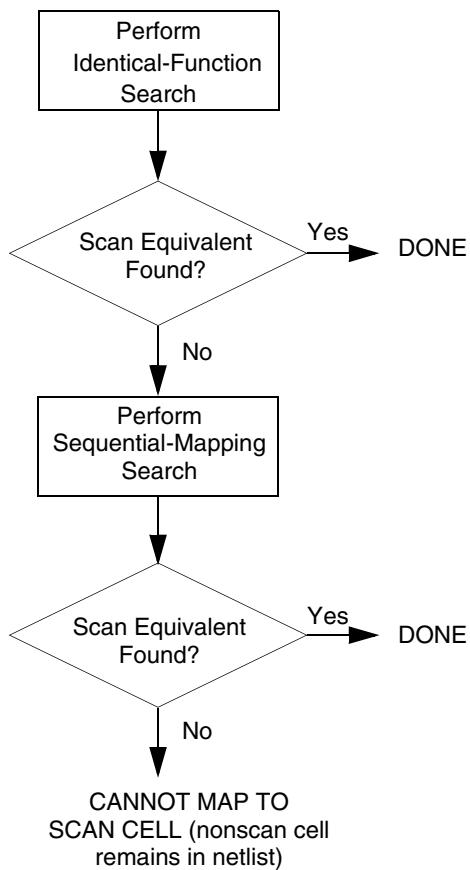
After test-ready compile, you have a design that contains unrouted scan cells (prescan design) and you are ready to perform scan assembly. When you use the `insert_dft` command, DFT Compiler stitches the scan registers to form a scan chain, similar to a serial shift register. During scan mode, a combination of patterns are applied to the primary input and shifted out through the scan chain, providing fault coverage for the combinational and sequential logic in the design. For more information, see the DFT Compiler documentation.

Scan Replacement

When you perform a test-ready compile on an unmapped design, the tool maps sequential cells directly to scan registers in your library. If you map a design without using the `-scan` option to the `compile_ultra` command or the `compile` command and then include the `-scan` option in a subsequent compile, the tool uses the scan replacement algorithm as shown in [Figure 15-10](#). However, the recommended method for test-ready compile is to use the `-scan` option of the `compile_ultra` command or the `compile` command on an unmapped design.

Figure 15-10 Mapping to Scan Cells

In the scan replacement flow shown in [Figure 15-11](#), compile first maps sequential cells to nonscan cells from the logic library and then scan-replaces the nonscan cells with scan cells from the logic library.

Figure 15-11 Scan Replacement

Therefore, it is important to have a complete set of nonscan cells as well as equivalent scan cells to obtain the best mapping for test-ready compile. If your library is missing a nonscan equivalent cell for a scan cell, then the tool does not use that scan cell. For example,

assume that you have a scan cell with a synchronous enable pin but you do not have a nonscan cell with this pin. Registers with an enable condition are mapped by using extra logic on the data pin because no nonscan cell is available with an enable pin for the initial mapping.

Identical-function search locates a scan equivalent, using the information in the `test_cell` group of the library description. A valid scan equivalent must meet the following conditions:

- The scan cell and the nonscan cell must have the same functional input and output pins.
- The functional description in the `test_cell` group of the library description must exactly match the functional description of the nonscan cell.

Sequential-mapping search locates a scan equivalent by using the Design Compiler sequential-mapping algorithms. DFT Compiler uses this search only for edge-triggered nonscan sequential cells. Sequential mapping selects the lowest-cost scan equivalent, which can be a scan cell plus combinational gates.

Identical-function search is faster than sequential-mapping mode but might not find a scan equivalent in some cases, such as for complex flip-flops. If neither identical-function search nor sequential-mapping search can find a scan equivalent, DFT Compiler leave the nonscan cell in the design without performing scan replacement and issues the following message:

```
TEST-120 (Warning) No scan equivalent exists for cell
```

If you did not use test-ready compile, DFT Compiler can also perform scan replacement during scan assembly. However, for best results, use test-ready compile to map to scan cells when your starting point is an HDL description.

For additional information about how to assemble scan structures and analyze scan operations, see the DFT Compiler documentation.

Selecting a Scan Style

If you are using test-ready compile to insert scan structures in your design, you must select a scan style before you perform logic synthesis. You must use the same scan style for all modules of your design.

Select a scan style based on your design style and on the types of scan cells available in your target logic library. See the DFT Compiler documentation for more information on considerations for selecting a scan style.

Specify the scan style by setting the `test_default_scan_style` variable. The scan style defined by the `test_default_scan_style` variable applies to all the designs in the current session. You can also use the `set_scan_configuration -style` command to specify the scan style. However, this command applies only to the current design. If your selected scan style differs from the default scan style, you must execute this command for each module.

See the DFT Compiler documentation for details on the `set_scan_configuration` command.

Table 15-4 shows the scan style keywords to use when specifying the scan style. You can use these keywords with either the `test_default_scan_style` variable or the `set_scan_configuration -style` command.

Table 15-4 Scan Style Keywords

Scan style	Keyword
Multiplexed flip-flop	<code>multiplexed_flip_flop</code>
Clocked scan	<code>clocked_scan</code>
Level-sensitive scan design (LSSD)	<code>lssd</code>
Auxiliary-clock LSSD	<code>aux_clock_lssd</code>

Mapping to Libraries Containing Only Scan Registers

Design Compiler automatically maps directly to scan registers from your library without going through a scan-replacement step. Scan-replacement is not possible with a scan-only library. You must always use the `-scan` option of the `compile_ultra` command for these libraries.

Mapping to the Dedicated Scan-Out Pin

Some registers in your library might have both a functional output pin and a dedicated scan-out pin. These dedicated scan-out pins must be identified by the setting following attribute on the output pins in the .lib file:

```
test_output_only:true;
```

By default, the `compile -scan` command does not use these pins for functional output connections. This behavior is controlled by the following integer variable (set to 1 by default):

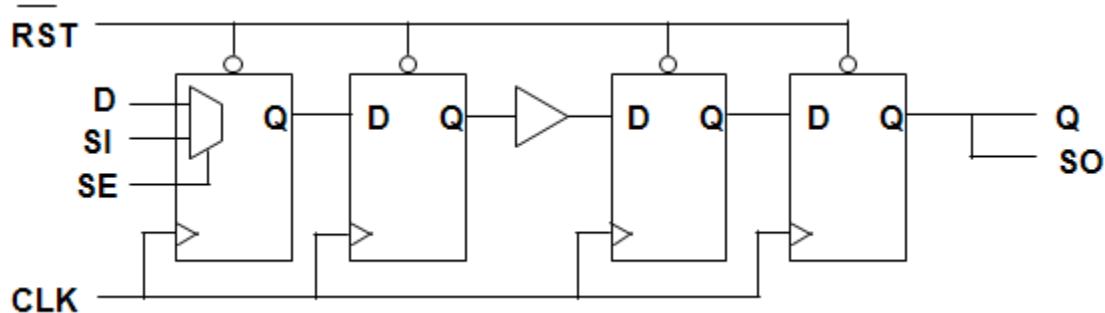
```
set compile_dont_use_dedicated_scanout 1
```

If you observe that `compile` is making use of these pins for functional connections, be sure that this variable is not set to 0 in your scripts, and make sure that your library pins have the correct attribute settings.

Automatic Identification of Shift Registers

Test-ready compile with DC Ultra can automatically identify shift registers in the design and perform scan replacement on only the first register. This capability improves the sequential design area and reduces congestion by using fewer scan-signals for routing. [Figure 15-12](#) shows an example.

Figure 15-12 Example of Shift Register Identification



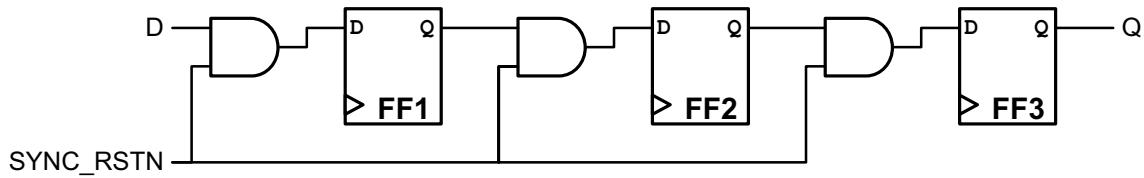
When DC Ultra identifies these shift registers, DFT Compiler recognizes these identified shift registers as shift-register scan segments. If required, DFT Compiler breaks these scan segments to satisfy the test setup requirements, such as maximum chain length.

Shift register identification is not performed across hierarchical boundaries when only a 2-bit shift register would be created. This typically occurs with registered interfaces where a single register at the output of one design is connected to a single register at the input of another design. This behavior reduces port punching performed during test insertion when the tool connects scan chains to shift registers that cross hierarchical boundaries, and it can help improve congestion and scan wire length results. Shift register identification is performed across hierarchical boundaries for 3-bit and longer shift registers.

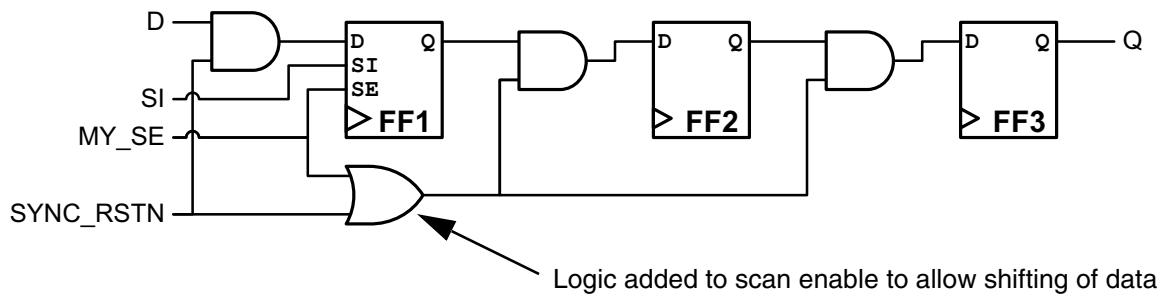
Shift registers containing synchronous logic between the registers can be identified if the synchronous logic is controlled such that the data can be shifted from the output of the first register to the input of the next register. This synchronous logic can either be internal to the register, such as synchronous reset and enable, or it can be external synchronous logic, such as multiplexor logic between the registers. [Figure 15-13](#) shows an example of shift register identification with synchronous reset.

Figure 15-13 Example of Shift Register Identification with Synchronous Reset

After compile_ultra -scan



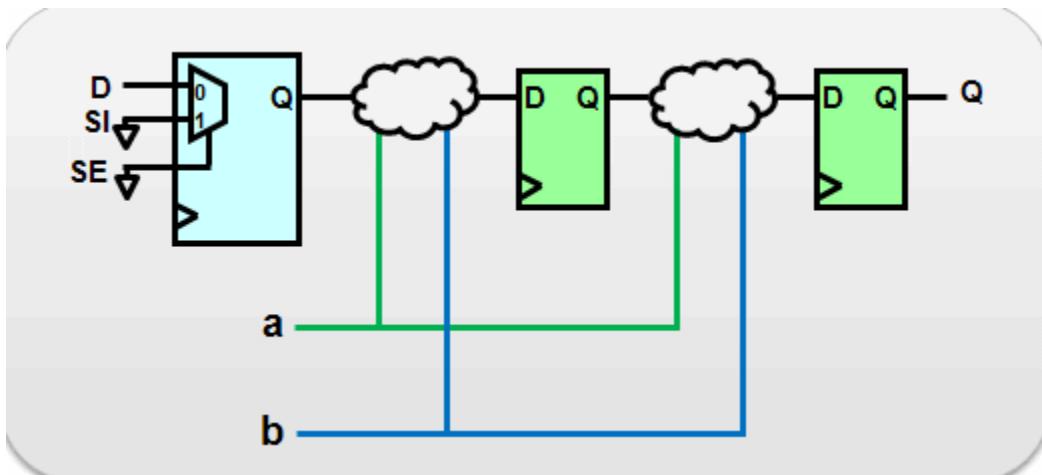
After insert_dft



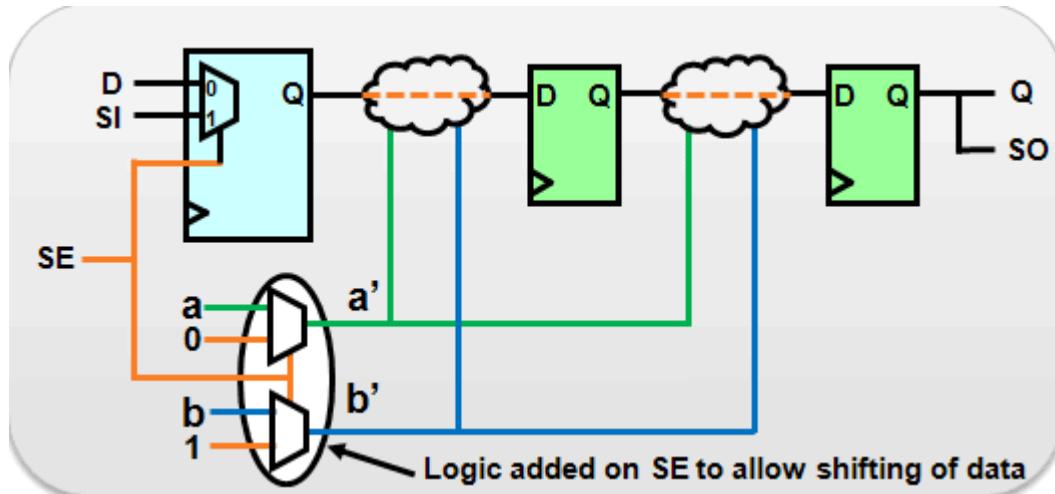
[Figure 15-14](#) shows an example of external synchronous logic with more complex logic between the registers.

Figure 15-14 Example of External Synchronous Logic With Complex Logic Between the Registers

After compile_ultra -scan



After insert_dft



For shift registers identified with synchronous logic between the registers, DFT Compiler adds additional logic to the scan-enable signal during scan insertion. The extra logic allows the data to be shifted between the registers in scan mode. This extra logic results in shared paths between the scan-enable signal and the functional logic. Therefore, not setting the `dont_touch_network` attribute on the scan-enable ports or signals is important. The `dont_touch_network` attribute on the scan-enable signal can propagate into functional logic paths, which prevents optimization of these paths and can lead to QoR degradation.

If the `dont_touch_network` attribute is found on the scan-enable signal in your design and shift registers have been identified where extra logic was inserted on the scan-enable signal, the following warning message is stored in the log file:

```
Warning: dont_touch_network attribute on scan-enable port or signal '%s' can result in QoR degradation after scan insertion. (TEST-2040).
```

On scan-enable ports, you can use the `set_case_analysis` command to disable timing optimization and the `set_ideal_network` command to disable design rule fixing. However, if you need to use the `dont_touch_network` attribute, use the `dont_touch_network -no_propagate` command instead. By using this command, you can avoid the propagation of the `dont_touch` attribute into the functional logic.

Shift-register identification is available only in DC Ultra with the `compile_ultra -scan` command. When the `compile_seqmap_identify_shift_registers` variable is set to its default, which is `true`, shift register identification is enabled.

The `compile_seqmap_identify_shift_registers_with_synchronous_logic` variable controls the ability to identify shift registers with synchronous logic between the registers. The default is `false`. The default setting preserves multibit registers better between the `compile` and `insert_dft` steps, which can provide additional area savings. However, you can enable the identification of shift registers with synchronous logic by setting the `compile_seqmap_identify_shift_registers_with_synchronous_logic` to `true` if it provides an area benefit for your particular design.

Note that setting the `compile_seqmap_identify_shift_registers` variable to `false` disables all forms of shift-register identification, including shift registers that contain synchronous logic between the registers.

If you want to exclude specific registers from being identified as shift registers, apply the `dont_use_for_shift_registers` attribute on the registers that should be excluded. For example,

```
prompt> set_attribute [get_cells regs_to_exclude] dont_use_for_shift_registers true \
           -type boolean
```

For best results, you should write out the design in .ddc format to preserve the information about the shift registers that are identified. When you write out the design in .ddc format, the `compile -incremental -scan` command or DFT Compiler can recognize already identified shift registers from a previous compile using the information stored in the .ddc binary file.

If your flow requires you to work with an ASCII netlist rather than a .ddc file, you must use the `set_scan_state` command when reading the netlist back into Design Compiler. The `set_scan_state` command updates the design to indicate that a test-ready compile has already been performed. The `set_scan_state` command also searches for and re-identifies shift registers in the design.

The following example reads in an ASCII design and uses the `set_scan_state` command to set the scan state status:

```
read_verilog mapped_design.v
current_design top
link
set_scan_state test_ready
```

DC Ultra also supports the identification of shift registers with synchronous logic during the `set_scan_state` command run in the ASCII netlist flow. To enable this capability, you must set the following variable to `true` before you run the `set_scan_state` command:

```
set_app_var compile_seqmap_identify_shift_registers_with_synchronous_logic_ascii true
```

The `compile_seqmap_identify_shift_registers_with_synchronous_logic_ascii` variable is set to `false` by default. When the variable is set to `true`, the `set_scan_state` command performs full identification of all types of shift registers in the design. As a result, the set of newly identified shift registers might be different from the original set of shift registers in the design.

Both of the following default variable settings should be set in all Design Compiler sessions to enable this flow:

```
set_app_var compile_seqmap_identify_shift_registers true
set_app_var compile_seqmap_identify_shift_registers_with_synchronous_logic true
```

When you enable the identification of shift registers with synchronous logic in the ASCII netlist flow, DC Ultra issues the following informational message when running the `set_scan_state` command:

```
Information: Performing full identification of complex shift registers.
(TEST-1190)
```

The following ASCII netlist flow script example includes shift registers with synchronous logic:

```
# Turn on ASCII flow support for shift registers with synchronous logic
set compile_seqmap_identify_shift_registers_with_synchronous_logic_ascii true
read_verilog design.vg
current_design top
link
read_sdc design.sdc
if {[shell_is_in_topographical_mode]} {
    extract_physical_constraints floorplan.def
}
# Reapply preexisting DFT constraints (for example, set_scan_element false)
set_scan_state test_ready
# Continue with DFT Compiler flow
```

Using Register Replication

Design Compiler can replicate registers to address timing quality of results (QoR), congestion, and fanout issues. This section describes the register replication features that are supported both in Design Compiler wire load mode and topographical mode. For information about additional register replication features that are supported only in topographical mode, see [Additional Register Replication Features in Topographical Mode](#).

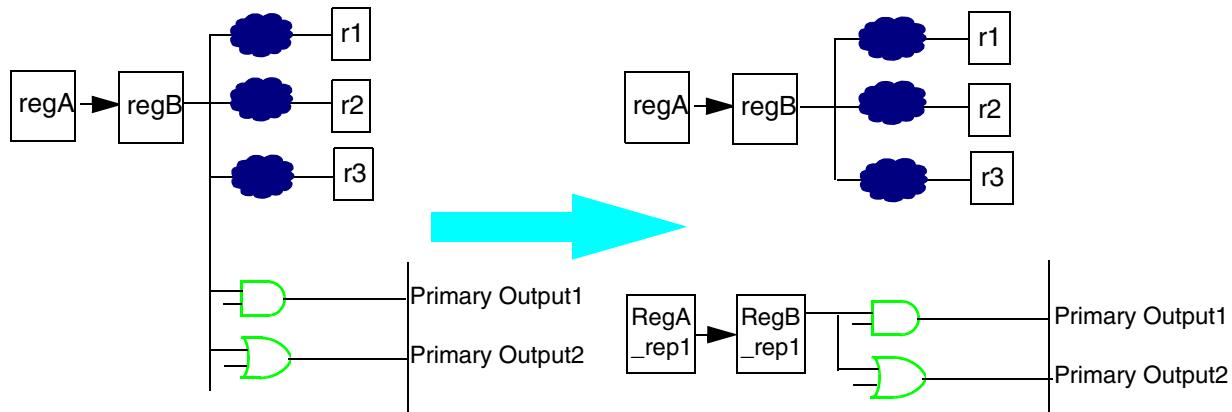
To enable register replication in wire load or topographical mode, use the `set_register_replication` command. The command sets the `register_replication` attribute on the specified sequential cells, allowing register replication on the objects. The load of the original replicated register is evenly distributed among the new replicated registers.

You can set the `register_replication` attribute only on registers and not on the design or current design. After the register replication feature is enabled, the `compile_ultra` or `compile` command automatically invokes the `set_register_replication` command to replicate registers during optimization. Register replication is supported in both full and incremental compilation.

[Figure 15-15](#) shows how the `compile_ultra` command replicates the `regA` register after replicating the `regB` register when you use the following commands, because the fanout of `regA` becomes more than one after replicating the `regB` register.

```
dc_shell-topo> set_register_replication -num_copies 2 regA
dc_shell-topo> set_register_replication -num_copies 2 regB
```

Figure 15-15 Replicating Registers For the Transitive Fanout



To specify the value to replicate the registers n times, where n is greater than or equal to 2, use the `-num_copies` option. To specify the maximum fanout value of the `register_replication` attribute, use the `-max_fanout` option. You can either set a `max_fanout` limit on the target register or specify the number of replications with the `num_copies` option.

When you use the `-num_copies` option, the number of register copies is determined by whichever is smaller: the number specified by the `-num_copies` option or the fanout of the register at the time register replication occurs. If you specify a `-num_copies` value of 10 and the register has a fanout of 6, the register is only replicated 5 times for a total of 6 registers, one for each fanout.

The tool implements the fanout or `-num_copies` value that you set by replicating registers, not by inserting buffers.

You can replicate post-compile mapped logic containing high fanout within a few logic levels before or after the register by using the `-include_fanin_logic` and `-include_fanout_logic` options with the `set_register_replication` command. You can specify any startpoint or endpoint in the fanin and fanout of the register.

The following example replicates a path from the `A_reg` register to a `U1` cell that is in the fanout of `A_reg`:

```
prompt> set_register_replication -num_copies 2 A_reg -include_fanout_logic U1
```

Note the following when you enable the register replication capability:

- If the fanout of the register is 17 and you specified `-fanout 8` or `-num_copies 3`, the tool replicates the register three times with fanout 6, 6, and 5. That is, it evenly distributes the fanouts to each register.
- If boundary optimization is disabled, Design Compiler does not replicate registers across hierarchical boundaries.
- If you use both the `set_max_fanout` command and the `set_register_replication -max_fanout` command, the `set_register_replication` command has a higher priority.
- If the `-max_fanout` and `-num_copies` options are applied together, the `-num_copies` option has the higher priority and the tool reports a warning message that the `-max_fanout` option will be ignored.

[Example 15-4](#) shows how to enable register replication on registers.

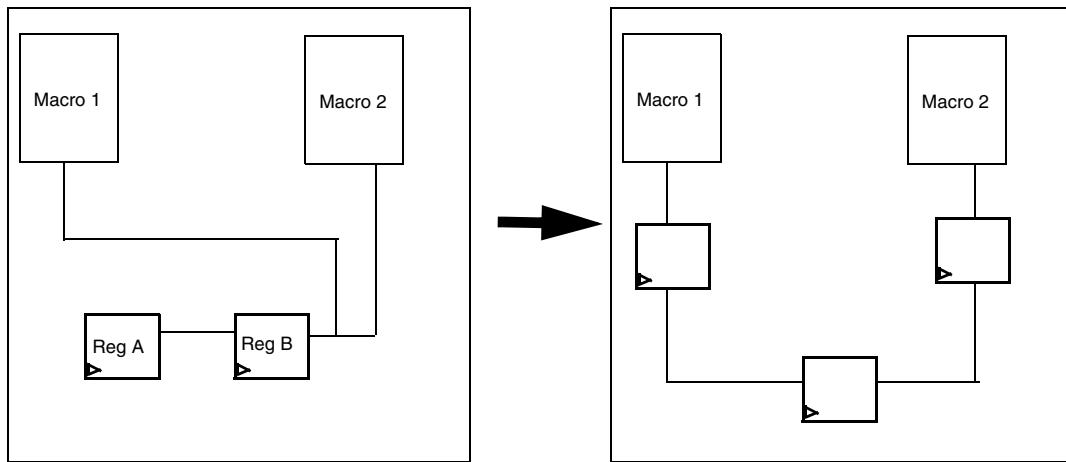
Example 15-4 Enabling Register Replication

```
prompt> set_register_replication -max_fanout 10 [get_cells -h * \
    -f "@ref_name==*regA*"]
prompt> set_register_replication -num_copies 10 [get_cells -h * \
    -f "@ref_name==*regA*"]
```

To specify the style used in naming replicated registers, use the `register_replication_naming_style` variable. The default format is `<%s_rep%d>`. For example, if the original register is named `PLL`, by default, the replicated registers would be named `PLL_rep1`, `PLL_rep2`, `PLL_rep3`, and so forth.

Figure 15-16 shows a design in which register replication has been enabled. The figure shows a chip that contains two macros. Based on the speed and distance across the logic area of the chip, register B cannot be placed at any location so that timing is met. With register replication enabled, the tool can replicate register B so that timing is met.

Figure 15-16 Register Replication Example



Register replication is not performed on the following:

- Retimed logic
- Multibit registers
- Scan-stitched designs

Additional Register Replication Features in Topographical Mode

In topographical mode, Design Compiler provides the following capabilities:

- Automatic register replication

Register replication is automatically enabled to replicate timing-critical registers in the design. This type of replication takes placement into consideration and can help reduce congestion. It is enabled by both the `compile_ultra` and `compile_ultra-incremental` commands when you use the following flows:

- The physical guidance flow using the `-spg` option
- The topographical flow with the `compile_register_replication` variable set to true

When this variable is set to `true`, which is the default when you use the `-spg` option, the `compile_ultra` command tries to identify registers in the current design that can be split to balance the loads for better QoR.

- Replication of registers with the `-driven_by_original_register` option

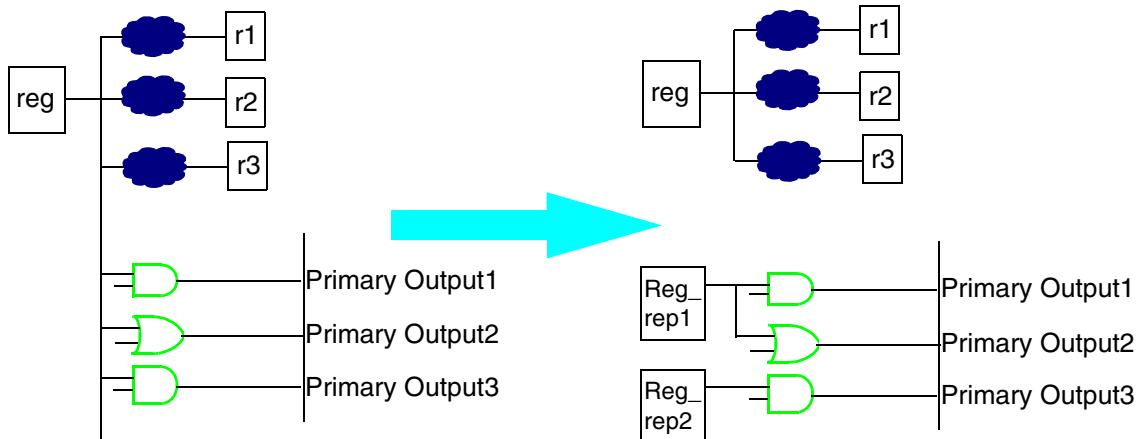
To specify endpoint cells (registers or primary output ports) that are driven by the original register for replicating registers as shown in [Figure 15-17](#), use the `-driven_by_original_register` option. For example

```
dc_shell-topo> set_register_replication -num_copies 3 reg \
    -driven_by_original_register {r1 r2 r3}
dc_shell-topo> compile_ultra
```

After manual register replication,

- The original register drives the registers (r1, r2, and r3).
- The replicated register drives other fanout logic.

Figure 15-17 Replicating Registers By Using the `-driven_by_original_register` Option



- Replication of registers whose clock pins are connected to a `dont_touch` net

This behavior is on by default and is the same behavior in both manual and automatic register replication.

- Replication of registers with a `size_only` attribute

Registers with a `size_only` attribute are considered candidates for automatic register replication by default. Replication of registers with a `size_only` attribute is also supported when you use the `set_register_replication` command. The `size_only` attributes are preserved in the `.ddc` file.

You can disable register replication for registers with the `size_only` attribute by setting the `compile_register_replication_do_size_only` variable to `false`. Similarly, you can disable register replication for registers without the `size_only` attribute by setting

the `size_only` attribute on the register and then setting the `compile_register_replication_do_size_only` variable to `false`.

- Register replication across hierarchies

When the `compile_register_replication_across_hierarchy` variable is set to `true` in topographical mode, Design Compiler enables register replication across hierarchies. In addition, it creates new ports on instances of subdesigns while performing register replication if it is necessary to improve the timing of the design. Register replication across hierarchies might change the interfaces among subdesign instances. If the interfaces need to be preserved, this variable should be set to `false` (the default).

16

Adaptive Retiming

Adaptive retiming allows the tool to move registers and latches during optimization to improve timing. Design Compiler in wire load mode and topographical mode automatically performs adaptive retiming when you use the `compile_ultra` command with the `-retime` option. Before you read further, see [Optimization Flow](#) to understand how register retiming fits into the overall compile flow.

Adaptive retiming is intended for use in optimizing general designs; it does not replace the pipelined-logic retiming engine available with the `optimize_registers` and `set_optimize_registers` commands. For information about pipelined-logic retiming, see [Pipelined-Logic Retiming](#).

To learn how to use adaptive retiming, see the following topics:

- [Comparing Adaptive Retiming With Pipelined-Logic Retiming](#)
- [Adaptive Retiming Examples](#)
- [Performing Adaptive Retiming](#)
- [Controlling Adaptive Retiming](#)
- [Reporting the dont_retime Attribute](#)
- [Removing the dont_retime Attribute](#)
- [Verifying Retimed Designs](#)

Comparing Adaptive Retiming With Pipelined-Logic Retiming

Adaptive retiming moves registers and latches to improve worst negative slack (WNS). For datapath designs, you should still use either the `optimize_registers` command or the `set_optimize_registers` command followed by the `compile_ultra` command.

You can use both adaptive retiming and pipelined-logic retiming if you use the `set_optimize_registers` command on the pipelined portions of the design prior to running `compile_ultra -retime`, as shown in the following example:

```
set_optimize_registers [get_designs pipelined_ALU]
compile_ultra -retime
```

The commands in the previous example apply pipeline retiming on the pipelined portions of the design and adaptive retiming on the remainder of the design.

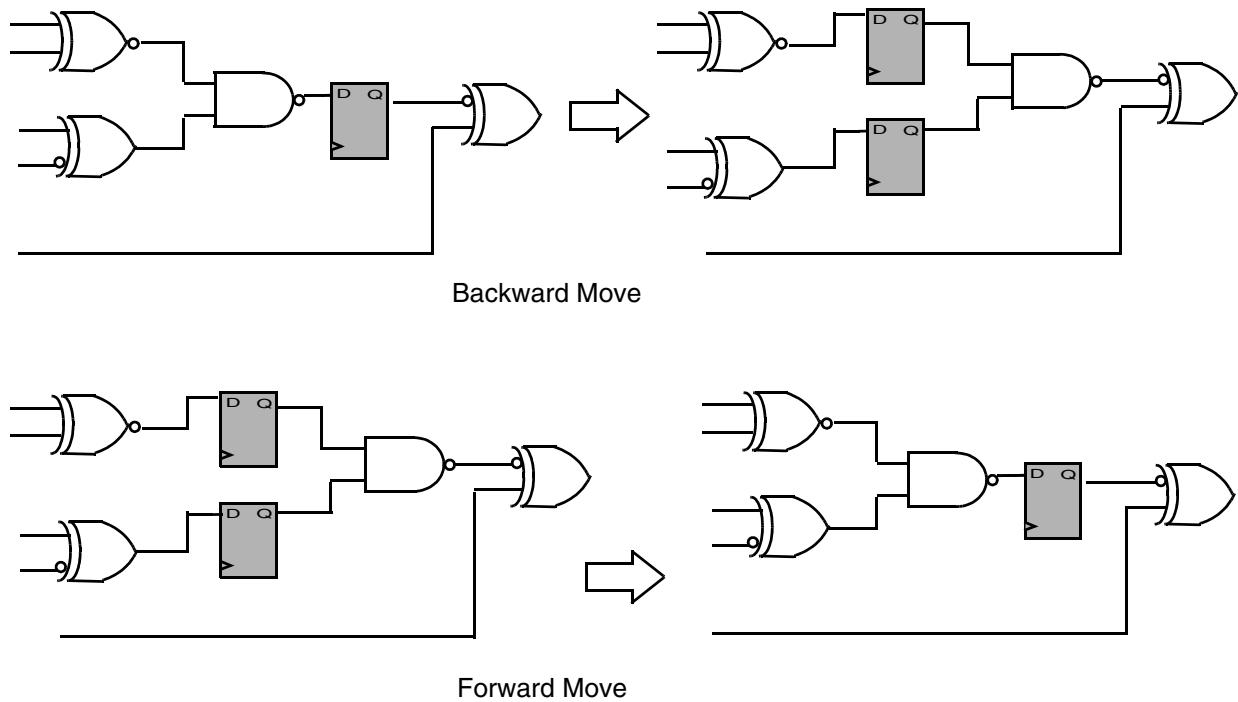
See Also

- [Pipelined-Logic Retiming](#)
-

Adaptive Retiming Examples

When you describe circuits prior to logic synthesis, it is usually time-consuming and difficult to find the optimal register locations and code them into the HDL description. With retiming, the locations of the registers and latches in a sequential design can be automatically adjusted to equalize as nearly as possible the delays of the stages. This capability is particularly useful when some stages of a design exceed the timing goal while other stages fall short. If no path exceeds the timing goal, adaptive retiming can be used to reduce the number of registers, where possible.

During retiming, registers are moved forward or backward through the combinational logic of a design as shown in [Figure 16-1](#).

Figure 16-1 Adaptive Retiming

Design Compiler can improve worst negative slack by performing the following tasks during adaptive retiming:

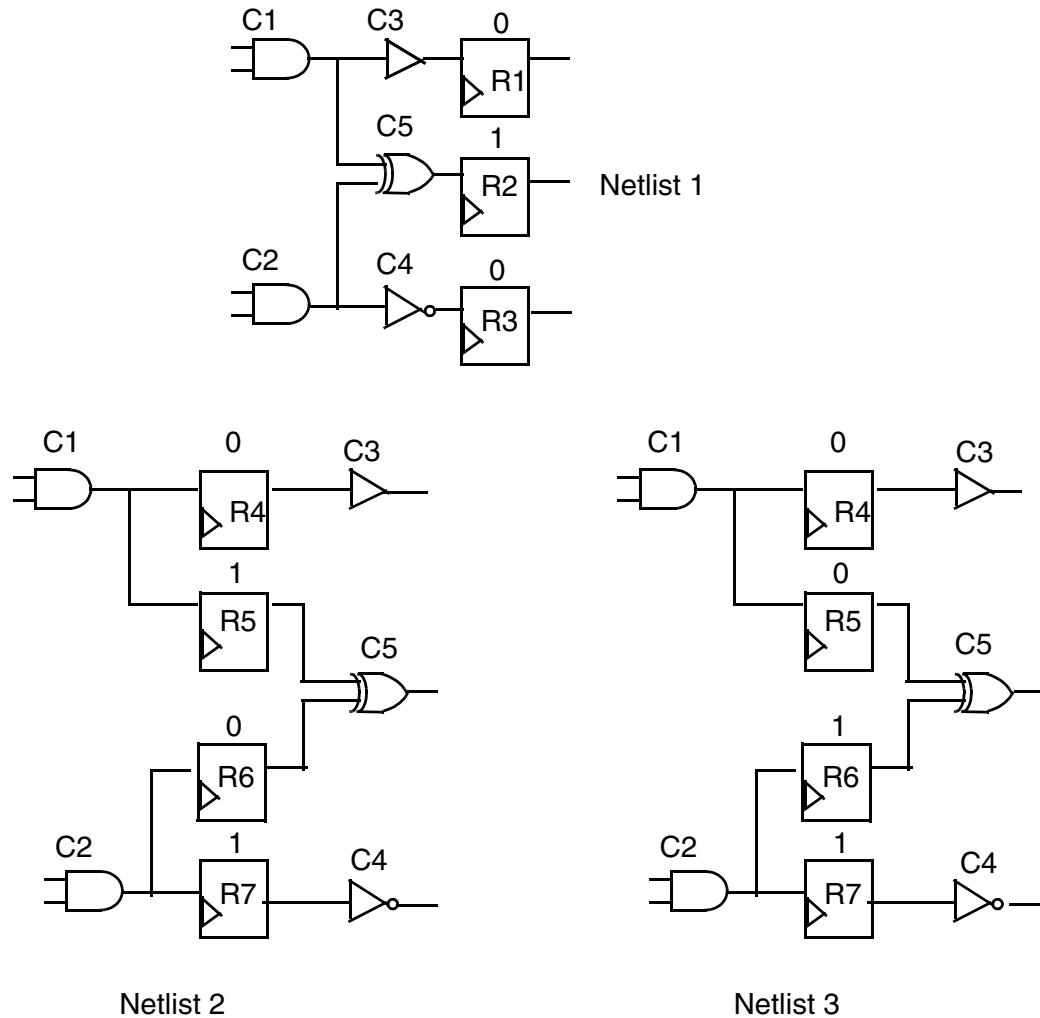
- Retiming registers with conflicting reset requirements
- Retiming registers with extra synchronous input pins (tied high or low)
- Decomposing registers without SEQGEN correspondence

Design Compiler can also improve area by performing the following tasks:

- Allowing forward moves on non-critical registers
- Merging registers with equal and opposite next states

[Figure 16-2](#) shows an example of how Design Compiler handles registers with conflicting reset requirements.

Figure 16-2 Adaptive Retiming for Registers with Conflicting Reset Values

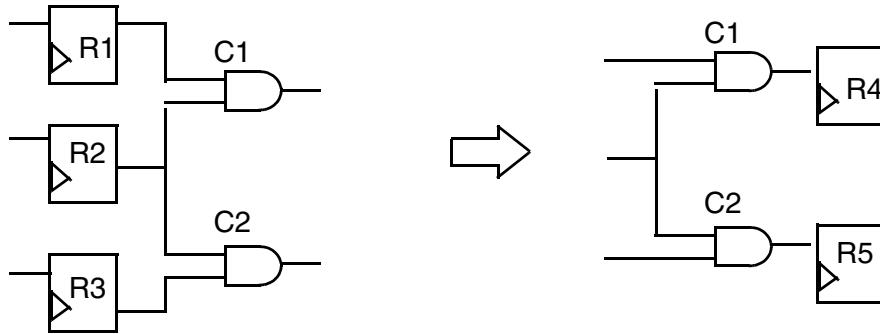


The values (0 or 1) above the registers in the netlists indicate their reset values when a common reset or preset signal (not shown) becomes active. Assume that all three registers R1, R2, R3 in the first netlist move backward across the driver cells of their input nets. The second netlist is a possible result of the moves. Registers R4 and R7 have unique reset or preset values; however, R5 and R6 can have either preset (1) for R5 and reset (0) for R6 as in the second netlist or the reverse as in the third netlist.

Design Compiler can retime registers with these conflicting reset requirements. The two input XOR gate has two possible input assignments resulting in an output value of 1. In the second netlist, conflicting reset values can prevent further backward moves. The third netlist, however, allows such backward moves if they are required.

Figure 16-3 shows how Design Compiler can execute forward local retiming moves on non-critical registers to improve area by decreasing the number of registers.

Figure 16-3 Forward Moves for Non-Critical Registers



Design Compiler can move registers with common timing exceptions, including `max_delay`, `min_delay`, `multicycle_path`, `false_path`, and `group_path`, as long as all the registers being moved have the same exceptions. After executing a move involving registers with exceptions, the new registers inherit all the exceptions from the original registers.

Two registers have the same exceptions if they appear in the same point-to-point timing exception setting commands, such as `set_max_delay`, `set_min_delay`, `set_multicycle_path`, `set_false_path`, and `group_path`.

In the following example, registers r0 and r1 have the same exceptions.

```
set_max_delay 10 -to [list r0/D r1/D]
```

Alternatively, if this `set_max_delay` command is split into two `set_max_delay` commands, both with the same delay value, the two registers still have the same exceptions:

```
set_max_delay 10 -to r0/D
set_max_delay 10 -to r1/D
```

However, if the two `set_max_delay` commands have different delay values, the two registers have different exceptions, and they cannot be moved together:

```
set_max_delay 10 -to r0/D
set_max_delay 15 -to r1/D
```

By default this feature is disabled: to enable this feature, set the `compile_retime_exception_registers` variable to true.

Performing Adaptive Retiming

You use the `-retime` option of the `compile_ultra` command to perform adaptive retiming and to improve the delay during optimization.

Adaptive retiming honors attributes, such as `dont_touch`, `size_only`, and `dont_retime`; retiming is prevented if these attributes are set. Registers that are moved as a result of adaptive retiming are renamed with a prefix of R and a numbered suffix (R_xxx); you cannot match these retimed registers to the original registers.

Adaptive retiming supports all `compile_ultra` options except the following:

- `-top`
- `-only_design_rule`

Controlling Adaptive Retiming

You can use the `set_dont_retime` command to include or exclude designs or cells from being retimed. For example, the following command specifies that the design a1 should not be retimed:

```
set_dont_retime [get_designs a1] true
```

Setting the `dont_retime` attribute on a hierarchical cell implies that the attribute is also set on all cells below it. However, you can override the attribute at a lower-level of the hierarchy by explicitly setting the `dont_retime` attribute at that level. For example, the following command specifies that the cells `z1_reg` and `z2_reg` should be retimed:

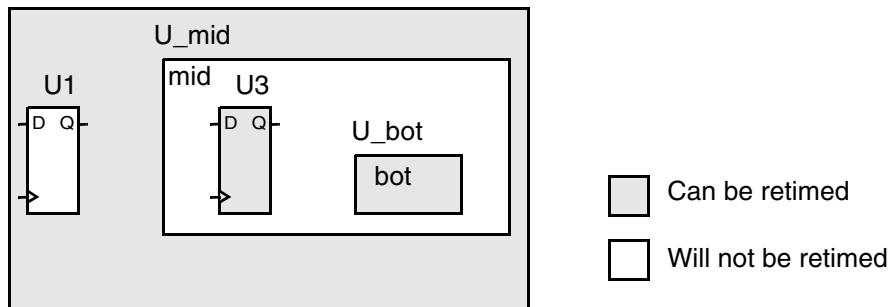
```
set_dont_retime [get_cells {z1_reg z2_reg}] false
```

If you use the `set_dont_retime` command without specifying either true or false, a value of true is assumed.

The following sequence of commands will create the scenario shown in [Figure 16-4](#):

```
set_dont_retime [get_cells U1]
set_dont_retime [get_designs mid] true
set_dont_retime [get_cells U_mid/U3] false
set_dont_retime [get_cells U_mid/U_bot] false
compile_ultra -retime
```

Figure 16-4 Design With Both Retimable and Nonretimable Cells



Note:

The `dont_retime` attribute on a child cell has priority over the attribute set on any ancestor cell or design.

You can also use the `set_dont_retime` command to prevent retiming on specific cells or designs when you use the `optimize_registers` command. For more information, see [Pipelined-Logic Retiming](#).

Reporting the `dont_retime` Attribute

To check which cells or designs have the `dont_retime` attributes, use the `report_attribute` or `get_attribute` command.

Removing the `dont_retime` Attribute

To remove the `dont_retime` attribute, use the `remove_attribute` command.

Verifying Retimed Designs

When Design Compiler performs any retiming, an automated setup file is necessary for verification with the Formality tool. The automated setup file includes retiming optimization information that helps Formality verify retimed designs. To remind you that Formality requires an automated setup file, Design Compiler displays the following message when retiming is performed:

Information: Retiming is enabled. SVF file must be used for formal verification. (OPT-1210)

If you run multiple `compile_ultra -retime` commands, you must run multiple formal verifications, each one verifying only the changes made by the most recent `compile_ultra -retime` command.

The following example shows a Design Compiler script that runs the `compile_ultra -retime` command followed by an incremental compile with the `-retime` option. In the script, the verification is run by comparing the RTL to the `compile_1.ddc` netlist using the `compile_1.svf` verification setup file and comparing the `compile_1.ddc` netlist to the `compile_2.ddc` netlist using the `compile_2.svf` verification setup file.

```
set_svf compile_1.svf
read_verilog rtl.v
compile_ultra -retime
write_file -hierarchy -output compile_1.ddc
set_svf compile_2.svf
compile_ultra -incremental -retime
write_file -hierarchy -output compile_2.ddc
```

See Also

- [Verifying Functional Equivalence](#)
Provides information about design verification
- [The *Formality User Guide*](#)

17

Pipelined-Logic Retiming

Register retiming is a sequential optimization technique that DC Ultra uses to move registers through the combinational logic gates of a design to optimize timing and area. In this context, the term *register* refers to both edge-triggered registers and level-sensitive latches unless stated otherwise. Both types of sequential cells can be retimed.

You enable pipelined-logic retiming by using the `optimize_registers` command or by using the `set_optimize_registers` command followed by the `compile_ultra` command.

To learn how to retime circuits using pipelined-logic retiming, see the following topics:

- [Pipelined-Logic Retiming Overview](#)
- [Register Retiming Concepts](#)
- [Retiming the Design](#)
- [Analyzing Retiming Results](#)
- [Verifying Retimed Designs](#)

Pipelined-Logic Retiming Overview

DC Ultra supports the following methods for retiming registers:

- Adaptive retiming

You enable this retiming method by using the `-retime` option with the `compile_ultra` command. In this case, DC Ultra locally adjusts the register location.

This method is not covered in this chapter. For more information, see the [Adaptive Retiming](#) chapter.

- Pipelined-logic retiming

You enable this retiming method by using either the `optimize_registers` command or the `set_optimize_registers` command followed by the `compile_ultra` command.

This method is designed to balance the multiple stages of pipelined registers into combinational logic. This is particularly useful to pipeline datapath blocks.

When you describe circuits at the RTL level before logic synthesis, it is usually very difficult and time consuming to find the optimal register locations and code them into the HDL description. With register retiming, the locations of the flip-flops in a sequential design can be automatically adjusted to equalize as nearly as possible the delays of the stages.

The retiming techniques that the `optimize_registers` and `set_optimize_registers` commands use are particularly useful to pipeline datapath blocks. You first specify the desired number of pipeline stages and place all the pipeline registers at the inputs or outputs of the combinational logic in your RTL. During retiming, the tool moves these pipeline registers into the combinational logic, balances the delay of each pipeline stage, and archives the optimal timing and area results based on the given clock period.

Register retiming leaves the behavior of the circuit at the primary inputs and primary outputs unchanged (unless you add pipeline stages or choose special options that do not preserve the reset state of the design). Therefore, you do not need to change any simulation test benches developed for the original RTL design.

Retiming does, however, change the location, contents, and names of registers in the design. A verification strategy that uses internal register inputs and outputs as reference points will no longer work. Retiming can also change the function of hierarchical cells inside a design and add clock, clear, set, and enable pins to the interfaces of the hierarchical cells.

The following topics describe the commands that enable pipelined-logic retiming and an example of a circuit before and after retiming:

- [Pipelined-Logic Retiming Commands](#)
- [Register Retiming Example](#)

Pipelined-Logic Retiming Commands

You can use either of the following command sequences to enable pipelined-logic retiming:

- Using the `optimize_registers` command

This method performs retiming on a mapped netlist. You must compile the design using the `compile_ultra` command before enabling retiming with the `optimize_registers` command.

This method is also known as two-pass retiming because retiming is performed in two steps: The first step moves and minimizes the sequential cells, and the second step performs an incremental compile, which adjusts the design to the changed fanout structure.

- Using the `set_optimize_registers` command followed by the `compile_ultra` command

This method performs retiming during synthesis when you run the `compile_ultra` command after reading your RTL. You must run the `set_optimize_registers` command before running `compile_ultra`. The `set_optimize_registers` command sets the `optimize_registers` attribute on the specified design or on the current design so that the attribute is automatically invoked during compile and the design is retimed during optimization.

This method is also known as one-pass retiming because retiming is performed in one step by the `compile_ultra` command.

Register Retiming Example

During retiming, registers are moved forward or backward through the combinational logic of a design. [Figure 17-1](#) and [Figure 17-2](#) illustrate an example of delay reduction through backward retiming of a register.

Figure 17-1 Circuit Before Retiming

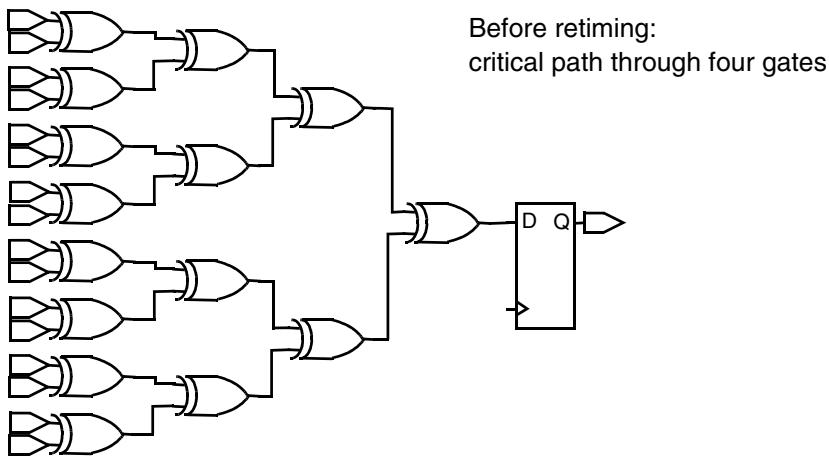
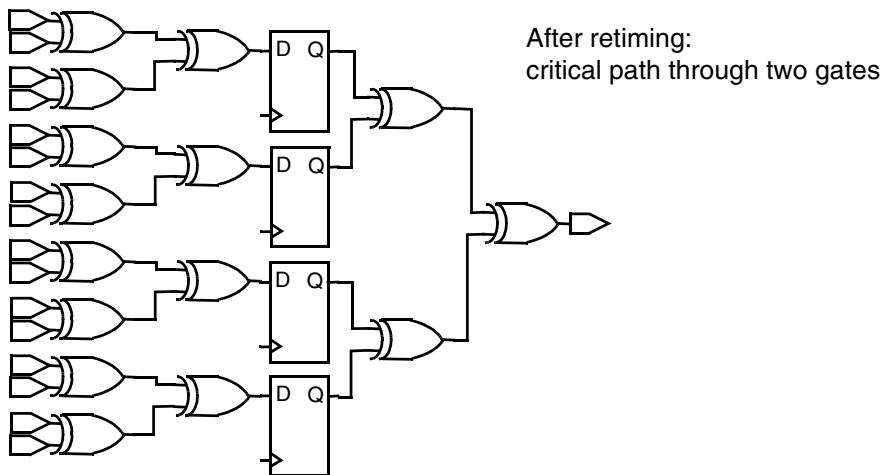


Figure 17-2 Circuit After Retiming



In this example, before register retiming there are four levels of combinational logic and only one register at the endpoint of the critical path. After retiming, the register, which has been replaced by four registers, has been moved back through two levels of logic, and the critical path now consists of two stages. The critical path delay in each stage is less than the critical path delay in the initial single stage design. As in this example, delay reduction through retiming often leads to an increase in the number of registers in the design, but usually this increase is small.

Register Retiming Concepts

The following topics describe the fundamentals of register retiming to help you make the best use of the Design Compiler register retiming capabilities:

- [Basic Definitions and Concepts](#)
 - [Forward Retiming](#)
 - [Backward Retiming](#)
 - [Register Transformation Methods](#)
 - [Reset State Justification](#)
-

Basic Definitions and Concepts

To understand how register retiming works, you must first understand certain basic definitions and concepts. In particular, you must understand what sequential generic elements (SEQGENs), control nets, and register classes are. These are important because during retiming, mapped registers are temporarily replaced by SEQGENs according to their classifications as determined by their control nets.

Note:

The following terms are not defined in this topic because they are used the same way as in other Synopsys documentation: design, cell, leaf cell, hierarchical cell, combinational cell, and sequential cell.

Flip-Flops and Registers

Flip-flop, register, synchronous register, asynchronous register, and latch are familiar terms; however, with respect to register retiming, these terms have the following specialized usage:

- A *flip-flop* is an element of a technology library (target library) that has, unlike the combinational cells, a state and a distinguished clock input. Flip-flops can be edge triggered or level sensitive.
- A *register* is technically an instance of an edge-triggered flip-flop in the design. But for the purposes of register retiming, because both edge-triggered flip-flops and level-sensitive latches can be retimed, the term *register* refers to both types of sequential devices unless stated otherwise.
- A *synchronous register* is a register that can change its state only at the active edge of the clock signal.

- An *asynchronous register* is a register that, in addition to changing its state on a clock edge, can also change its state according to the control levels of asynchronous signals, which are independent of its clock signal.
- A *latch* is an instance of a level-sensitive flip-flop in the design. Register retiming supports designs with latches and retimes them instead of the registers if the `-latch` option is used.

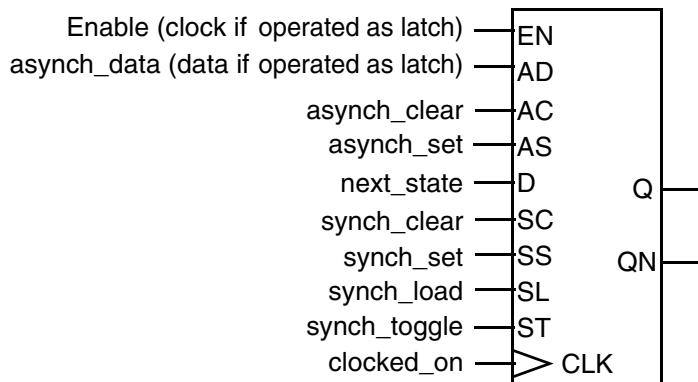
The essential point here to understand is that flip-flops are technology library elements, while registers and latches are their design instances.

SEQGENs

A SEQGEN is a generic sequential element that is used by Synopsys tools to represent registers and latches in a design. SEQGENs are created during elaboration and are usually mapped to flip-flops during compilation. Because mapped flip-flops are temporarily replaced by SEQGENs during register retiming, it is important that you understand the basic functionality of these elements.

[Figure 17-3](#) shows the pins that are used when the SEQGEN cell describes a synchronous or asynchronous register. (Additional pins occur when the SEQGEN is operated as a latch.)

Figure 17-3 Relevant Pins of the SEQGEN Cell



In register retiming, the operation of SEQGEN cells is as follows:

- The synchronous toggle pin (ST) has an inactive value of 0. Therefore, the SEQGEN cell is retimed only if the ST pin is connected to a constant net with value 0.
- The clock (CLK) pin is always connected to the clock net of the design.
- The synchronous state changes occur at the rising edge of the clock signal.
- If set to 1, the synchronous load (SL) pin enables the next-state D input to become the next state. The SL pin should be tied to a constant 1 net when unused.

- The synchronous clear (SC) pin sets the state to 0 if active. This pin preempts the SL input and, to be inactive, must be tied to a constant 0 net.
- The synchronous set (SS) pin sets the state to 1 if active. This pin preempts the SL input and, to be inactive, must be tied to a constant 0 net.
- If both SC and SS are active, the constant set as an attribute on the particular SEQGEN instance becomes the new state.
- The asynchronous inputs AC and AS override all settings of the synchronous inputs; these pins change the state and output of a SEQGEN instance, independent of the clock input.
- The AC input sets the Q output to 0 if active; the AS input sets the Q output to 1 if active. Both inputs must be 0 to be inactive.
- The EN input replaces the CLK clock pin if the SEQGEN cell operates as a level-sensitive latch.
- The AD pin input replaces the D pin input if the SEQGEN cell operates as a level-sensitive latch.

Control Nets

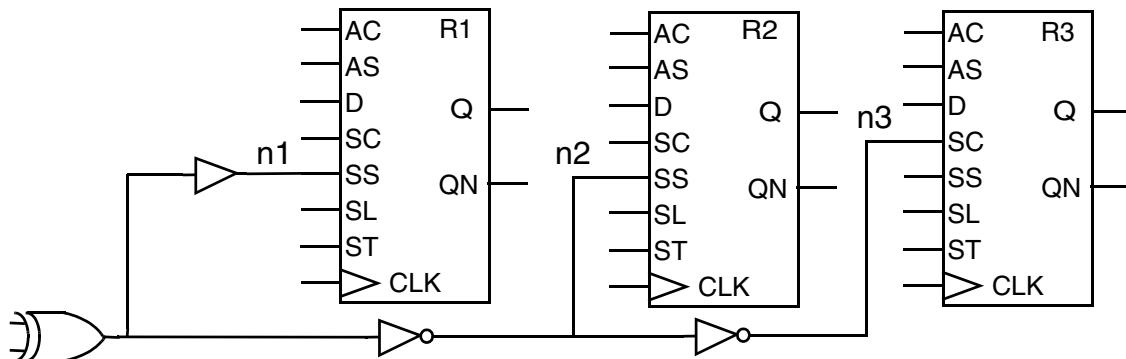
A control net is a net connected to one of the SL, SC, SS, AC, or AS pins of a SEQGEN instance. The equivalence of control nets plays a crucial role in the movement of registers during retiming.

By definition, a set of control nets are equivalent if they meet either of the following conditions:

- All the nets can be reached from a common source, and between this common source and the SEQGEN instances, all the nets have an odd number of inverters or all the nets have an even number of inverters. (A net with no inverters is regarded as having an even number of inverters.)
- The control nets are constant and have the same constant values (0 or 1).

Thus, two nonconstant nets with a common source, one that includes an odd number of inverters and the other an even number of inverters, are not equivalent nets. Note that any number of buffers is allowed between the common source and the SEQGEN pin of an equivalent net.

[Figure 17-4](#) shows an example of equivalent and nonequivalent nets. Nets n1 and n3, which have even number of inverters (0 and 2), are equivalent, while net n2, which has an odd number of inverters (1), is not equivalent to either of them.

Figure 17-4 Equivalent Control Net Example

Register Classes

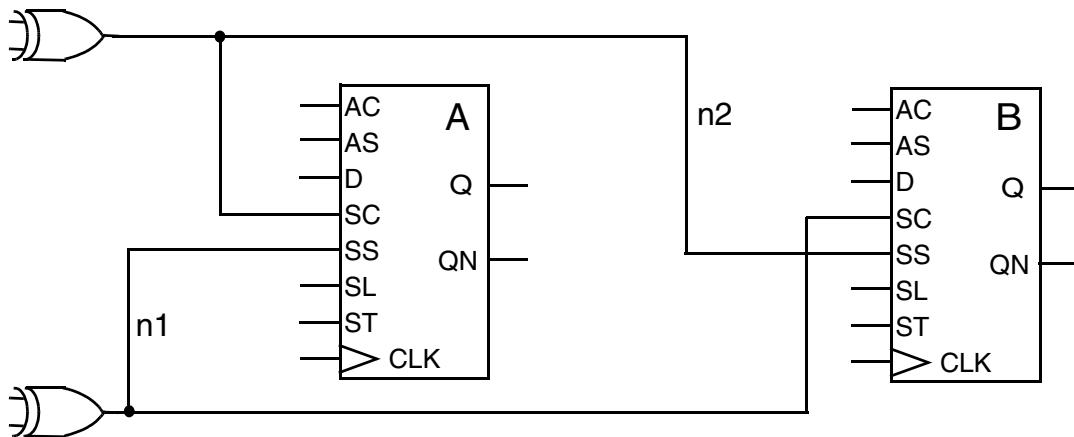
The SEQGEN instances are grouped into register classes according to their connections to control nets. Grouping is necessary because only registers belonging to the same group can be moved together across a combinational gate without violating the circuit logic.

Two SEQGEN instances belong to the same register class if the following conditions are fulfilled:

- Their SL pins are connected to equivalent control nets.
- Their SC and SS pins are connected to equivalent control nets. That is, a given control net can be connected to the corresponding SC or SS pins in the SEQGEN instances or to the SC pin of one SEQGEN instance and to the SS pin of the other.
- Their AC and AS pins are connected to equivalent control nets. The same conditions hold for the asynchronous pins as for the synchronous pins.

[Figure 17-5](#) shows an example in which registers A and B belong to the same class.

Figure 17-5 Swapping of Control Net for Registers in the Same Class



In [Figure 17-4 on page 17-8](#), if all pins not connected to nets n1, n2, and n3 are connected to their inactive constants, registers R1 and R3 belong to the same class, but register R2 belongs to a different class.

When registers belonging to the same class are moved, it is possible to swap their control nets as needed to accomplish the retiming. This swapping capability is true for both synchronous and asynchronous register pins.

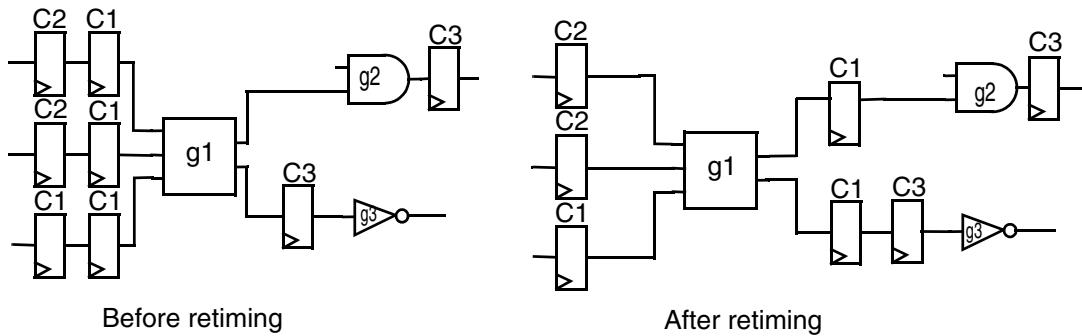
Forward Retiming

To retime forward across a combinational cell, each net in the immediate fanin of the cell must be driven by the Q pin of a register, and all these registers must belong to the same class. After the forward retiming move, the registers in the fanout of the cell belong to the same class as those in the fanin before the move.

[Figure 17-6](#) shows an example of retiming forward across the combinational cell g1.

Note:

In this and the following sections, the explicit control nets for registers are not drawn unless there is a special reason to do so. Register classes are denoted by class names (for example, C1 and C2).

Figure 17-6 Forward Retiming Example

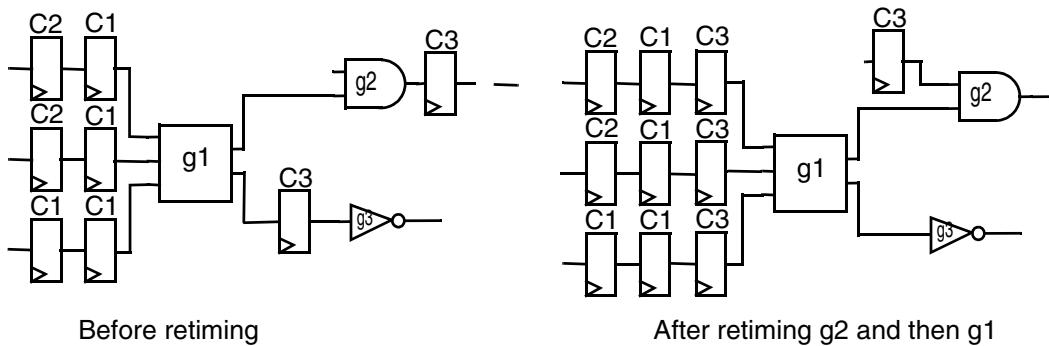
Before the retiming move is executed, all the registers in the immediate fanin of the g1 cell belong to class C1. Notice that after the move the three registers have been replaced by two registers belonging to the same class C1. It is not possible to retime forward the next slice of registers in the fanin of the g1 cell because not all these registers belong to the same class. Also, after the first retiming, it is not possible to retime forward the class C1 register that drives one of the g2 cell inputs because only one input pin of the cell has a register driving it.

If during retiming the maximum number of forward retiming moves across a cell has been performed, the cell has reached its forward retiming boundary limit.

Backward Retiming

Rules similar to the forward retiming rules govern backward retiming across a combinational cell. All nets in the fanout of a combinational cell must fan out to the D pin of the registers, and all these registers must belong to the same class. After the backward retiming move, the registers in the fanin of the cell belong to the same class as those in the fanout before the move.

[Figure 17-7](#) shows how the combinational cell g1 can be retimed backward *after* the cell g2 has been retimed backward. Note that two backward timing moves have been carried out.

Figure 17-7 Backward Retiming Example

If during retiming the maximum possible number of backward retiming moves across a cell has been performed, the cell has reached its backward retiming boundary limit.

Register Transformation Methods

You can choose which register transformation method pipelined-logic retiming uses when it moves registers. Design Compiler supports the following transformation methods:

- [Transforming Synchronous Input Pins Through Combinational Decomposition](#)
- [Multiclass Retiming](#)

Transforming Synchronous Input Pins Through Combinational Decomposition

Combinational decomposition transforms synchronous input pins by

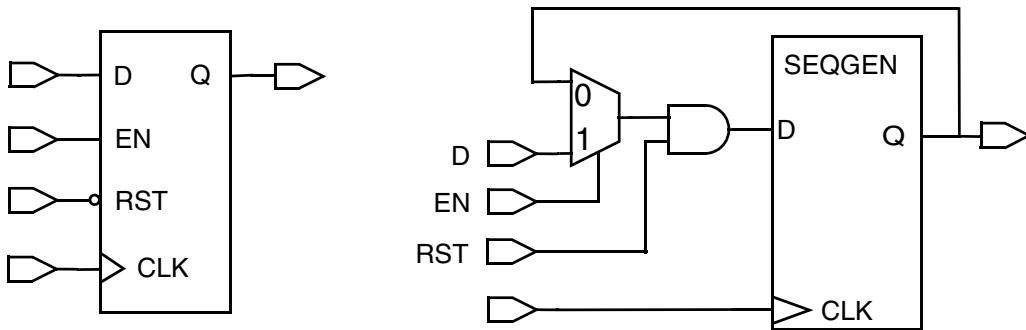
- Using combinational cells to implement the synchronous functionality, and
- Connecting the output of the combinational cells to the D pin of a SEQGEN cell

Note:

Combinational decomposition can be applied to the synchronous pins of asynchronous registers.

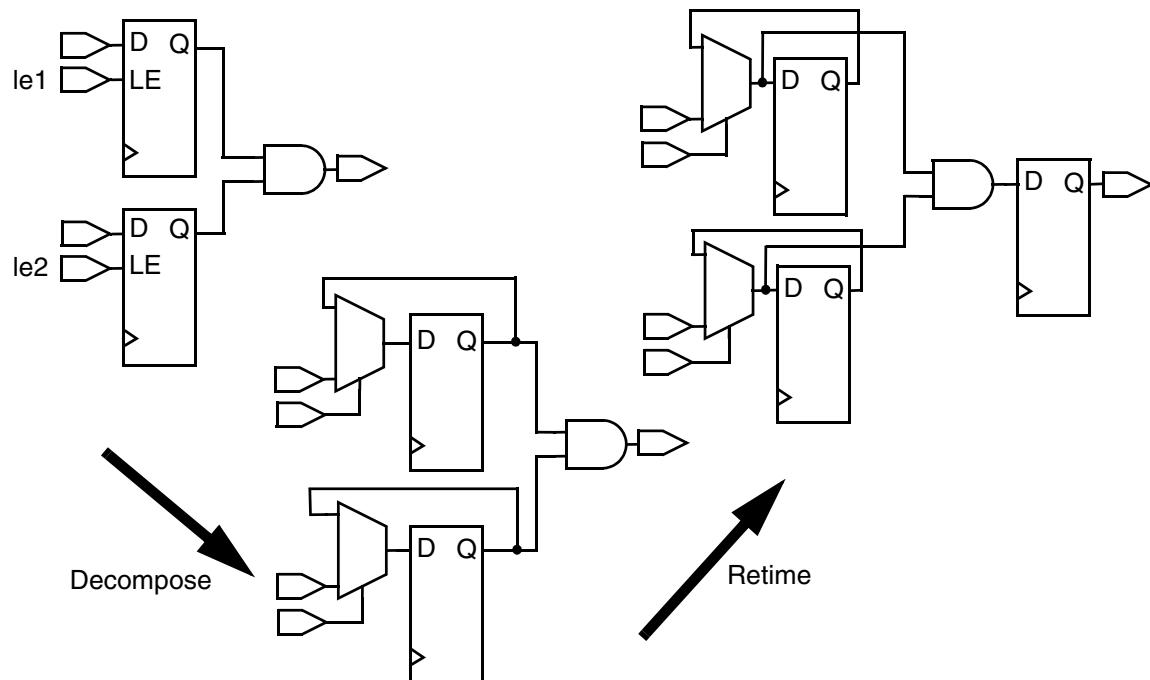
[Figure 17-8](#) shows the combinational decomposition of the register with synchronous clear and enable signals.

Figure 17-8 Transformation by Decomposition Example

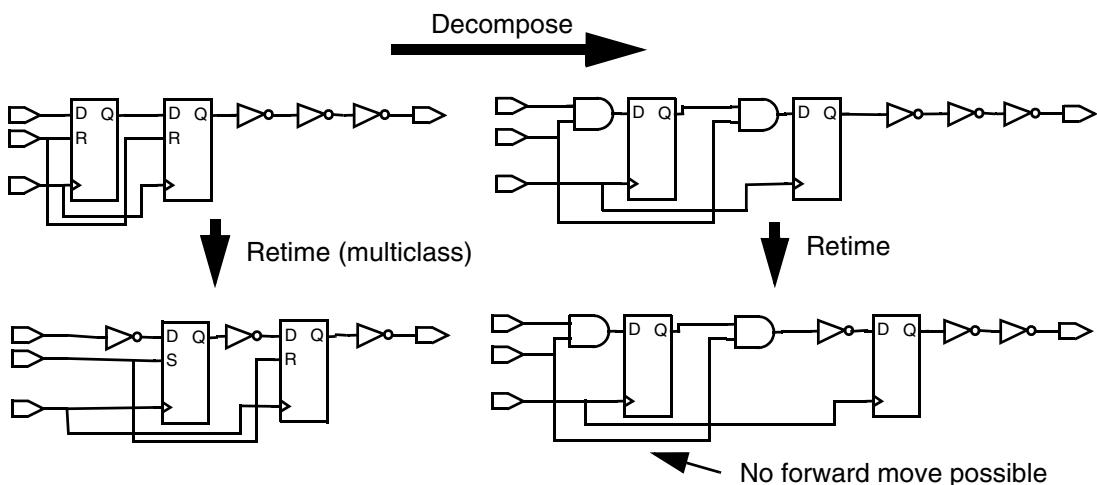


An advantage of combinational decomposition is that all purely synchronous registers belong to the same class after they are transformed to SEQGENs. Consequently, there are no limits to the number of forward or backward moves possible at multiple input or output gates due to registers belonging to different classes. Not using decomposition (multiclass retiming) can lead to register class conflicts, which ultimately limit the number of forward or backward retiming moves possible.

Figure 17-9 shows how registers with *different* enable control nets can be moved forward after decomposition. (To simplify the figure, the clock net is not drawn.) These registers could not be moved after multiclass retiming. Notice, however, that two D flip-flops remain with the fanouts belonging to synchronous combinational logic and cannot be forward retimed; the third flip-flop is free to move by forward retiming.

Figure 17-9 Forward Retiming of Decomposed Cells With Load Enable

Combinational decomposition can also limit the movability of registers. [Figure 17-10](#) shows how decomposition applied to a sequence of two synchronous clear registers leaves the left register without the possibility of a forward move because the newly introduced AND gate does not have a register at its second port. (The right register is forward retimed through an inverter.) Alternatively, multiclass retiming allows both registers to move forward: The right register can be moved across two inverters and the left register across one buffer.

Figure 17-10 Reduced Mobility After Decomposition

See Also

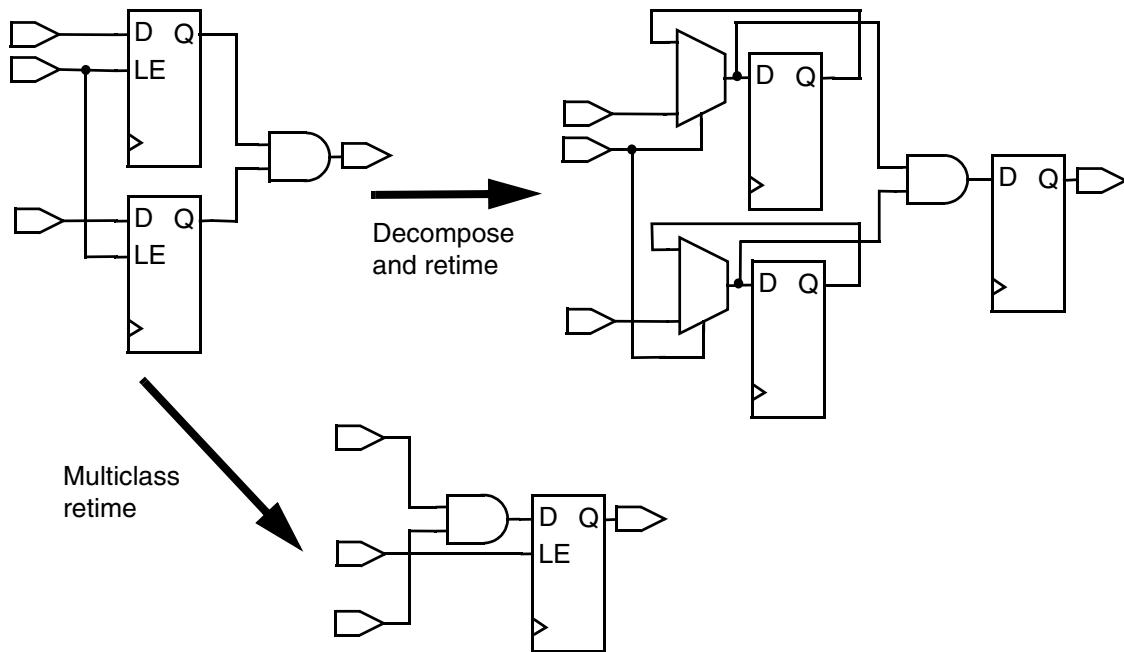
- [Selecting Transformation Options](#)

Describes how to specify a register transformation method

Multiclass Retiming

Multiclass retiming can offer significant area savings compared with retiming after decomposition. A multiclass example is shown in [Figure 17-11](#). The situation is similar to the example in [Figure 17-9](#) except that the two load enable registers belong to the same class and therefore can be moved across the AND gate, leading to a single register. Using decomposition leads to a higher number of registers and additional cells after retiming.

Figure 17-11 Reduced Area Through Multiclass Retiming



See Also

- [Selecting Transformation Options](#)

Describes how to specify a register transformation method

Reset State Justification

When you move the registers in a circuit, it is not sufficient to follow only the rules for retiming of a single gate. In addition, it is usually necessary to preserve an equivalent reset state.

The circuit state is defined by the values of all the registers in the circuit at a given point in time. A circuit and its retimed version have an equivalent state if they produce the same sequences of values at corresponding primary outputs for identical sequences of values at the corresponding primary inputs.

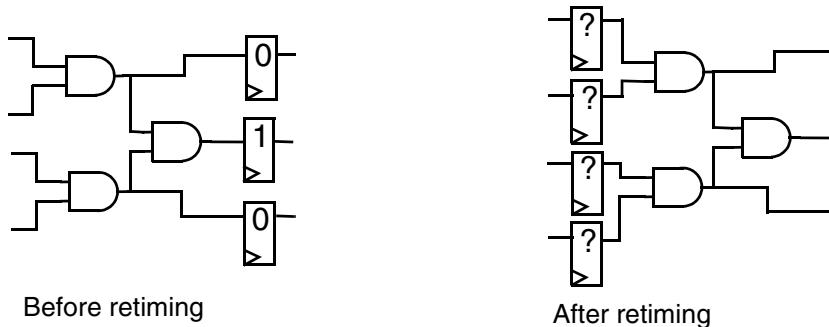
When power is switched on in a circuit, its state is unknown. Depending on the type of circuit, you might need to have an external reset or set input to reset the registers to a known value. This step ensures that the circuit has reproducible behavior after the input becomes active for the first time.

If a circuit is designed this way, by default, register retiming ensures that the reset state of the retimed circuit is equivalent to that of the original circuit, and that the behavior is identical when a finite number of clock cycles has passed after the activation of the reset. If all registers are properly reset, output sequences should match immediately.

However, a typical case where the first few output values might not match is that of a pipelined datapath where the registers do not have any set or clear connections. The maximum duration of the mismatch is the number of stages of the pipeline multiplied by the clock period.

The computation of the equivalent reset state is called *justification*. Justification for registers that have been moved forward across combinational gates is always possible and does not require significant amounts of CPU time. Justification for registers moved backward across combinational gates can be more complicated. [Figure 17-12](#) shows an example of the difficulty with backward justification.

Figure 17-12 Impossible Backward Justification Example



The numbers inside the register symbols are the given reset values. When the registers are moved to the post-retiming positions, it is not possible to find an equivalent state for the

circuit. Register retiming handles this case by finding a position for the registers where an equivalent state is found that is as close to the optimal position as possible.

Backward justification also can cost more CPU time than forward justification. If the circuit to be retimed has a reset but does not need to have an equivalent reset state after retiming, there is a method available that does not perform justification. This method can be applied to pipelined datapaths, but it is not suitable for controllers.

To specify the justification effort level during backward justification, use the `-justification_effort` option with the `optimize_registers` or `set_optimize_registers` command. The option can take one of the following values: `low`, `high`, or `medium`.

Retiming the Design

The following topics describes the register retiming steps and how to control retiming:

- [Register Retiming Steps](#)
 - [Preventing Retiming](#)
 - [Selecting Transformation Options](#)
 - [Retiming Designs With Multiple Clocks](#)
 - [Retiming Registers With Path Group Constraints](#)
 - [Netlist Changes Performed by Register Retiming](#)
 - [Delay Threshold Optimization](#)
-

Register Retiming Steps

When you enable register retiming with either retiming method, Design Compiler moves registers using the following two-step approach:

1. Minimizes the clock period

The registers are moved to ensure the smallest clock period for the retimed circuit.

2. Minimizes the register count

The minimum clock period determined in the first step is compared to a user-defined clock period target. If the minimum clock period is smaller than or equal to the target clock period, the target clock period is used and a register distribution is computed that accommodates the target clock period with the smallest number of registers possible. If the target clock period is smaller than the minimum clock period, the number of registers is minimized for the minimum clock period.

Preventing Retiming

Sometimes it is best to avoid retiming some registers in a design. For example, registers driving primary outputs that have to stay in place because of a particular design style should not be retimed. In this case, set the `dont_retime` attribute on these output registers.

Another example of not moving certain registers occurs when you want to keep the controller registers of a design in place while allowing the datapath registers to move. Keeping the controller registers in place lets you easily identify these registers and relate them to the original HDL code. In this case, you can set the `dont_retime` attribute on the registers in the controller or on the controller cell itself (if it is a hierarchical cell).

The `dont_retime` attribute prevents a register from moving during retiming but allows sequential mapping to map the register to a different flip-flop.

Use the `set_dont_retime` command to control the designs or cells that can be retimed. When set to `true` (the default), the command sets the `dont_retime` attribute on specific cells and designs in the current design so that sequential cells are not moved during retiming optimizations. For example,

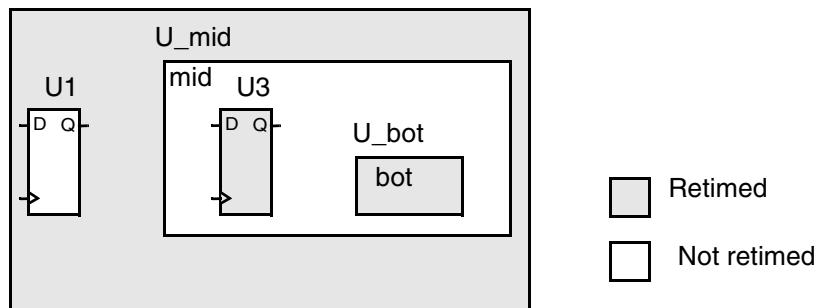
```
prompt> set_dont_retime [get_cells {z1_reg z2_reg}] true
```

Setting the `dont_retime` attribute on a hierarchical cell implies that the attribute is set on all sequential cells below it that do not have the `dont_retime` attribute set to `false`. A leaf-level cell that has the `dont_retime` attribute is not retimed if it is a sequential cell. When the `dont_retime` attribute is set on a design, all sequential cells that do not have the `dont_retime` attribute set to `false` are not retimed. For example, consider the following sequence of commands:

```
prompt> set_dont_retime [get_cells U1]
prompt> set_dont_retime [get_designs mid] true
prompt> set_dont_retime [get_cells U_mid/U3] false
prompt> set_dont_retime [get_cells U_mid/U_bot] false
prompt> optimize_registers
```

[Figure 17-13](#) shows that cells U1 and design mid are not retimed.

Figure 17-13 Cells U1 and Design mid Are Not Retimed



Note:

The `set_dont_retime` command overrides the `set_transform_for_retimimg` command. That is, if the value of the `dont_retime` attribute is `true`, setting the `transform_for_retimimg` attribute to `decompose` or `multiclass` does not make a register retimable. A movable register cannot be moved across a register with the `dont_retime` or the `dont_touch` attribute.

Selecting Transformation Options

The `optimize_registers` and `set_optimize_registers` commands allow you to specify how the mapped registers are transformed to SEQGEN cells for retiming. There is one transformation option for synchronous registers and another for asynchronous registers. A synchronous register does not have any asynchronous input pins and an asynchronous register has at least one asynchronous input pin.

The transformation option for synchronous registers is `-sync_transform`. The values are `multiclass`, `decompose`, and `dont_retime`. The `multiclass` value specifies that the synchronous clear, set, and enable functionality is moved with the synchronous sequential cells (if they are moved during retiming). The `decompose` value specifies that any synchronous sequential cell is decomposed (transformed into an instance of a D flip-flop or latch and additional combinational logic to create the necessary synchronous functionality). The `dont_retime` value specifies that these registers are not to be moved. The default for this option is `multiclass`.

The transformation option for asynchronous registers is `-async_transform`. The values are `multiclass`, `decompose`, and `dont_retime`. The `multiclass` value specifies that the asynchronous clear and set as well as any synchronous clear, set, and enable functionality are moved with the asynchronous sequential cells (if they are moved during retiming). The `decompose` value specifies that any asynchronous sequential cell is decomposed. The `dont_retime` value specifies that these registers are not to be moved. The default for this option is `multiclass`.

Registers that are already SEQGEN instances are not affected by these settings. Their set, clear, and enable connections are controlled by HDL Compiler options.

Recommended Transformation Options for Pipelines

For pipelined designs, it is recommended that you use the following transformation options with the `optimize_registers` or `set_optimize_registers` command:

```
-sync_transform multiclass -async_transform multiclass
```

Because there are no class conflicts preventing registers in pipelines from being moved across combinational cells, using `multiclass` retiming for all types of registers is best. These settings give the best timing results with the smallest possible register count and area.

Note:

Individual attribute settings on cells or their parent cells override these option settings.

Recommended Transformation Options for Nonpipelines

For nonpipelined designs, it is recommended that you use the following transformation options with the `optimize_registers` or `set_optimize_registers` command:

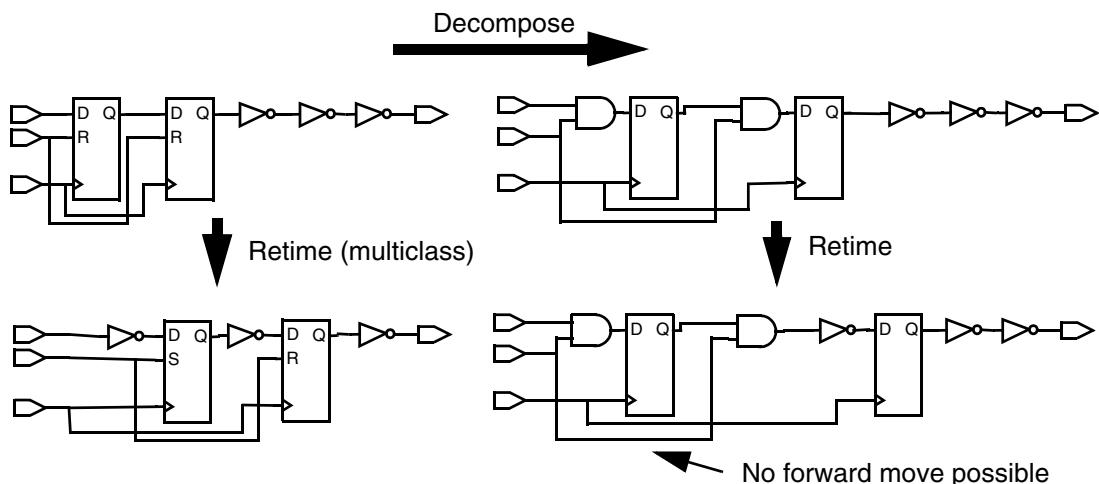
```
-sync_transform decompose -async_transform decompose
```

In most cases, decomposing all synchronous functionality ensures that no unnecessary class conflicts occur to limit the movability of the registers. The solution with the smallest possible delay or target delay should be found. An exception to this result can occur when the forward movability of registers is limited because of additional AND gates or OR gates as shown in [Figure 17-10](#). In this situation, setting the individual retiming attributes might help.

Note:

Individual attribute settings on cells or their parent cells override these option settings.

Figure 17-14 Reduced Mobility After Decomposition



Retiming Designs With Multiple Clocks

If the registers of the design are triggered by multiple different clocks or by both the rising and the falling edge of the same clocks, the retiming can be performed on only one clock at a time. The `optimize_registers` and `set_optimize_registers` commands offer you two ways to achieve this retiming, namely, by using the `-clock` option or not using this option.

With the `-clock` option you can specify that the registers for a single clock are retimed during one invocation of the `optimize_registers` or `set_optimize_registers` command. By default, only the registers triggered by the rising edge of the clock are retimed. If you want to retime the registers triggered by the falling edge of the clock, you have to use the `-edge` option with the value `fall`.

If the `-clock` option is not used, registers for all clocks are retimed during the first (register moving) phase of retiming. The retiming is performed one clock at a time. Clocks with a larger clock period are retimed before clocks with a smaller clock period. If two clocks have the same clock period, the clock with the larger number of registers is retimed first. For each single clock, the registers triggered by the rising edge are retimed before those triggered by the falling edge. Note that this default order might not yield the best possible results. Also, retiming all clocks in the first phase means that there is no incremental optimization of the combinational logic when different clocks are retimed. Therefore it is recommended that you determine the best order for retiming clocks yourself and apply that order by using multiple retiming runs with the `-clock` option.

When retiming latches, a two-phase clock system is being retimed. This means that the rising and falling edges of a clock or two or more different clocks have to be retimed together. If you specify a clock using the `-clock` option while using the `-latch` option, the `optimize_registers` or `set_optimize_registers` command retimes all the clocks and edges that need to be retimed with this clock. Note that even though the `-clock` option take only one argument, the command finds the other clock of the two-phase clock system.

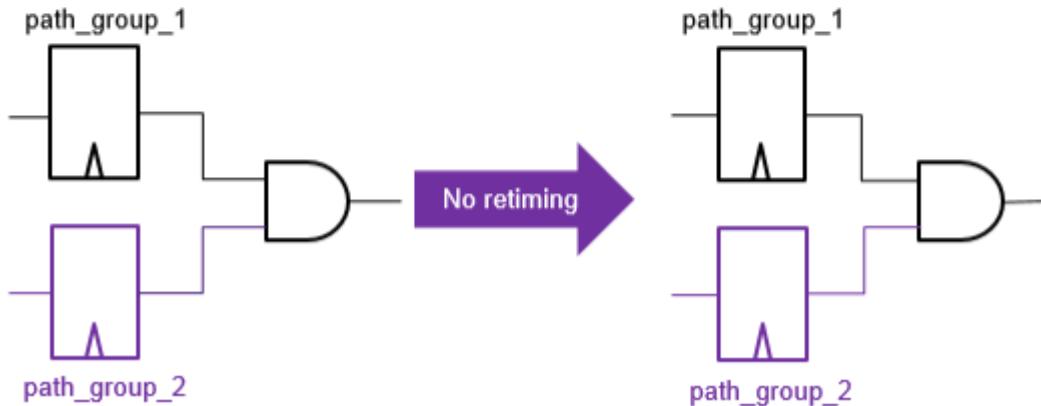
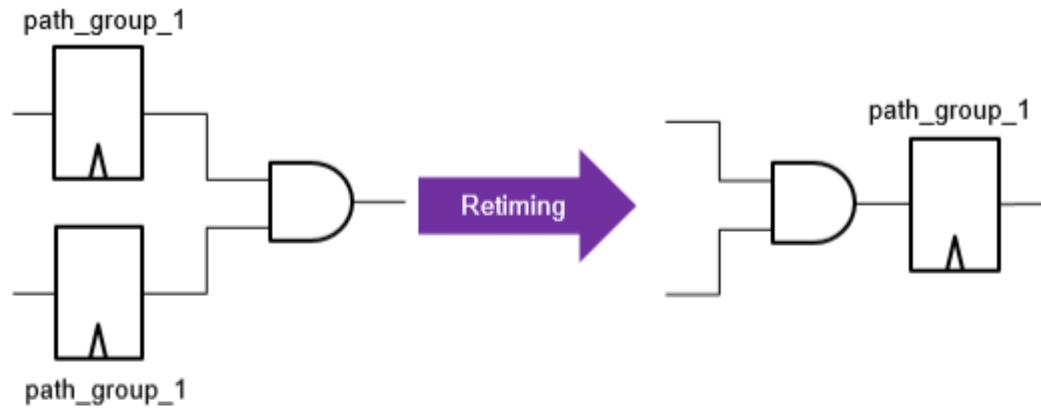
Retiming Registers With Path Group Constraints

Registers with path group constraints can be retimed if the new registers inherit the constraints from the original registers and contain no other timing exceptions except those defined on the path groups. Registers that belong to different path groups are considered separately during pipelined-logic retiming.

The tool allows the following pipelined-logic retiming for registers with path group constraints:

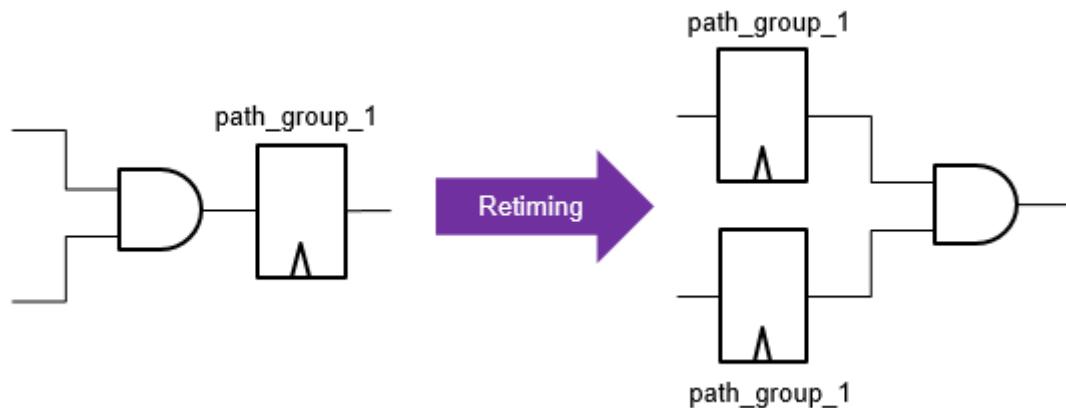
- Forward moves

The tool allows forward moves when the registers belong to the same path groups; otherwise, no retiming is performed. Registers resulting from combinational logic through forward moves inherit the path group constraints from the original registers before retiming. For example,



- Backward moves

Duplicate registers created as a result of backward moves inherit the path group constraints from the original registers before retiming. For example,



- Retiming in multicorner-multimode designs

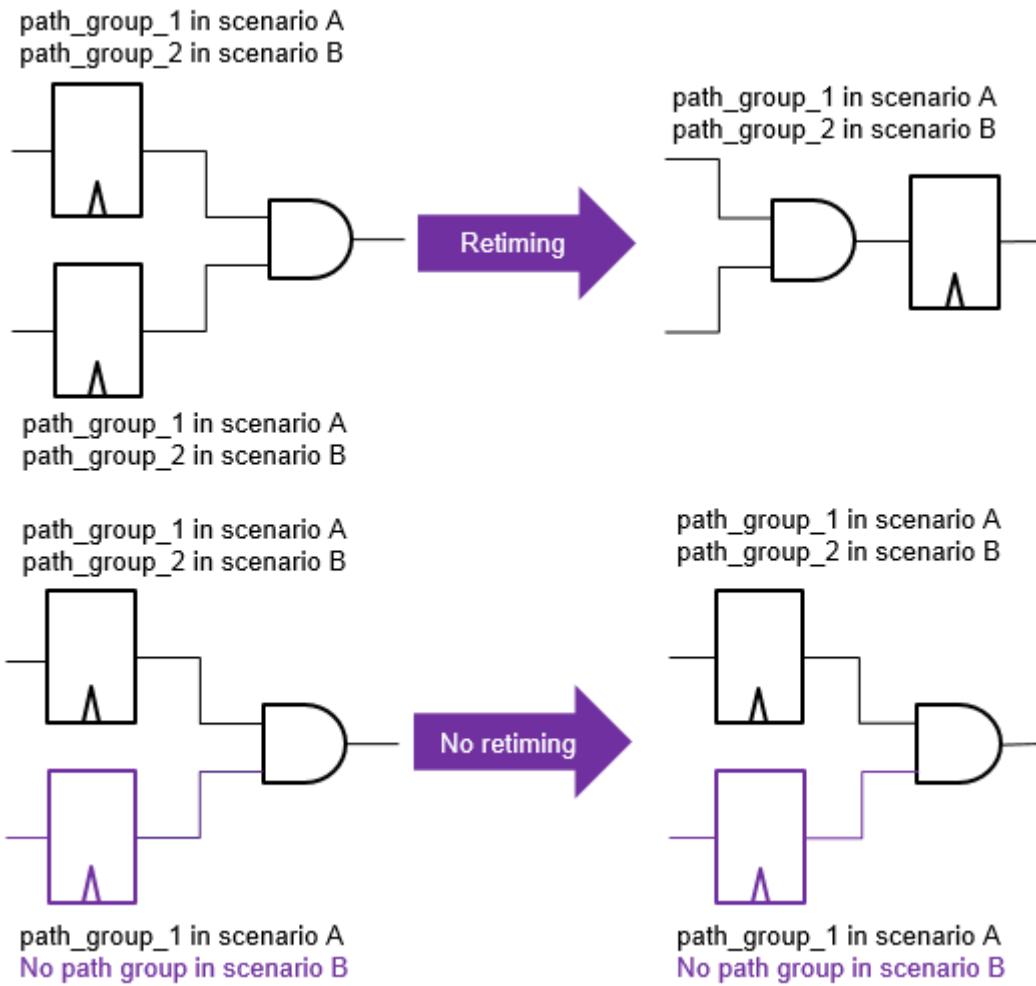
The tool identifies registers that belong to the same path groups in all scenarios and performs retiming. Retiming is not allowed across path groups.

In the following table, the reg_0 and reg_1 registers share the same path group specifications of all multicorner-multimode scenarios, so they are grouped together during retiming. However, the reg_2 and reg_3 registers do not share the same path group specifications; they remain two separate groups.

Scenario	reg_0	reg_1	reg_2	reg_3
S0	pg_A	pg_A	pg_A	pg_B
S1	pg_C	pg_C	pg_C	pg_C
S2	no group	no group	pg_D	pg_D
group_01		group_2		group_3

- Forward moves in multicorner-multimode designs

The tool allows forward moves only when the registers share the same path group specifications across all scenarios. For example,



The tool does not allow retiming in the following situations:

- Path groups are created using register pins instead of register cells. For example,
`create_path_group -to A_reg/D`
- Path groups include paths originating from and ending at the same register. For example,
`create_path_group -from [get_cells ff_reg*] -to [get_cells ff_reg*]`

Netlist Changes Performed by Register Retiming

Because retiming moves registers in the design, it is no longer possible to associate each register in the retimed design with exactly one register in the design before retiming. Therefore, new names have to be given to the registers. Registers that have the `dont_touch` or `dont_retime` attribute set on them are not retimed and not renamed.

Also, registers can be moved into hierarchical cells where there were no registers before retiming or into hierarchical cells where registers are not connected to the same set, clear, or enable signals. In these situations, additional pins have to be added to the hierarchical cells to have the necessary clock, set, clear, and enable nets. Adding these pins to a hierarchical cell changes its name and the name of its design.

Finally, if retiming cannot improve the delay or reduce the number of registers of the circuit, no register is moved or renamed and no incremental compilation is performed. The design is unchanged.

Delay Threshold Optimization

By default, retiming stops optimization when it detects paths that cannot be further improved by moving registers. Design Compiler reports these paths that limit further delay optimization as the critical loop in the log file when you specify the `-print_critical_loop` option with the `optimize_registers` or `set_optimize_registers` command. For more information about critical loops, see [Displaying the Sequence of Cells That Limits Delay Optimization](#).

When you specify the `optimize_registers` or `set_optimize_registers` command with the `-delay_threshold` option, retiming continues to improve paths until subcritical paths have delays less than or equal to the specified threshold.

Analyzing Retiming Results

When register retiming is running, the tool displays information to help you understand and improve results. In addition to the standard information output to `dc_shell`, you can get additional information by using various options with the `optimize_registers` or `set_optimize_registers` command.

For information about the messages that are displayed during retiming and the options you can use to find potential problems in the design, see the following topics:

- [Standard Output](#)
- [Checking for Design Features That Limit the Quality of Results](#)

- [Displaying the Sequence of Cells That Limits Delay Optimization](#)
-

Standard Output

By default, the tool provides the following informational messages that are in addition to the warning and error messages:

- Name and setup time of the preferred flip-flop.
 - Worst, best, and median clock-to-Q delay obtained, using the preferred flip-flop.
 - A table with histogram information for the clock-to-Q delays found from computing the median clock-to-Q delay.
 - Two values for the combinational delay between registers after registers are moved. These values, obtained from the retiming delay calculator, are referred to as the lower bound estimate and the critical path length. They are computed using slightly different methods.
 - The value used for the clock correction and its components (setup time, clock-to-Q delay, and clock uncertainty).
-

Checking for Design Features That Limit the Quality of Results

Sometimes it is useful to obtain more statistical information about the design being retimed. You can do this by using the `-check_design` option with the `optimize_registers` or `set_optimize_registers` command. The additional information can help you find potential problems in the design.

You can display the cells by name by using the `-verbose` option with the `-check_design` option. Using the `-verbose` option might produce many lines of output for large designs, but it can help identify the exact cause of a problem.

To properly analyze retiming results, you need to examine the output before and after the registers are moved. The [Output Before Registers Are Moved](#) and [Output After Registers Are Moved](#) topics describe the types of output.

Output Before Registers Are Moved

You should analyze the following output before the registers are moved by retiming:

- All base clocks in the design that trigger registers

For each base clock, all gated clocks that are derived from this base clock are printed. And for each gated clock, its polarity relative to its base clock is provided. A positive polarity means that a rising edge of the base clock results in a rising edge of the gated

clock. A negative polarity means that a rising edge of the base clock results in a falling edge of the gated clock. If the `-verbose` option is also used, then for each gated clock, all the registers derived from the gated clock and their polarities relative to the gated clock are printed.

- The five timing arcs with the largest delay in all the combinational cells

If a single cell has a large delay, this can severely limit the smallest delay the retiming can achieve. Therefore cells with a delay larger than a particular percentage (for example, ten percent) of the target clock period should be avoided. Two reasons such cells might exist are as follows:

First, a `dont_touch` attribute was put on a combinational hierarchical cell. Such a hierarchical cell appears as a single cell during the register moving phases. Consider removing the `dont_touch` attribute if the cell's delay is too large.

Second, the presence of a combinational cell from the library that is either very complex or has low drive strength and therefore a large delay. Consider compiling the design again after putting a `dont_use` attribute on this particular type of library cell.

- Delay distribution for all timing arcs in the design

The histogram information can indicate whether there are a few cells with a particularly large delay compared to others.

- Detailed description of the selection process for the preferred flip-flop
- Total number of combinational leaf cells in the design

The larger this number, the more complex the retiming becomes, and as a result CPU times might increase.

- Number of hierarchy cells with the `dont_touch` attribute
- Number of black box cells

Black box cells are cells without timing information. No registers are moved across them. If you do not want to have black box cells, check the linking of your design and the completeness of the library information.

- Total number of movable sequential cells
- Number of movable synchronous sequential cells with the `decompose` attribute
- Number of movable asynchronous sequential cells with the `decompose` attribute
- Number of movable synchronous sequential cells with the `multiclass` attribute
- Number of movable asynchronous sequential cells with the `multiclass` attribute
- Number of movable sequential cells with an asynchronous clear pin
- Number of movable sequential cells with an asynchronous set pin

- Total number of immovable sequential cells
If this number is large relative to the number of movable cells or if you suspect that some registers are not movable because of attributes or constraints you are unaware of, check the categories next in this list to find and possibly change the movability of some cells.
- Number of sequential cells that are not movable due to having the `dont_touch` attribute set
If the `dont_touch` attribute is not necessary, remove it.
- Number of sequential cells that are not movable due to having the `dont_touch` attribute set on one of their parent cells
If the `dont_touch` attribute is not necessary, remove it.
- Number of sequential cells that are not movable due to point-to-point exceptions
Check whether your design has to be implemented using multicycle or false paths. Change the design or timing constraints, if possible, to eliminate these immovable sequential cells.
- Number of sequential cells that are not movable due to insufficient technology library information
The information given on the flip-flop in the technology library is not sufficient to transform the instances of these flip-flops to SEQGEN cells. Try to compile the design before retiming, after you put a `dont_use` attribute on these flip-flop library cells.
- Number of asynchronous sequential cells that have the `dont_retime` attribute set
If the attribute is not necessary, remove it.
- Number of synchronous sequential cells that have the `dont_retime` attribute set
If the attribute is not necessary, remove it.

Output After Registers Are Moved

You should analyze the following output after the registers are moved by retiming:

- Total number of movable sequential cells.
- Number of movable sequential cells with an asynchronous clear pin.
- Number of movable sequential cells with an asynchronous set pin.

Displaying the Sequence of Cells That Limits Delay Optimization

In addition to using the `-check_design` argument with the `optimize_registers` or `set_optimize_registers` command, you can also use the `-print_critical_loop` option to find the part of the design that is limiting delay improvement.

When you use the `-print_critical_loop` option, the tool displays a sequence of combinational cells, ports, and nonmovable registers in the design. The location of the registers between these cells before and after retiming is also displayed. For each cell, the rise and fall delays and the total delays from the last register or port to the output of the cell are displayed. The names of the cells and output pins are those used in the netlist before retiming. Therefore you can recognize them by looking at a schematic for this netlist in a graphical display tool such as Design Vision.

The only exceptions are the cells inserted into the netlist when registers are decomposed. You cannot find these cells in the netlist before retiming, but some of them can show up in the critical loop display. Sometimes the same sequence of cells is displayed several times. This has no particular significance.

The three different classifications of critical loops that can be printed before the sequence of cells are as follows:

- Loop without primary Input/Outputs

In this case, the loop does not contain a primary output, a primary input, or a pin of a register that cannot move. Such a register is regarded as fixed (for example, because it has the `dont_touch` attribute set). The total delay of the cells in the loop and the number of registers in the loop determine the minimal delay that retiming can achieve. To further reduce delay, you might have to reduce the delay inside the loop (for example, by recoding the design or recompiling with different constraints). If the design allows such a modification, you can also add an additional register to the loop.

- Loop from primary input to primary output

This case includes loops that go from actual primary input to primary outputs as well as those that begin or end at a fixed register. If the loop begins or ends at a fixed register, you might want to check whether the `dont_touch` or `dont_retime` attribute placed on that register, or on any of its parent cells, is really needed. If the startpoint and endpoint are a primary input and a primary output, you can either reduce the combinational delay between the ports by recoding and recompiling the design, or you can add registers to the design at one or more of these ports (which increases the latency).

- Loop limited by node bounds

This case occurs when different classes of registers are present in the fanin or fanout of a cell (also referred to as a node), limiting the delay that can be achieved by retiming. To find which cells contribute to the register class conflict, look at the cells at the beginning and the end of the sequence. If the cells are combinational cells (that is, not fixed

registers, black boxes, or primary ports), they are the cells responsible for the class conflict.

If the cell at the beginning of the sequence is combinational, look at the registers in its fanin. Some of them will belong to different classes or be clocked by different base clocks. If this class difference is due to the control nets to synchronous pins, you might be able to reduce the delay further by using the `decompose` option or putting the `decompose` transformation attribute on these registers.

Alternatively, if the combinational cell at the end of the sequence has the bound, you can apply a similar process to the registers in the fanout of the cell.

[Example 17-1](#) shows the critical loop output with a node bound.

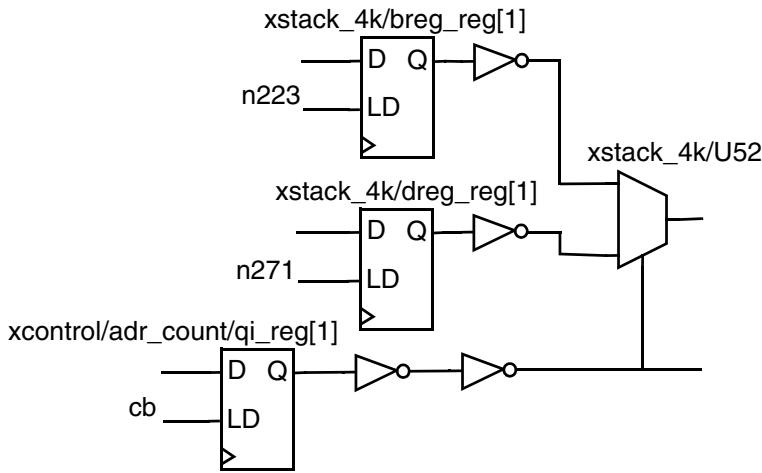
Example 17-1 Critical Loop Output With a Node Bound

```
Critical Loop(s) for Minimum Period Retiming
-----
---- loop limited by node bound(s) ----

Point           Incr          Path
-----
xstack_4k/U52/Z (MX21LC)      ( 0.19 0.18)  0.19 0.18
xstack_4k/U47/Z (MX21LB)      ( 0.12 0.13)  0.30 0.32
xstack_4k/U65/Z (ND2C)        ( 0.06 0.09)  0.38 0.39
xalu/U90/Z (MX21LC)          ( 0.16 0.19)  0.55 0.57
xalu/U91/Z (N1F)             ( 0.08 0.10)  0.66 0.66
xalu/add_59/plus/plus/U24/Z (NR2C) ( 0.12 0.10)  0.78 0.76
xalu/add_59/plus/plus/U18/Z (EON1C) ( 0.07 0.10)  0.83 0.88
xalu/add_59/plus/plus/U31/Z (EN3A)  ( 0.21 0.22)  1.09 1.10
xalu/U96/Z (N1B)              ( 0.05 0.06)  1.15 1.15
*** 1 register(s) AFTER min. period retiming here ***
xalu/U86/Z (OR2B)            ( 0.15 0.19)  0.15 0.19
xalu/U85/Z (ND2B)             ( 0.22 0.27)  0.41 0.42
*** 1 register(s) BEFORE min. period retiming here ***
xseg7dec/U6/Z (NR2L)          ( 0.22 0.11)  0.64 0.52
xseg7dec/U13/Z (AO2A)          ( 0.17 0.28)  0.69 0.92
xseg7dec/U30/Z (ND2B)          ( 0.06 0.08)  0.98 0.77
g/**outside** (**out_port**)    ( 0.34 0.34)  1.32 1.32
(++)
-----
(+) I/O port resulting from a pin of a fixed register or black box
(++) Delay may have been corrected by clock to Q (input) or setup
(output)
```

The cell `xstack_4k/U52` is the one that has the conflict. [Figure 17-15](#) shows the fanin of this cell.

Figure 17-15 Analyzing Node Bounds Example



The three registers in the fanin all belong to different classes because their LD (load enable) pins are driven by different nets. Because the design is not a pure pipeline, you could improve performance by following the general recommendation of using decomposition for all registers. If this increases the area too much, an alternative tactic is to set the `decompose` attribute on these registers individually while using the `multiclass` option for all other registers in the `optimize_registers` or `set_optimize_registers` command.

```

prompt> set_transform_for_retiming \
           [get_cells "xstack_4k/breg_reg[1]"] decompose
prompt> set_transform_for_retiming \
           [get_cells "xstack_4k/dreg_reg[1]"] decompose
prompt> set_transform_for_retiming \
           [get_cells "xcontrol/adr_count/qi_reg[1]"] decompose
prompt> optimize_registers -sync_transform multiclass \
           -async_transform multiclass
    
```

You might have to repeat this process for class conflicts at other nodes, depending on the outcome of subsequent runs of the `optimize_registers` or `set_optimize_registers` command.

Verifying Retimed Designs

When Design Compiler performs any retiming, an automated setup file is necessary for verification with the Formality tool. The automated setup file includes retiming optimization information that helps Formality verify retimed designs. To remind you that Formality

requires an automated setup file, Design Compiler displays the following message when retiming is performed:

Information: Retiming is enabled. SVF file must be used for formal verification. (OPT-1210)

The Formality tool supports the verification of designs that are one-pass retimed by the `set_optimize_registers` command followed by the `compile_ultra` command. To verify these designs, Formality uses an automated checkpoint verification flow that verifies checkpoint netlists in the .svf file generated by Design Compiler.

To enable checkpoint verification in Formality, Design Compiler automatically generates .svf guidance with checkpoints when retiming is triggered by the `compile_ultra` command. You must use the `set_optimize_registers` command before running the `compile_ultra` command to enable this feature.

At each checkpoint, Design Compiler creates a netlist and writes a `guide_checkpoint` guidance command to the .svf file. When a `guide_checkpoint` command is found in the .svf file, the Formality tool verifies the checkpoint netlist against the RTL. Formality verifies the checkpoint netlists in sequence using a single `verify` command. If the verification succeeds, Formality replaces the reference design with the verified implementation checkpoint netlist for the subsequent verification of either the next checkpoint netlist or the final retimed netlist. However, if the checkpoint verification is not successful, the reference design is retained for the verification of the subsequent netlist.

Using checkpoint guidance to verify the designs reduces the inherent complexity of the two-pass verification flow, results in higher completion rates, and results in better QoR.

The reports and functionality to debug the verifications also support the use of checkpoint netlists.

Limitation:

Checkpoint verification is not supported in some cases. You still need to use the two-pass verification flow for successful verification in the following cases:

- When retiming is performed multiple times in various scenarios, typically with these commands:
 - The `compile_ultra -retime` command followed by the `compile_ultra -incremental -retime` command
 - The `set_optimize_registers` and `compile_ultra` commands followed by the `compile_ultra -retime` command

See Also

- [Verifying Functional Equivalence](#)
Provides information about design verification
- The *Formality User Guide*

18

Gate-Level Optimization

During gate-level optimizations, Design Compiler implements the final netlist by making optimal selections of library cells. It performs tasks such as delay optimization, design rule fixing, and area recovery.

Before you read further, see [Optimization Flow](#) to understand how gate-level optimizations fit into the overall compile flow.

Gate-level optimization performs the following tasks:

- [Delay Optimization](#)
- [Power Optimization](#)
- [Design Rule Fixing](#)
- [Area Recovery](#)

Delay Optimization

In this phase, Design Compiler attempts to fix existing delay violations by traversing the critical path. It applies local transformations such as upsizing, load isolation and splitting, and revisits mapping of sequential paths.

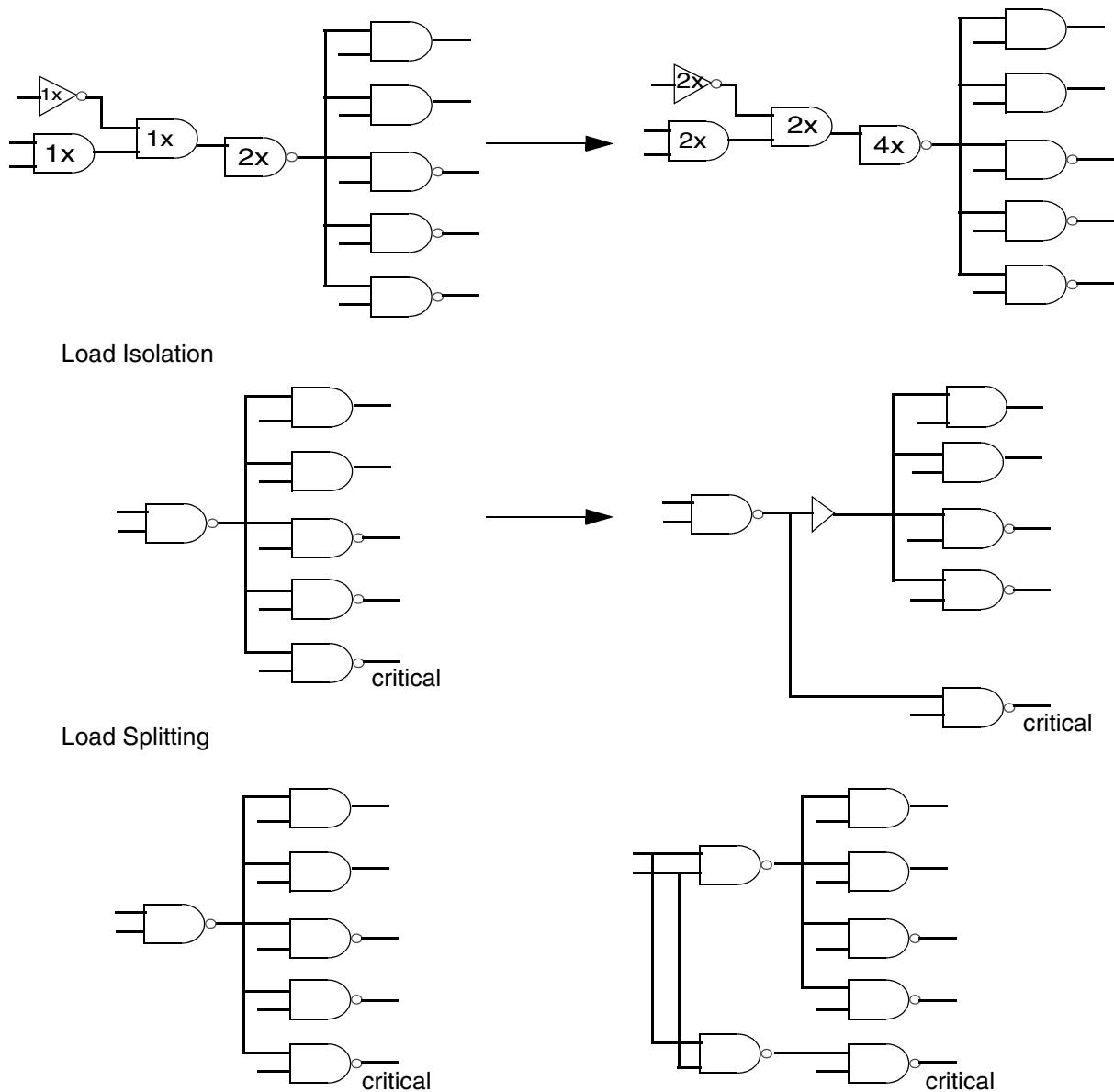
Design Compiler also identifies the critical path and attempts to do a full compile on only the logic along that path. This process then repeats on the new critical path. Additionally, Design Compiler resizes or speeds up sequential cells on the critical path. The tool also uses a high-effort algorithm to remove sequential elements for which the logic leading to a constant value is particularly complex. If you do not want Design Compiler to perform this extra optimization, set the `compile_seqmap_propagate_high_effort` variable to `false` (the default is `true`). For more information, see [Automatically Removing Unnecessary Registers](#).

The following strategies can help achieve a faster design:

- Automatically ungroup hierarchies along the critical path by using the `compile_ultra` command. See [Automatic Ungrouping](#).
- Ungroup all or part of the hierarchy. This can give optimization more freedom to change logic that previously spanned a hierarchical boundary. See [Removing Levels of Hierarchy](#).
- Enable boundary optimization. See [Optimizing Across Hierarchical Boundaries](#).
- Use the `group_path` command to create path groups. See [Creating Path Groups](#).
- Specify a critical range, so that Design Compiler optimizes not only the critical path but paths within that range of the critical path as well. See [Optimizing Near-Critical Paths](#).
- In the design exploration phase, you might want to give delay a higher priority than design rules by using the `set_cost_priority -delay` command.
- Fix heavily loaded nets by using the `balance_buffer` command or by fixing design rules. See [Fixing Heavily Loaded Nets](#).

[Figure 18-1](#) shows some common local optimization steps. Design Compiler takes design rules into account during this phase. When two circuit solutions offer the same delay performance, Design Compiler implements the solution that has the lower design rule cost.

Figure 18-1 Delay Optimization Steps
Upsizing



The compile log displays the delay optimization phase as shown in [Example 18-1](#).

Example 18-1 Delay Optimization in Compile Log

Beginning Delay Optimization Phase

ELAPSED TIME	AREA	WORST SLACK	NEG SLACK	TOTAL RULE	NEG COST	DESIGN ENDPOINT
0:00:05	136	2.11	23.2	18.0		out_reg[10]/D
0:00:05	138	1.53	16.9	18.0		out_reg[10]/D

Power Optimization

Design Compiler automatically performs leakage power optimization (except in DC Expert, in which case you must specifically enable the optimization). During leakage power optimization, Design Compiler tries to reduce the overall leakage power in your design without affecting the performance. To do this, the optimization is performed on paths that are not timing-critical. When the target libraries are characterized for leakage power and contain cells characterized for multiple threshold voltages, Design Compiler uses the library cells with appropriate threshold voltages to reduce the leakage power of the design.

When using DC Expert, use the following command to enable leakage power optimization:

```
prompt> set_leakage_optimization true
```

You can also perform dynamic power optimization to reduce dynamic power consumption without affecting the performance. Dynamic power optimization requires switching activity information. The tool optimizes the dynamic power by placing nets with high switching activity close to each other. Because the dynamic power saving is based on the switching activity of the nets, you need to annotate the switching activity by using the `read_saif` command. You can also optimize dynamic power incrementally to provide better QoR and reduce the runtime.

To enable dynamic optimization in all Design Compiler tools, use the following command:

```
prompt> set_dynamic_optimization true
```

For more information about dynamic power optimization, see the *Power Compiler User Guide*.

See Also

- [Leakage Power and Dynamic Power Optimization](#)

Design Rule Fixing

Design rules are provided in the vendor logic library to ensure that the product meets specifications and works as intended. During design rule fixing, the goal is to correct design rule violations by inserting buffers or resizing existing cells. Design Compiler tries to fix the violations without affecting timing and area results, but if necessary, it does violate the optimization constraints. Whenever possible, Design Compiler fixes design rule violations by resizing gates across multiple logic levels—as opposed to adding buffers to the circuitry.

The compile log displays the design rule fixing phase as shown in [Example 18-2](#).

Example 18-2 Design Rule Fixing in Compile Log

```
Beginning Design Rule Fixing
(max_capacitance) (max_fanout) (max_capacitance)
-----
ELAPSED          WORST NEG TOTAL NEG DESIGN
TIME      AREA     SLACK    SLACK   RULE COST      ENDPOINT
-----
0:00:08      153    0.60      6.6       8.0
0:00:08      146    0.60      6.6       5.0
```

You can direct Design Compiler to avoid design rule fixing or to compile with only design rule fixing. For more information, see [Managing Constraint Priorities](#).

In topographical mode, you can perform additional design rule fixing, if it is needed, by setting the `compile_final_drc_fix` variable. This variable is supported when you use the `compile_ultra` or the `compile_ultra -spg` command. Set the variable before running a full or incremental compile. Use the available values to specify whether to fix the maximum transition, maximum fanout, maximum capacitance, or all of the design rule constraints.

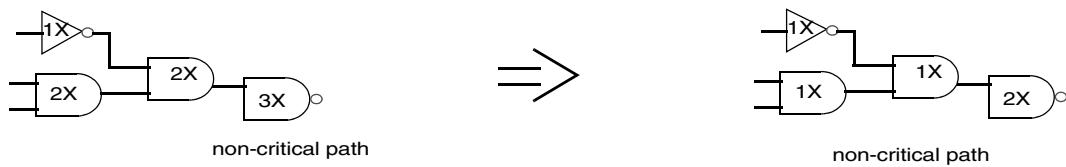
Area Recovery

During area optimization, the Design Compiler tool attempts to minimize the number of gates in the design without degrading delay cost.

In DC Ultra (in wire load mode and topographical mode) and in the Design Compiler Graphical tool, you can use the `compile_ultra` and `optimize_netlist -area` commands to provide significant area improvements. To maximize area benefits, run synthesis using the `compile_ultra` command and then run the `optimize_netlist -area` command as the final area recovery step at the end of the synthesis flow. The `optimize_netlist -area` command performs monotonic gate-to-gate optimization to improve area without degrading timing or leakage. You can also use the command on legacy netlists.

In DC Expert, if you have placed area constraints on your design with the `set_max_area` command, Design Compiler now attempts to minimize the number of gates in the design. The goal is to reduce design area without introducing design rule and delay violations. The tool downsizes cells to recover area. It performs downsizing only on paths that have positive slack as shown in [Figure 18-2](#).

Figure 18-2 Example of Downsizing Cells During Area Recovery



In addition, the tool includes a fast back-end sequential mapper that does automatic sequential area recovery. It identifies clusters of registers with similar functionality and timing and optimizes the area of these register clusters as a whole. See [Automatically Removing Unnecessary Registers](#).

The compile log displays the area recovery phase as shown in [Example 18-3](#).

Example 18-3 Area Recovery in Compile Log

Beginning Area-Recovery Phase (max_area 145)					
ELAPSED TIME	AREA	WORST NEG SLACK	TOTAL NEG SLACK	DESIGN RULE COST	ENDPOINT
0:00:09	149.0	1.05	8.5	0.0	

If you do not place area constraints on your design, Design Compiler performs a limited series of downsizing and area cleanup steps as shown in [Example 18-4](#).

Example 18-4 Area Recovery in Compile Log

Beginning Area-Recovery Phase (cleanup)					
ELAPSED TIME	AREA	WORST NEG SLACK	TOTAL NEG SLACK	DESIGN RULE COST	ENDPOINT
0:00:10	16413.4	0.00	0.0	0.0	
0:00:10	16413.4	0.00	0.0	0.0	
0:00:10	16408.2	0.00	0.0	0.0	

Using the `-map_effort` or `-area_effort` option of the `compile` command, you can direct Design Compiler to put a medium, or high effort into area optimization.

- Medium effort
Design Compiler does gate sizing and buffer and inverter cleanup.
- High effort
Design Compiler tries still more gate minimization strategies. The tool adds gate composition to the process and allocates more CPU time than medium effort.

Note:

Whichever area optimization effort level you choose, the overall constraints cost vector (described in [Compile Cost Function](#)) prevails. Even during area optimization, if the Design Compiler tool finds a new opportunity to improve delay cost, it makes the change—even if it increases area cost. Area always has a lower priority than delay.

19

Using Topographical Technology

Topographical technology allows you to accurately predict post-layout timing, area, and power during RTL synthesis without the need for timing approximations based on wire load models. It uses Synopsys' placement and optimization technologies to drive accurate timing prediction within synthesis, ensuring better correlation with the final physical design. This technology is built in as part of the DC Ultra feature set and is available only by using the `compile_ultra` command in topographical mode.

Design Compiler topographical mode requires a DC Ultra license and a DesignWare license. A Milkyway-Interface license is also necessary if the Milkyway flow is used; this license is automatically included with the DC Ultra license.

To learn how to use Design Compiler in topographical mode, see

- [Overview of Topographical Technology](#)
- [Inputs and Outputs in Design Compiler Topographical Mode](#)
- [Defining the Design Environment](#)
- [Performing Automatic High-Fanout Synthesis](#)
- [Performing Manual High-Fanout Synthesis](#)
- [Test Synthesis in Topographical Mode](#)
- [Compile Flows in Topographical Mode](#)
- [Reducing Runtime](#)

- Handling Unsupported Commands, Options, and Variables
- Using Design Compiler Graphical

Overview of Topographical Technology

In ultra deep submicron designs, interconnect parasitics have a major effect on path delays; accurate estimates of resistance and capacitance are necessary to calculate path delays. In topographical mode, Design Compiler leverages the Synopsys physical implementation solution to derive the “virtual layout” of the design so that the tool can accurately predict and use real net capacitances instead of statistical net approximations based on wire load models. If wire load models are present, they are ignored.

In addition, the tool updates capacitances as synthesis progresses. That is, it considers the variation of net capacitances in the design by adjusting placement-derived net delays based on an updated virtual layout at multiple points during synthesis. This approach eliminates the need for overconstraining the design or using optimistic wire load models in synthesis. The accurate prediction of net capacitances drives Design Compiler to generate a netlist that is optimized for all design goals including area, timing, test, and power. It also results in a better starting point for physical implementation.

When you run the `compile_ultra` command in topographical mode, Design Compiler automatically uses topographical technology. All `compile_ultra` command options are supported.

Note:

Design Compiler in topographical mode supports only a subset of `dc_shell` commands, command options, and variables. For more information, see [Handling Unsupported Commands, Options, and Variables](#).

Topographical technology supports all synthesis flows, including the following:

- Automatic high-fanout synthesis
- Test-ready compilation flow (basic scan and DFT MAX scan compression)
- Clock-gating flow
- Register retiming

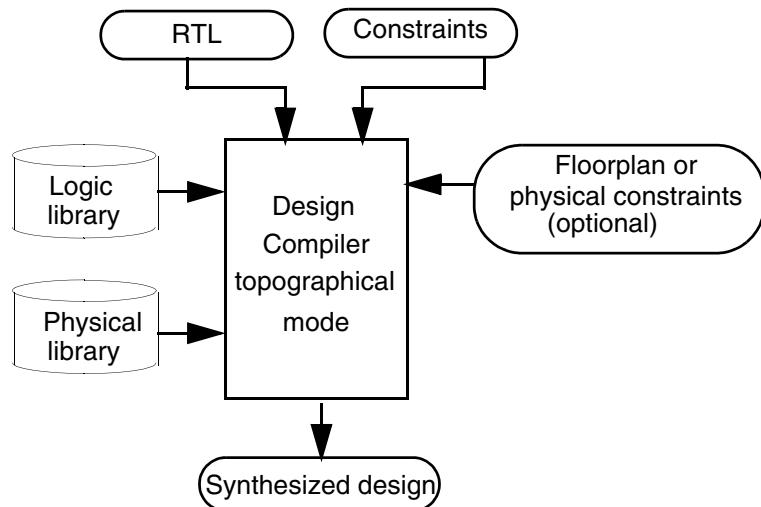
See Also

- [DC Ultra](#)
- [Topographical Mode](#)
- [The Synthesis Flow](#)

Inputs and Outputs in Design Compiler Topographical Mode

[Figure 19-1](#) shows the inputs and outputs in Design Compiler topographical mode. The sections that follow provide more information.

Figure 19-1 Inputs and Outputs in Design Compiler Topographical Mode



[Table 19-1](#) and [Table 19-2](#) describe the inputs and outputs in topographical mode.

Table 19-1 Inputs in Design Compiler Topographical Mode

Input	Description
Design	RTL or gate-level netlist.
Constraints	Timing and optimization constraints. Note that if you specify wire load models, Design Compiler ignores them.
Logic library	Liberty format (.lib or .db). For more information, see Specifying Logic Libraries .
Physical library	Milkyway format. For more information, see Specifying Physical Libraries . Note: The .pdb format is not supported.
Floorplan or physical constraints	High-level physical constraints that determine items such as core area and shape, port location, macro location and orientation, voltage areas, placement blockages, and placement bounds. For more information, see Using Floorplan Physical Constraints .

Table 19-2 Outputs in Design Compiler Topographical Mode

Output	Description
.ddc	This format contains back-annotated net delays and constraints. Subsequent topographical mode sessions restore virtual layout data. The .ddc format is recommended for subsequent topographical mode optimizations and verification.
Milkyway	This format contains back-annotated net delays and constraints. The Milkyway format cannot be read back into topographical mode for subsequent optimizations. This format is recommended if you intend to use Synopsys tools for the back-end flow.
ASCII	This format does not contain back-annotated delays or Synopsys design constraints (SDC constraints). Use the <code>write_sdc</code> command to write out the SDC constraints and the <code>write_parasitics</code> command to write out parasitics. This format is recommended only if you intend to use a third-party tool.

Note:

You cannot use the Milkyway format to store design data for unmapped designs or non-uniquified designs. Before you use the `write_milkyway` command, run the following command:

```
uniqualify -force -dont_skip_empty_designs
```

Defining the Design Environment

Before a design can be optimized, you must define the environment in which the design is expected to operate. You define the environment by specifying operating conditions and system interface characteristics. For information about defining the design environment in topographical mode, see [Defining the Environment Using Topographical Mode](#).

See Also

- [Comparing Design Compiler Topographical and IC Compiler Environments](#)

Performing Automatic High-Fanout Synthesis

Design Compiler in topographical mode performs automatic high-fanout synthesis by default. During high-fanout synthesis, the tool analyzes the buffer trees to determine the fanout thresholds, and then it removes and builds buffer trees to improve performance. The high-fanout synthesis engine does not buffer nets that are set as ideal nets, nets that are disabled due to design rule constraints, or nets that have a `dont_touch_network` setting.

To specify the constraints to be used when running automatic high-fanout synthesis, use the `set_ahfs_options` command. For example, you can control port punching by specifying the `-enable_port_punching` and `-no_port_punching` options, and you can specify a list of buffers or inverters to be used when constructing new buffer or inverter trees by specifying the `-default_reference` option.

After automatic high-fanout synthesis, Design Compiler automatically sets a global design-based attribute that prevents IC Compiler from doing automatic high-fanout synthesis. If you need to enable automatic high-fanout synthesis in IC Compiler, for example, if you are changing any ideal or `dont_touch_network` settings, apply the `set_ahfs_options` command in the back-end script.

By default, Design Compiler uses the virtual router during automatic high-fanout synthesis. However, you can enable Zroute-based global buffering in Design Compiler Graphical to reduce congestion along narrow channels in macro-intensive designs. Zroute-based global buffering provides more accurate congestion information than virtual routing.

To enable Zroute-based global buffering during automatic high-fanout synthesis, set the `-global_route` option of the `set_ahfs_options` command to `true` before compile. When enabled, the command takes effect during the compile and incremental compile stages. Zroute-based global buffering might increase runtime and impact area and timing; therefore, use it only on macro-intensive designs with narrow channels.

To report all the options set for running automatic high-fanout synthesis, use the `report_ahfs_options` command:

```
prompt> report_ahfs_options
Report AHFS options:
*****
AHFS options for the design:
  Enable port punching: ON
  Default Reference : buffer_1
  Preserve Boundary Phase : OFF
  No Port Punching on these hier cells : ""
  Global Route: ON
```

See Also

- [Reducing Routing Congestion](#)
- [Performing Manual High-Fanout Synthesis](#)

Performing Manual High-Fanout Synthesis

By default, Design Compiler in topographical mode performs automatic high-fanout synthesis when you run the `compile_ultra` command. You can manually run high-fanout synthesis to identify and resolve any suboptimal buffering that was performed by the `compile_ultra` command.

Perform manual high-fanout synthesis on a mapped netlist using the following flow:

1. Run the `report_buffer_tree` command to report the current buffer tree implementation.
2. Identify any suboptimal buffering that was performed by the `compile_ultra` command.
3. Remove the existing buffer trees and create additional buffer trees as needed using the `clean_buffer_tree` and `create_buffer_tree` commands, respectively.

You can preserve preexisting buffer trees by specifying the `-incremental` option when you run the `create_buffer_tree` command. In this case, the `create_buffer_tree` command constructs a buffer tree, if needed, on each net specified by the `-from` option to reduce the high fanout of each net. The tool does not remove any existing buffers or inverters; the scope of buffer tree creation is a single specified net.

The inserted buffers and inverters might be optimized further by other optimization commands, such as `compile_ultra -incremental` and `optimize_netlist -area`. You can preserve specific buffers and inverters during optimization by setting optimization restriction attributes on them, such as `size_only` or `dont_touch`.

See Also

- [Performing Automatic High-Fanout Synthesis](#)

Test Synthesis in Topographical Mode

Topographical mode supports test synthesis. The flow is similar to the one in Design Compiler wire load mode except that the `insert_dft` command is used for stitch-only. Topographical mode supports basic scan and scan compression.

To perform scan insertion within topographical mode, use a script similar to the following:

```
dc_shell -topographical_mode  
read_ddc top_elaborated.ddc  
source top_constraint.sdc  
source physical_constraints.tcl  
compile_ultra -gate_clock -scan  
  
## Provide DFT specifications
```

```
set_dft_signal ...  
  
create_test_protocol  
dft_drc  
preview_dft  
insert_dft  
dft_drc  
  
compile_ultra -incremental -scan  
write_scan_def -output dft.scandef
```

For more information about scan insertion, see the DFT Compiler documentation.

See Also

- [Performing a Test-Ready Compile](#)

Compile Flows in Topographical Mode

In addition to supporting a top-down `compile_ultra` flow, as described in [Top-Down Compilation](#), Design Compiler topographical mode supports incremental and hierarchical flows:

- [Performing an Incremental Compile](#)

The `-incremental` option of the `compile_ultra` command allows you to employ a second-pass, incremental compile strategy.

- [Performing a Bottom-up Hierarchical Compile](#)

Topographical mode supports a hierarchical flow if you need to address design and runtime challenges or use a divide and conquer synthesis approach. In topographical mode, the `-top` option of the `compile_ultra` command enables you to stitch compiled physical blocks into the top-level design.

Performing an Incremental Compile

You can perform a second-pass, incremental compile by using the `-incremental` option with the `compile_ultra` command. This enables topographical-based optimization for post-topographical-based synthesis flows such as retiming, design-for-test (DFT), DFT MAX, and minor netlist edits. The primary focus in Design Compiler topographical mode is to maintain QoR correlation; therefore, only limited changes to the netlist can be made.

Use the incremental compile strategy to meet the following goals:

- Improve design QoR

- Fix the netlist after manual netlist edits or constraint changes
- Fix the netlist after various synthesis steps have been performed on the compiled design, for example, after `insert_dft` or register retiming
- Control design rule fixing by using the `-no_design_rule` or `-only_design_rule` option in combination with the `-incremental` option

The incremental compile also supports adaptive retiming with the `compile_ultra -incremental -retime` command.

Running the `compile_ultra -incremental` command on a topographical netlist results in placement-based optimization only. This compile should not be thought of as an incremental mapping.

Note:

When you use the `insert_buffer` command and `remove_buffer` command described in [Editing Designs](#), the `report_timing` command does not report placement-based timing for the edited cells. To update timing, run the `compile_ultra -incremental` command.

When using the `-incremental` option, keep the following in mind:

- Marking library cells with the `dont_use` attribute does not work for an incremental flow when it is applied to a topographical netlist. Make sure to apply any `set_dont_use` attributes before the first pass of a topographical-based synthesis.
- If you intend to use boundary optimization and scan insertion, apply them to the first pass of a topographical-based synthesis.
- Avoid significant constraint changes in the incremental pass.

Note:

Physical constraint changes are not supported.

Performing a Bottom-up Hierarchical Compile

In a bottom-up hierarchical compile flow, you compile the subdesigns separately and then incorporate them in the top-level design.

The recommended strategy is a top-down compile flow. However, topographical mode supports a hierarchical bottom-up flow if you need to address design and runtime challenges or use a divide-and-conquer synthesis approach. In the bottom-up strategy, individual subblocks are constrained and compiled separately. The compiled subblocks are then included in subsequent top-level synthesis. In topographical mode, the tool can read the following types of hierarchical blocks:

- Netlists generated in topographical mode

- Block abstractions generated in topographical mode or IC Compiler

That is, you can compile the subblock in topographical mode and provide it to the top-level design as a full .ddc netlist or a block abstraction. Alternatively, you can continue working on the subblock in IC Compiler to create a placed-and-routed block abstraction, which you can then provide to the top-level design in topographical mode. Timing and physical information of the subblock are propagated to the top-level for physical synthesis in topographical mode. In addition, you can provide the placement location for a subblock during top-level synthesis to maintain correlation with IC Compiler.

In the hierarchical or bottom-up flow, use the `compile_ultra -scan -gate_clock` command to perform top-level design integration. The tool automatically propagates block-level timing and placement to the top level and uses them to drive optimization. In addition, you can specify the placement location for the subblock. Top-level optimization is placement-aware and can be driven with the same physical constraints as your back-end tool.

The following topics describe the Design Compiler topographical mode hierarchical flow:

- [Overview of Bottom-Up Compile](#)
- [Compiling the Subblock](#)
- [Compiling the Design at the Top Level](#)

Overview of Bottom-Up Compile

In the bottom-up flow, first you compile the subblock in topographical mode and save the mapped subblock as a netlist in .ddc format. Then, you can continue working on the subblock (.ddc netlist) in IC Compiler to generate a block abstraction.

Alternatively, you can compile the subblock in topographical mode and save the mapped subblock as a block abstraction in .ddc format.

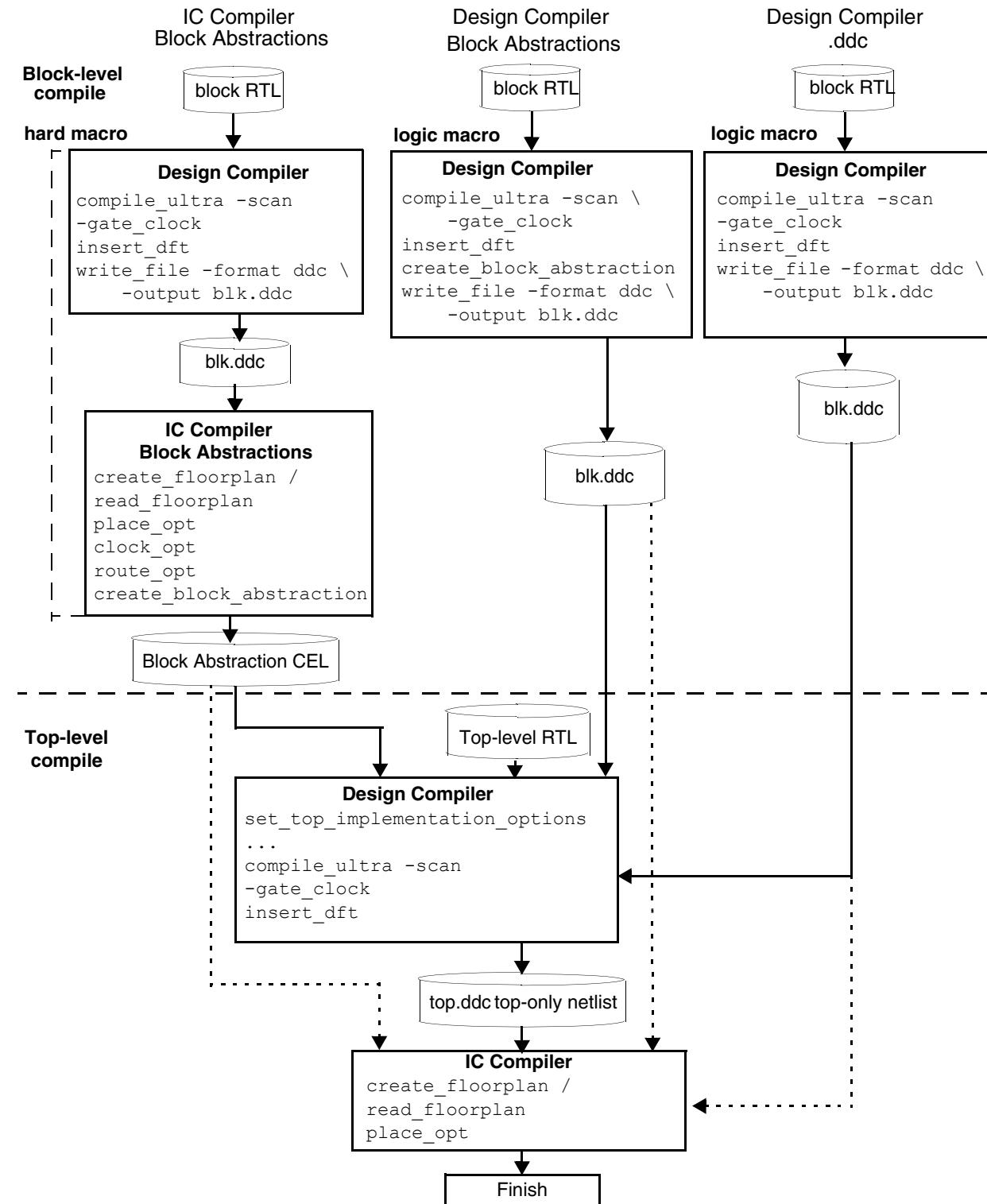
Top-level synthesis can accept the following types of mapped subblocks:

- .ddc netlist synthesized in topographical mode
- Block abstraction created in topographical mode
- Block abstraction created in IC Compiler

Note:

IC Compiler cannot accept Design Compiler block abstractions.

Figure 19-2 Overview of the Hierarchical Flow for Designs Containing Block Abstractions



The bottom-up flow requires these main steps:

1. Compile the subdesigns independently. See [Compiling the Subblock](#).
2. Read in the top-level design and any compiled subdesigns not already in memory; compile the top-level design. See [Compiling the Design at the Top Level](#).

Compiling the Subblock

Compiling the subblock requires the following steps:

1. Specify the logic and physical libraries as described in [Setting Up and Working With Libraries](#).
2. Read in the subblock (Verilog, VHDL, netlist, or .ddc) and set the current design to the subblock.
3. Apply block-level timing constraints and power constraints.
4. (Optional) Provide physical constraints as described in [Using Floorplan Physical Constraints](#).

You can provide floorplan constraints that determine core area and shape, port location, macro location and orientation, voltage areas, placement blockages, placement bounds, and so on. Using floorplan constraints ensures more accurate placement area and improved timing correlation with the post-place-and-route design.

5. (Optional) Visually verify the floorplan.

Use the GUI layout window to visually verify that your pre-synthesis floorplan is laid out according to your expectations. The layout window can display floorplan constraints read in with the `extract_physical_constraints` command or with Tcl floorplanning commands. You need to link all applicable designs and libraries to obtain an accurate floorplan.

For more information about using the GUI to view physical constraints, see the “Viewing the Floorplan” topic in Design Vision Help.

6. Check the design and libraries to make sure the necessary data is available by using the `compile_ultra -check_only` command.

The `-check_only` option reports potential problems that could cause the tool to stop during the `compile_ultra` run or to produce unsatisfactory correlation with your physical implementation.

For details, see [Checking Designs and Libraries Before Synthesis](#).

7. Perform a test-ready and clock-gating compile of the subblock by using the `compile_ultra -scan -gate_clock` command.

Use the `-scan` option to perform test-ready compilation. During test-ready compilation, the tool employs the scan insertion design-for-test technique, which replaces regular flip-flops with flip-flops that contain testability logic. Test-ready compilation reduces iterations and design time by accounting for the impact of the scan implementation during the logic optimization process.

Use the `-gate_clock` option to implement clock gating in the design according to options set by the `set_clock_gating_style` command. For more information about the `set_clock_gating_style` command, see the *Power Compiler User Guide*.

8. Specify the design-for-test (DFT) configuration and run the `insert_dft` command as described in [Test Synthesis in Topographical Mode](#).

You use the `insert_dft` command to insert scan chains, if the subblock is to be included in the top-level scan chain.

9. Perform an incremental compile by using the `compile_ultra -scan -incremental` command.
10. Use the `uniquify_naming_style` variable to specify the unique naming convention to be used by the `uniquify` command, and use the `uniquify -force` command to uniquify the subblock. This prevents naming conflicts during top-level design integration.
11. Run the `change_names -rules verilog -hierarchy` command to apply the Verilog naming rules to all the design objects prior to writing out the design data files.
12. Write the SCANDEF and core test language (CTL) model files by running the following commands:

```
dc_shell-topo> write_scan_def -output design.scandef  
dc_shell-topo> write_test_model -format ctl -output design.ctl
```

The SCANDEF file contains information about scan ordering requirements, and the .ctl file contains information about the DFT logic. The information in these files is also stored in the .ddc file, to be used by top-level DFT insertion and reordering. However, the ASCII files can be used for reference.

Note:

You must execute the `write_scan_def` command to annotate the scan ordering information onto the current design.

The steps you perform from this point forward depend on whether you are creating a full .ddc netlist for the subblock in topographical mode, generating a block abstraction for the subblock in topographical mode, or continuing to implement the subblock in IC Compiler:

- If you are creating a full .ddc netlist for the subblock in topographical mode, save the mapped subblock in .ddc format by using the `write_file -format ddc` command. During top-level synthesis, read in the mapped subblock or add it to the `link_library` variable. See [Compiling the Design at the Top Level](#).

- If you are generating a block abstraction for the subblock in topographical mode, continue with the steps in [Generating a Block Abstraction for the Subblock in Topographical Mode](#).
- If you are continuing to implement the subblock in IC Compiler, continue with the steps in [Generating a Block Abstraction for the Subblock in IC Compiler](#).

Generating a Block Abstraction for the Subblock in Topographical Mode

To generate a block abstraction for the subblock in topographical mode, continue with the following steps after completing the steps in [Compiling the Subblock](#):

1. Generate the block abstraction by running the `create_block_abstraction` command.
The `create_block_abstraction` command identifies the interface logic of the current design and annotates the design in memory with the interface logic.
2. Save the block abstraction by using the `write_file` command. For example,

```
dc_shell-topo> write_file -hierarchy -format ddc -output my_file.mapped.ddc
```

It is important that you use the `write_file` command to save the block abstraction immediately after you create it. The `write_file` command writes the complete design, including the block abstraction information, into a single .ddc file.

To use this block as a block abstraction or as a physical hierarchy .ddc at the top level, continue with the steps in [Compiling the Design at the Top Level](#).

Generating a Block Abstraction for the Subblock in IC Compiler

You can continue to implement the subblock in IC Compiler. To create a block abstraction for top-level integration when the block is completed in IC Compiler, perform the following steps. For more information about IC Compiler commands, see the IC Compiler documentation.

1. Write out the block-level design in Design Compiler using the `write_file -format ddc` command.
2. In IC Compiler, set up the design and Milkyway libraries.
3. Read in the .ddc format of the mapped subblock and block SCANDEF information generated in topographical mode.
4. Create the floorplan by using IC Compiler commands or read in floorplan information from a DEF file by using the `read_def` command.
5. Run the `place_opt -optimize_dft` command to perform placement and placement-aware scan reordering.
6. Run the `clock_opt` command to perform clock tree synthesis.

7. Run the `route_opt` command to route the design.
8. Run the `create_block_abstraction` command to create the block abstraction.
9. Run the `save_mw_cel` command to create the CEL view.

Compiling the Design at the Top Level

To compile the design at the top level,

1. Specify the logic and physical libraries as described in [Setting Up and Working With Libraries](#).
If you are using a block abstraction created by IC Compiler, you need to add the Milkyway design library that contains the block abstraction CEL view to the Milkyway reference library list at the top level.
2. For block abstractions created in Design Compiler topographical mode or IC Compiler, use the `set_top_implementation_options` command to specify which blocks should be integrated with the top-level design as block abstractions.
 - Block abstraction created in topographical mode (.ddc)
 - a. Use the `-block_references` option with the `set_top_implementation_options` command before opening the block. If you do not use this command, the full block netlist will be loaded.
 - b. (Optional) Set the `-optimize_block_interface` option to `true` to enable transparent interface optimization, which optimizes the interface logic within the blocks to achieve faster timing closure.
 - c. (Optional) Use the `-block_update_setup_script` option, which specifies the setup file to be sourced during the block update of the specified block. The file name should not include relative paths.
This setting is useful for passing any variable settings for the block update process.
 - d. (Optional) Use the `-block_update_cmd_script` option, which specifies the script file with the commands to be run to update the specified block. The file name should not include relative paths.
If this option is not specified, the tool automatically runs the `extract_rc -estimate` command for the block update.

For example,

```
dc_shell-topo> set_top_implementation_options \
    -block_references ${BLOCK_ABSTRACTION_DESIGNNS} \
    -optimize_block_interface true
```

- Block abstraction created in IC Compiler (CEL)

Use the `set_top_implementation_options` command to set the top-level design options for linking. If you do not use this command, the tool cannot link to the IC Compiler block abstraction.

The block abstraction is automatically loaded when you link the top-level design if the Milkyway design library that contains the block abstraction CEL view was added to the Milkyway reference library list at the top level.

Note:

A block abstraction can contain register-to-register paths that are entirely within the block. These paths are the result of combinational logic in the block abstraction that is shared by input-to-register and register-to-output paths. Paths entirely within blocks are optimized during block-level optimization, and ignoring them during top-level optimization enables transparent interface optimization to focus on other paths. For information about ignoring these register-to-register paths, see [Ignoring Timing Paths That Are Entirely Within Block Abstractions](#).

3. Read in the top-level design (Verilog, VHDL, netlist, or .ddc) file.
4. Read in the mapped subblock.

The subblocks can be any of the following:

- Complete netlist in .ddc format created in Design Compiler topographical mode.
If `set_top_implementation_options` command options are not set for the block, the full netlist .ddc is read in for the block.
- Block abstraction in .ddc format created in Design Compiler.
You must use the `set_top_implementation_options` command before the .ddc file is loaded in order to load the block as a block abstraction. Only the interface logic is loaded, not the full block.
- Block abstraction created in IC Compiler.
You must use the `set_top_implementation_options` command before linking the top-level design. The block abstraction is automatically loaded when you link the top level if the Milkyway design library that contains the block abstraction CEL view was added to the Milkyway reference library list at the top level.

Note:

You can add the .ddc file to the `link_library` variable instead of reading it in directly. If you do this, the tool automatically loads it when it is linked to the top level.

5. In topographical mode, set the current design to the top-level design.
6. Use the `set_physical_hierarchy` command for complete topographical netlists in .ddc format to specify that the subblock should be treated as a physical subblock. When you do this, top-level synthesis preserves both the logic structure and cell placement inside

the block. If you do not want to treat a topographical netlist as a physical block, omit the `set_physical_hierarchy` command. When you do this, the block is not treated as a physical block and the cells inside the block undergo virtual placement during top-level synthesis.

The `set_physical_hierarchy` and `set_dont_touch` commands do not need to be used on block abstractions. Block abstractions will automatically have these settings to preserve the logical structure and placement.

7. Use the `link` command to link the subblocks specified in the previous step to the top level.
8. Apply top-level timing and power constraints.
9. (Optional) Provide physical constraints as described in [Using Floorplan Physical Constraints](#).

You can specify locations for the subblock by using the `set_cell_location` command or by using the `extract_physical_constraints` command to extract physical information from the Design Exchange Format (DEF) file.

10. (Optional) Visually verify the floorplan.

Use the Design Vision layout window to visually verify that your pre-synthesis floorplan is laid out according to your expectations. The layout view automatically displays floorplan constraints read in with `extract_physical_constraints` or read in with Tcl commands. You need to link all applicable designs and libraries to obtain an accurate floorplan.

For more information about using the GUI to view physical constraints, see the “Viewing the Floorplan” topic in Design Vision Help.

11. Run the `compile_ultra -scan -gate_clock` command.

The `-scan` option enables the mapping of sequential cells to appropriate scan flip-flops. The `-gate_clock` option enables clock-gating optimization.

12. Apply the DFT configuration and run the `insert_dft` command to insert scan chains at the top level, followed by the `compile_ultra -scan -incremental` command.

13. Write out the SCANDEF information by using the `write_scan_def` command.

Top-level scan chain stitching connects scan chains up to the interfaces of test pins of the subblock. In the top-level SCANDEF, you can control whether physical blocks are described using the “BITS” construct or as complete scan chains.

If you are using Design Compiler block abstractions or physical hierarchy blocks, you use the `write_scan_def` command with the `-expand_elements` option to write out the SCANDEF information. You do not maintain the test hierarchy partition during top-level physical implementation in IC Compiler. The SCANDEF must directly reference the scan

leaf objects of the physical block instead of the BITS construct to run the IC Compiler top-level flat flow. For example,

```
write_scan_def -expand_elements block_instance
```

If you are using IC Compiler block abstractions, you maintain the test hierarchy partition during top-level physical implementation. Therefore, the top-level SCANDEF is written out with the block containing the BITS construct; you use the `write_scan_def` command without the `-expand_elements` option to write out the SCANDEF information.

If your flow contains both Design Compiler and IC Compiler models, you should only specify the `-expand_elements` option for Design Compiler block abstractions or physical hierarchies. Omit the blocks modeled by IC Compiler from the block list when you specify the `-expand_elements` option.

14. Write out the top-level netlist by using the `write_file -hierarchy -format ddc` command.

When you do this, all physical blocks are removed automatically. If you enabled transparent interface optimization, you need to save the updated blocks. Use the `write_file` command to save each optimized block abstraction to a separate .ddc file.

You can use the following script to save each optimized block abstraction to a .ddc file:

```
foreach design ${BLOCK_ABSTRACTION_DESIGNS} {
    write_file -hierarchy -format ddc \
        -output ${design}.mapped_tio.ddc \
        ${design}
}
```

The `write_file` command merges the changes with the original .ddc design and writes the complete block as a new .ddc file.

Reducing Runtime

The Design Compiler tool allows you to set numerous commands, variables, and options to give you flexibility and control over the Design Compiler flow. However, sometimes these settings are runtime intensive. You can use the `compile_prefer_runtime` command to override these runtime-intensive settings and use the settings described in [Table 19-3](#) to reduce runtime.

Use the `compile_prefer_runtime` command only if runtime is a concern. The command disables runtime-intense optimizations, and therefore it might impact QOR. The effect of the setting changes depends on the design. You might not see runtime improvements on all designs.

When you run the `compile_prefer_runtime` command in topographical mode,

- The tool overrides the runtime intensive settings described in [Table 19-3](#). User-specified settings overridden by the command during optimization are reverted back to their original settings at the end of the optimization flow.

The `compile_prefer_runtime` command settings take effect when you run the `compile_ultra`, `compile_ultra -incremental`, or `optimize_netlist -area` command.

- The tool reports the changed commands and variables and their settings in the log file under the UIO-232 message ID.
- Zroute is disabled for the `report_congestion` command.
- The `placer_max_cell_density_threshold` variable value is as follows:
 - If the design utilization is greater than 50%, the tool sets the default value of -1.
 - If the design utilization is low (less than 50%), the tool sets a value of 0.7.
 - If the design utilization is greater than 50% and you set a value less than 0.5, the tool overrides your setting and sets a value of -1.

Table 19-3 Values That Are Changed When You Use the `compile_prefer_runtime` Command

Command, variable, or setting	Default value (if not user-defined)	Value when you run the <code>compile_prefer_runtime</code> command
<code>set_ahfs_option -global_route</code>	false	false
<code>placer_enable_enhanced_router</code>	false	false
<code>placer_congestion_effort</code>	auto	auto
<code>placer_reduce_high_density_regions</code>	false	false
<code>placer_channel_detect_mode</code>	false	false
<code>spg_congestion_placement_in_incremental_compile</code>	false	false

Table 19-3 Values That Are Changed When You Use the compile_prefer_runtime Command

Command, variable, or setting	Default value (if not user-defined)	Value when you run the compile_prefer_runtime command
placer_max_cell_density_threshold	-1	<ul style="list-style-type: none"> -1 if utilization > 50% 0.7 if utilization < 50% Overridden to 0.7 if the user-specified setting is < 0.5 on low-density designs Overridden to -1 if the user-specified setting is < 0.5 on designs with utilization greater than 50%
compile_final_drc_fix	none	none
spg_enhanced_timing_model	false	false
spg_enable_via_resistance_support	false	false
compile_enable_register_merging	true	false
timing_enable_multiple_clocks_per_reg	true	Disabled if over 20 clocks per pin
spg_incremental_enhanced_placement	false	false
Layer optimization	Automatic layer optimization enabled	If you specify the <code>-no_auto_layer_optimization</code> option with the <code>compile_ultra</code> command to disable automatic layer optimization, the tool overrides the user setting and enables auto layer optimization.

See Also

- [Reporting Runtime Issues Related to SDC Constraints](#)

Handling Unsupported Commands, Options, and Variables

In topographical mode, if Design Compiler encounters an unsupported command or variable in a script, it issues an error message; however, the script *continues*. For example,

```
Error: Command set_wire_load_mode is not supported in DC Topographical mode. (OPT-1406)
```

Variables that are read-only in topographical mode are indicated as read-only variables in the error message.

If Design Compiler in topographical mode encounters any unsupported command options (except wire load model options), it issues an error message and the script stops executing. Wire load model options are ignored, and the script continues.

Check your scripts and update them as needed.

Using the Design Compiler Graphical Tool

The Design Compiler Graphical tool extends topographical technology by optimizing multicorner-multimode designs, reducing routing congestion, and improving both area correlation with IC Compiler and runtime in IC Compiler with Synopsys physical guidance. In addition, Design Compiler Graphical allows you to create and modify floorplans using floorplan exploration.

Running Design Compiler Graphical requires a DC Ultra license and a Design Compiler Graphical license.

See Also

- [Using Design Compiler Graphical](#)

20

Using Design Compiler Graphical

The Design Compiler Graphical tool extends topographical technology by

- Optimizing multicorner-multimode designs
- Reducing routing congestion
- Improving correlation with IC Compiler and IC Compiler II
- Improving runtime in IC Compiler and IC Compiler II by using Synopsys physical guidance
- Allowing you to create and modify floorplans using floorplan exploration

To learn how to use the Design Compiler Graphical tool, see

- [Using Synopsys Physical Guidance](#)
- [Floorplan Exploration Floorplanning With IC Compiler](#)
- [Floorplan Exploration Floorplanning With IC Compiler II](#)
- [Working With Via Ladders](#)
- [Working With Must-Join Pins](#)
- [Inbound Cell Optimization](#)
- [Optimizing Multicorner-Multimode Designs](#)

Using Synopsys Physical Guidance

Synopsys physical guidance technology enables Design Compiler Graphical to perform enhanced placement that is consistent with the IC Compiler `place_opt` command functionality and enhanced post-placement delay optimization in order to provide a better optimized starting point for physical implementation. The placement information is passed to the IC Compiler `place_opt` command and is used as seed placement to guide physical implementation of the design. As a result, placement is aligned between Design Compiler and IC Compiler, improving runtime, quality of results (QoR), correlation, and routability. In addition, physical guidance enables congestion optimization, which reduces routing-related congestion.

Using the physical guidance flow, IC Compiler no longer needs to create a coarse placement by running commands such as `create_placement`, `remove_buffer_tree`, or `psynopt`.

The following topics describe the physical guidance functionality:

- [Physical Guidance Overview](#)
- [Reducing Routing Congestion](#)
- [Controlling Placement Density](#)
- [Specifying Design Constraints and Power Settings](#)
- [Using Layer Optimization to Increase the Accuracy of Net Delay Estimation](#)
- [Enabling Net Promotion to Multiple Layers](#)
- [Working With Nondefault Routing Rules](#)
- [Enabling the Physical Guidance Flow](#)
- [Enabling the Physical Guidance Incremental Flow](#)
- [Using Physical Guidance in IC Compiler](#)
- [Using the Design Compiler Graphical and IC Compiler Hierarchical Flow](#)
- [Improved Timing Correlation Between Design Compiler Graphical and IC Compiler II](#)
- [Incremental ASCII Flow With a Third-Party DFT Flow Example](#)
- [Exporting the Design](#)
- [Reporting Physical Guidance Information](#)
- [Physical Guidance Limitations](#)

Physical Guidance Overview

The physical guidance flow is jointly supported in Design Compiler and IC Compiler to improve runtime, correlation, and routability and to reduce routing-related congestion. Physical guidance significantly speeds up IC Compiler runtime because IC Compiler uses the physical guidance information from Design Compiler Graphical as its starting point and therefore goes through fewer placement steps.

To ensure good QoR and correlation between Design Compiler and IC Compiler, you need to use consistent design and tool setup between Design Compiler Graphical and IC Compiler. You should use the same sets of logic and physical libraries to drive both tools and use consistent library settings. Also, use consistent design goal specifications, such as timing constraints and design rule settings to drive logical and physical implementation.

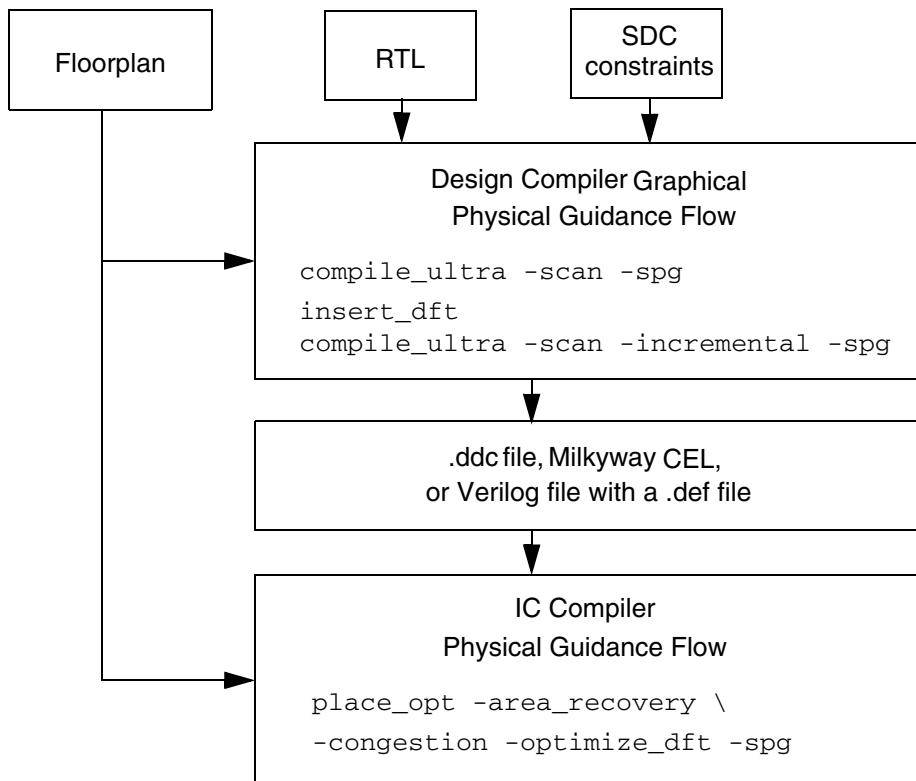
You enable the physical guidance flow by using the `-spg` option with the `compile_ultra` command in Design Compiler Graphical and by using the `-spg` option with the `place_opt` command in IC Compiler. The `-spg` option works seamlessly with the existing `compile_ultra` options in Design Compiler and with the `place_opt` options in IC Compiler.

Physical guidance performs the following functions:

- In Design Compiler Graphical, the `-spg` option provides the following:
 - Further refined placement that is consistent with the `place_opt` command in IC Compiler
 - Enhanced post-placement delay optimization to provide a better optimized starting netlist for physical implementation
 - Optimizes the design for congestion to improve routability
- You can save the placement information derived by Design Compiler Graphical in binary format, in a `.ddc` file or a Milkyway CEL view, or you can save it in a `.def` file.
- In IC Compiler, the `-spg` option enables the `place_opt` command to use the Design Compiler Graphical placement as its seed placement. The `place_opt` command is enhanced to use topographical mode placement as the starting point in physical guidance mode.

[Figure 20-1](#) shows an example of a typical physical guidance flow.

Figure 20-1 Physical Guidance Flow Example



The physical guidance flow in IC Compiler supports all the compile options used by Design Compiler in topographical mode, such as `-scan`, `-clock_gate`, and `-incremental`. In addition, the flow supports the `-congestion` option, which reduces routing-related congestion. Note that the `-congestion` option is not needed in Design Compiler because Design Compiler reduces routing-related congestion by default with the `-spg` option.

The physical guidance flow in IC Compiler also supports the multicorner-multimode based flow, the multivoltage-UPF flow, the power flow, the DFT flow, and the hierarchical flow.

Use the following guidelines to maintain good QoR and correlation between Design Compiler and IC Compiler when you use the physical guidance flow:

- Use consistent design and tool setup between Design Compiler topographical mode and IC Compiler. That is, you should use the same sets of logic and physical libraries to drive both tools and use consistent library settings, including the `dont_use` list. Also, use consistent design goal specifications, such as timing constraints and design rule settings to drive logical and physical implementation.

You should load the same sets of the following files in both tools:

- Logic library (.db)
- Milkyway reference library
- Technology file (.tf)
- Mapping file (maps the technology file layer names to TLUPlus layer names)
- Min and max TLUPlus files for net parasitics

Make sure that each standard cell and macro in the logic libraries has a corresponding abstract physical view in the Milkyway reference library. Otherwise, the cells are treated as `dont_use` cells for mapping and optimization.

In addition, you should use consistent library-specific attribute settings with IC Compiler. For instance, your library cell `dont_use` list, `dont_touch` settings, and design rule settings (maximum allowable fanout load and transition time) should be aligned with the settings used in IC Compiler.

- Apply a complete set of physical constraints by using a floorplan DEF file or Tcl commands before the first `compile_ultra` command step in Design Compiler. Also, reapply the same sets of physical constraints in IC Compiler to preserve the placement consistency for physical guidance.

The same DEF file must be used by both tools. The DEF file contains floorplan information that must be consistent between tools. The tools issue various errors and warnings if your constraints, floorplan information, and libraries are not consistent.

- Since the physical guidance information is restored when you run the IC Compiler `place_opt` command, it is recommended that you run the `place_opt -spg` command as the first placement-related command in IC Compiler so that Design Compiler placement is restored properly. Using other placement-related commands (such as `create_placement`, `remove_buffer_tree`, and so on) might degrade correlation and can lead to the loss of some physical guidance information.

Reducing Routing Congestion

Designs are considered congested when the wires needed to connect design components do not fit in the space available to put the wires. The following sections describe how physical guidance reduces routing congestion:

- [Routing Congestion Overview](#)
- [Reducing Congestion in Highly Congested Designs](#)
- [Enabling MUX Congestion Optimization](#)
- [Optimizing High-Pin-Density Cells](#)
- [Reducing Congestion in Incremental Compile](#)
- [Reducing Congestion by Optimizing RTL Structures](#)
- [Specifying Block-Level Congestion Optimization](#)
- [Controlling Congestion Optimization](#)
- [Reporting Congestion](#)
- [Viewing Congestion With the Design Vision Layout Window](#)
- [Congestion Map Calculations](#)
- [Reducing Congestion in the Floorplan](#)

Routing Congestion Overview

Wire-routing congestion in a design occurs when the number of wires traversing a particular region is greater than the capacity of the region. If the ratio of usage-to-capacity is greater than 1, the region is considered to have congestion problems. Congestion can be caused by standard cells or the floorplan. Typically, standard cell congestion is caused by the netlist topology, for example, large multiplexer trees, large sums of products (ROMs), test decompression logic, or test compression logic. Floorplan congestion can be caused by macro placement or port location.

If Design Compiler passes a congested design to IC Compiler, it might be difficult and time consuming for IC Compiler to reduce the congestion. Design Compiler can reduce congestion more easily than IC Compiler because Design Compiler can optimize RTL structures. Traditionally, to minimize congestion related to the netlist topology, you would use scripting solutions to restrict technology mapping tools, such as Design Compiler and IC Compiler, from choosing library cells that you perceive as poor for congestion. But these solutions do not significantly improve congestion; instead, they often create poor timing or area results.

Design Compiler Graphical predicts wire-routing congestion hot spots during RTL synthesis and uses its interactive capability to visualize the design's congestion. When the tool determines that the design has congestion problems, it minimizes congestion through congestion optimization, resulting in a better starting point for layout.

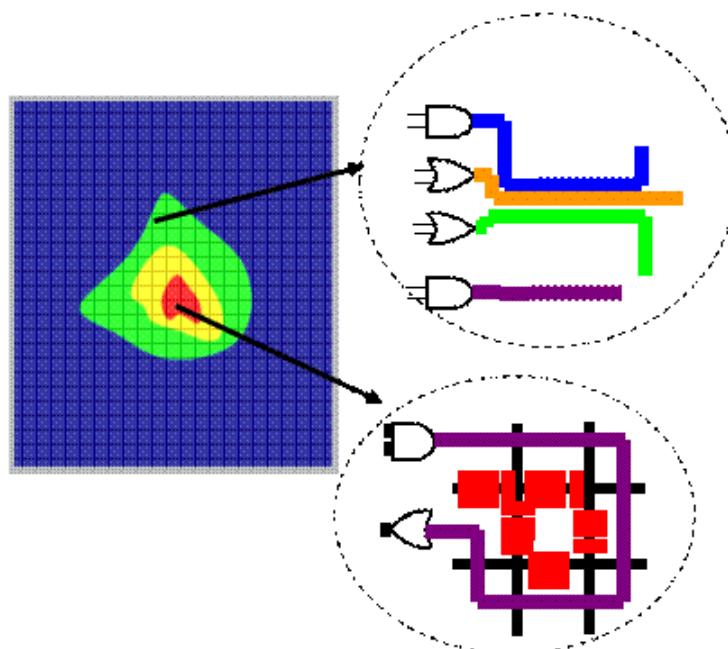
Design Compiler uses the Synopsys virtual routing technology to predict wire-routing congestion. In this technology, congestion is estimated by dividing the design into a virtual grid of global routing cells followed by a global route to count the number of wires crossing each grid edge. The capacity and size of each global routing cell is calculated by using the physical library information. A cell is considered congested if the number of wires passing through it is greater than the number of available tracks.

Design Compiler predicts the congestion number by subtracting the maximum wires allowed for a particular edge direction from the total number of wires crossing per edge. Any number over zero is a congestion violation. This calculation is represented by the following equation:

$$\text{Congestion} = (\text{number of wires}) - (\text{maximum wires allowed})$$

[Figure 20-2](#) shows how Design Compiler Graphical predicts congestion. The red areas indicate congestion.

Figure 20-2 Predicting Congestion Using Virtual Grid Route



The tool considers floorplan-related physical constraints when it estimates routing congestion. In particular, the wiring keepout physical constraint ensures that the tool is aware of areas that must be avoided during routing. This physical constraint helps to achieve consistent congestion correlation with layout.

Reducing Congestion in Highly Congested Designs

For highly congested designs, you can use Zroute-based congestion-driven placement to enable more accurate, congestion-aware placement instead of using congestion estimation based on virtual routing, which estimates congestion based on wire-routing predictions. When you set the `placer_enable_enhanced_router` variable to `true`, Zroute-based congestion estimation is enabled on designs that meet the internal congestion threshold. Any designs that are not congested enough to meet the threshold are estimated by virtual routing (the default).

Design Compiler Graphical and IC Compiler use the same internal congestion thresholds. As a result, Zroute-based congestion-driven placement provides better optimization and congestion correlation between Design Compiler Graphical and IC Compiler than virtual routing. However, using Zroute congestion-driven placement might increase runtime and affect area and timing. Therefore, use it only on highly congested designs.

To control which congestion estimator is used during congestion-driven placement, use the `placer_congestion_effort` variable. For the `placer_congestion_effort` variable settings to take effect, you must first set the `placer_enable_enhanced_router` variable to `true`.

When the `placer_congestion_effort` variable is set to `auto` (the default) and the `placer_enable_enhanced_router` variable is set to `true`, the tool first runs a fast internal estimator to determine how much congestion there is in the placement. If the tool determines that there is not enough congestion to meet the internal congestion threshold, it uses the fast estimator's placement map to drive congestion-driven placement. If the congestion meets the internal threshold, the tool uses the Zroute global router to perform a more accurate congestion estimation and uses that placement map to drive congestion-driven placement.

To enable Zroute-based congestion estimation even if the design does not meet the internal congestion threshold, set the `placer_congestion_effort` variable to `medium`.

See Also

- [Performing Automatic High-Fanout Synthesis](#)

Provides information about using Zroute-based global buffering to reduce congestion along narrow channels in macro-intensive designs

Enabling MUX Congestion Optimization

You can implement more MUXes in a design by using the `compile_prefer_mux` variable in the Design Compiler Graphical tool. Enabling the variable can lead to lower congestion for designs that have congestion due to multiplexing logic in the RTL.

Enable the `compile_prefer_mux` variable by setting it to `true` before running the initial compile:

```
dc_shell-topo> set_app_var compile_prefer_mux true
dc_shell-topo> compile_ultra -spg
```

The `compile_prefer_mux` variable is supported only in the initial compile step, not in incremental compile. You can use the variable as an alternative to the `map_to_mux` attribute, which requires you to manually investigate where the attribute needs to be applied.

Optimizing High-Pin-Density Cells

You can replace high-pin-density cells in regions with high utilization to improve routability with minimal impact to QoR. The Design Compiler Graphical tool divides the design into smaller areas and computes the regional utilization of these regions. When the utilization of a region is above the user-specified threshold, the high-pin-density cells in these regions are replaced with equivalent low-pin-density electrically equivalent (EEQ) cells that can be routed more easily.

Note:

The library must contain EEQ cells.

To replace high-pin-density cells in regions with high utilization,

1. Enable high-pin-density cell optimization.

```
dc_shell-topo> set_app_var \
compile_high_pin_density_cell_optimization true
```

The variable is off (`false`) by default.

2. Specify the regional utilization threshold.

```
dc_shell-topo> set_app_var \
compile_high_pin_density_cell_optimization_utilization_threshold 0.3
```

The utilization threshold is 0.5 by default.

3. Run synthesis in physical guidance mode.

```
dc_shell-topo> compile_ultra -spg -scan
```

During compile, the Design Compiler Graphical tool replaces the high-pin-density cells in high-utilization regions with equivalent low-pin-density electrically equivalent cells.

Reducing Congestion in Incremental Compile

To enable congestion-driven placement during incremental compile, set the `spg_congestion_placement_in_incremental_compile` variable to `true`. The variable improves congestion while preserving quality of results. The variable also improves congestion correlation results, comparing post-incremental compile results in Design

Compiler and post placement optimization results in IC Compiler. You do not need to enable Zroute to use the variable. If Zroute is enabled, it runs congestion-driven placement in incremental compile by default.

Reducing Congestion by Optimizing RTL Structures

Congestion optimization is primarily beneficial for designs in which RTL logic structures result in congestion. Minimizing the congestion for such designs often requires significant changes in the netlist topology, which cannot be done during place and route. Such large changes are best done early in the flow during synthesis.

You can report pre-synthesis congestion analysis for the current design by using the `analyze_rtl_congestion` command. The command evaluates the RTL for potential congestion issues and reports the issues so you can resolve them early in the design flow. The report includes the line numbers in the RTL where the issues occurred to help you identify where RTL changes might be needed.

The following example is an excerpt from an RTL congestion report:

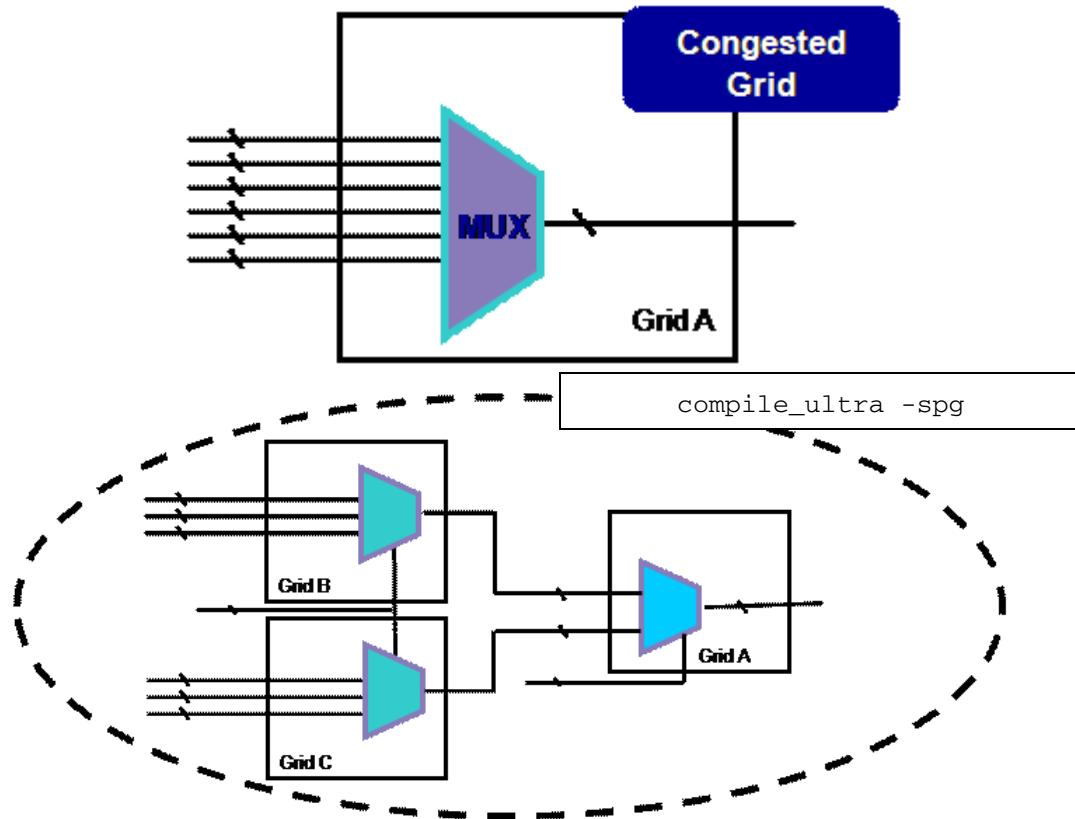
```
dc_shell-topo> analyze_rtl_congestion
*****
Report : RTL congestion
Design : my_design
...
...
*****
#####
# Large MUX details
#####
-----
Structure   Cell name   Size (width)   Line No   RTL File
-----
MUX_OP      C10567     16 x 1 (163)    83        /u/9000560869.v
MUX_OP      C10566     16 x 1 (163)    82        /u/9000560869.v
...
...
```

Design Compiler Graphical automatically reduces congestion when you use the `-spg` option with the `compile_ultra` command. The tool produces logic structures that are better suited for a placement that minimizes congestion and improves routability.

In addition, you can specify options, such as the amount of resources available for a given layer, by using the `create_route_guide` command. For more information, see [Controlling Congestion Optimization](#).

Figure 20-3 shows an example of how the tool optimizes an RTL structure to minimize congestion. In this example, the tool optimizes the single large multiplexer into three smaller multiplexers, which reduces routing congestion by restructuring the RTL.

Figure 20-3 Optimizing RTL Structures to Minimize Congestion



Specifying Block-Level Congestion Optimization

By default, Design Compiler optimizes the complete design for congestion. However, you can also enable or disable congestion optimization for a specific design or instance by using the `set_congestion_optimization` command. The design must be loaded in memory. When you specify the command on an instance, it is hierarchical and applies to all subinstances.

In the following example, congestion optimization is disabled for cell `sub_a`:

```
dc_shell-topo> set_congestion_optimization [get_cells top/sub_a] false
```

To specify congestion optimization on only one instance, first disable the optimization on all designs by setting the `set_congestion_optimization` command to `false` on the top-level design. Then, enable optimization on the instance, as shown:

```
dc_shell-topo> set_congestion_optimization [current_design] false
```

```
dc_shell-topo> set_congestion_optimization [get_cells top_level/i_sub]\ntrue
```

After you have enabled or disabled the optimization as needed, you must run the `compile_ultra` command with the `-spg` option. The `-spg` option is required for the `set_congestion_optimization` settings to be applied.

Controlling Congestion Optimization

Specifying congestion optimization options helps achieve consistent congestion correlation between Design Compiler in topographical mode and IC Compiler and ensures that both tools see the same congestion setup.

You can control congestion optimization by using the `create_route_guide` command. For example, you can specify how much routing resource is available to be used for specific metal layers. The following example specifies 85% availability on layers M2 and M3:

```
dc_shell-topo> create_route_guide -coordinate {0.0 0.0 100.0 100.0} \n      -horizontal_track_utilization 0.85 \\n      -vertical_track_utilization 0.85 \\n      -track_utilization_layers {M2 M3}
```

The route guides with utilization are stored in the .ddc file, but they are not passed to IC Compiler. They must be reapplied in IC Compiler.

The tool minimizes congestion by moving cells from congested regions to uncongested regions. To specify the maximum utilization the tool allows as cells migrate into uncongested areas, use the `set_congestion_options` command with the `-max_util` option. For example, setting a value of 0.9 allows the tool to place cells in an area with up to 90 percent utilization.

To remove congestion options, use the `remove_congestion_options` command.

See Also

- [Reporting Congestion](#)

Reporting Congestion

To report details about congestion, run the `report_congestion` command. This executes global routing, which generates a log that shows statistical information for global routing and the intermediate congestion results at each global routing phase. The final text-based report provides a quick estimate of the congestion status of the design. You can use this report to assess the severity of congestion in the design.

In cases where the design is reported as significantly congested, you can further analyze congestion by generating a congestion map. A congestion map is a graphical representation of the congested areas that you can view and analyze in the Design Vision layout window.

For an overview, see [Viewing Congestion With the Design Vision Layout Window](#). For more details, see the *Design Vision User Guide* or Design Vision Help.

To specify the global routing effort level when the tool generates a global route congestion map, use the `-effort` option with the `report_congestion` command. The option allows you to control the tradeoff between runtime and accuracy. The lower effort has a faster runtime but results in a more pessimistic congestion number. The higher effort has a better congestion result but requires a longer runtime. The default is `medium`.

[Example 20-1](#) shows a sample output generated by the `report_congestion` command.

Example 20-1 The report_congestion Command Output

```
*****
Report : congestion
Design : my_design
...
*****
Both Dirs: Overflow = 4225 Max = 10 (1 GRCs) GRCs = 2397 (0.75%)
H routing: Overflow = 675 Max = 4 (5 GRCs) GRCs = 503 (0.16%)
V routing: Overflow = 3550 Max = 10 (1 GRCs) GRCs = 1894 (0.59%)
```

You interpret this report as follows:

- *Overflow* is the total number of wires in the design global routing cells that do not have a corresponding track available. The following equation shows how overflow is measured:

$$\text{Overflow} = \sum_{i=1}^{\text{Max}} \text{violation}(i) \times \text{GRCs with violation}(i)$$

- *Max* corresponds to the highest number of over-utilized wires in a single global routing cell. It is the worst-case violation and the number of global routing cells that have the violation. The report shows that the worst-case violation is five and it occurs for ten global routing cells in the design.
- *GRC* is the total number of over-congested global routing cells in the design. The report indicates that there are 3401 violating GRCs in the design, which account for 0.59% of the design.

Viewing Congestion With the Design Vision Layout Window

You can identify areas of high congestion in your design by viewing the congestion map in the Design Vision layout window. Visually examining congested areas in your design allows you to determine whether the design is routable and identify the causes of the congestion if the design is not routable.

You can inspect the RTL code for cells that are causing congestion by cross-probing cells from the layout view or congestion map to the RTL browser. You can cross-probe cells from the list of cells in congested areas or all cells in highly congested areas. After identifying the RTL code that is causing congestion, open the RTL source file and modify the RTL code as needed. For details, see the Design Vision Help.

You can display or hide the congestion map at any time when a layout view is open in the layout window. In some cases, it can be useful to open the layout window and display the congestion map or visual mode images with your script. You can also save an image of the layout in a file.

To generate a congestion map, you must load a .ddc netlist synthesized in topographical mode. For information about displaying the congestion map and viewing cells in congested areas, see the *Design Vision User Guide* and Design Vision Help.

See Also

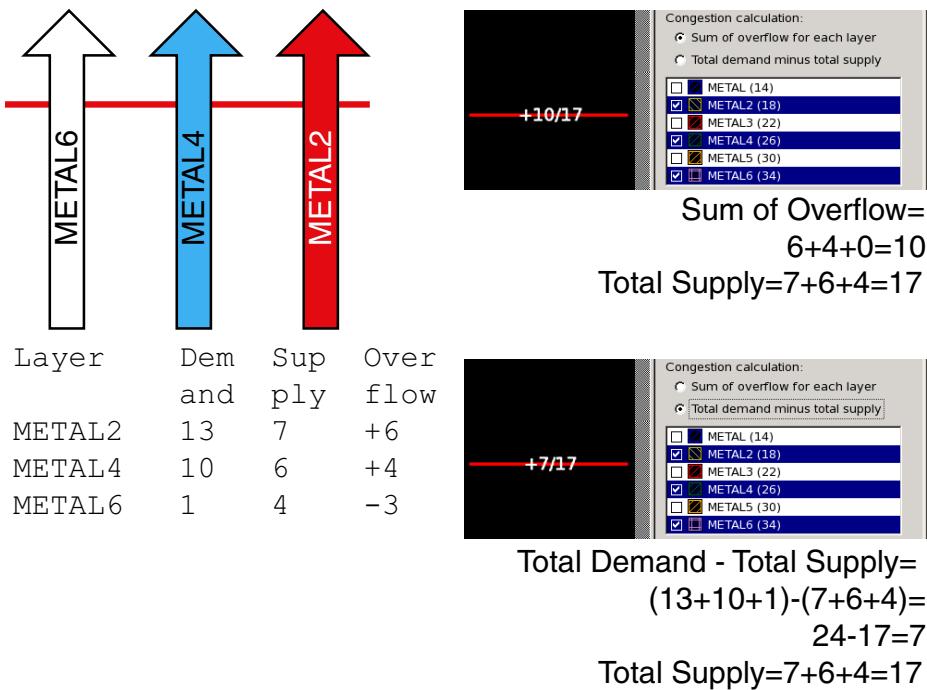
- [Congestion Map Calculations](#)
- The “Cross-Probing the RTL for Cells or Paths” topic in Design Vision Help for information about identifying RTL code that causes congestion
- The “Cross-Probing Cells in Congested Areas” topic in Design Vision Help

Congestion Map Calculations

You can control how Design Compiler in topographical mode calculates and reports global route congestion (GRC) by selecting one of the following congestion calculation options on the Map Mode panel of the GUI. Both congestion calculation options are for reporting purposes only and do not affect optimization.

- To calculate congestion as the sum of the overflow for each layer, select the new “Sum of overflow for each layer” option and click Apply. This option is selected by default.
- To calculate congestion as the total demand minus the total capacity, select the “Total demand minus total supply” option and click Apply.

The following figure shows a calculation example for both modes. The example explains one direction through a global route cell in a design with congestion.

Figure 20-4 Congestion Example

In this example, Layer METAL2 is heavily congested, with a demand of 13 routing tracks and a supply of only 7 tracks, for an overflow of 6. Layer METAL4 is moderately congested with an overflow of 4. Layer METAL6 is not congested and contains an underflow of 3. Other layers are not used in the congestion calculation in this example.

The sum of overflow calculation is the default calculation mode and considers only the overflow for each layer. In this example, the overflow on METAL2 is 6, the overflow on METAL4 is 4, for a total sum of overflow of $6+4=10$. No overflow exists on METAL6 and this layer is not used in the calculation, even though it is selected. In a region that contains only underflow, the tool sums the underflow for each selected layer.

The total demand minus total supply calculation sums the total demand and subtracts the total supply. In this example, the total demand is 24, the total supply is 17, and the result is $24-17=7$. Note that the total demand minus total supply calculation produces a more optimistic congestion result in regions that contain both overflow and underflow.

See Also

- [Viewing Congestion With the Design Vision Layout Window](#)
- The “Analyzing Congestion” topic in Design Vision Help

Reducing Congestion in the Floorplan

You can improve congestion in the floorplan by using Design Compiler Graphical floorplan exploration. Floorplan exploration allows you to perform floorplanning tasks, including floorplan analysis and modification from within the synthesis environment. For example, if your macro placement appears to be causing congested “hot spots,” you can move the macro locations to reduce congestion. For more information about using floorplan exploration to view and modify floorplans, see [SolvNet article 030613, “Design Compiler Graphical Floorplan Exploration Application Note.”](#)

Controlling Placement Density

You can control how densely cells are packed during placement by using the following strategies:

- [Separating High-Pin-Density Cells](#)
- [Allowing Cell Clusters](#)
- [Improving Placement of Wide Cells](#)

Separating High-Pin-Density Cells

High-pin-density cells are hard to route when they are placed too close together. You can reduce congestion and improve routability for designs with high-pin-density cells by setting the following variables before running the `compile_ultra` command:

1. To direct the tool to spread out cells with a high pin density, set the `compile_high_pin_density_cell_spreading` variable to `true`:

```
dc_shell-topo> set_app_var \
    compile_high_pin_density_cell_spreading true
```

The variable is `false` by default.

2. To specify a pin-density threshold percentage, specify a value with the `compile_high_pin_density_cell_spreading_threshold` variable. The cells with a pin density that exceeds the threshold percentage are spread out during placement.

```
dc_shell-topo> set_app_var \
    compile_high_pin_density_cell_spreading_threshold 0.75
```

If you enable the `compile_high_pin_density_cell_spreading` variable but do not specify a pin-density threshold, the default threshold is 0.9 percent.

Allowing Cell Clusters

For placement that is not focused on removing congestion, you can enable a mode of coarse placement in which cells are not distributed evenly across the surface of the chip but are allowed to clump together. This avoids long route QoR issues for low-utilization designs.

To set the threshold of how tightly the cells are allowed to clump, use the `placer_max_cell_density_threshold` variable. Set the variable to a value that is above the overall utilization of the design but below 1.0. For example, if your overall utilization is 50 percent (0.5), a reasonable value for this variable is a value between 0.5 and 1.0. The higher the value, the more tightly the cells clump together.

For example, if the global utilization is 50 percent, choose a value between 0.5 and 1:

```
dc_shell-topo> set_app_var placer_max_cell_density_threshold 0.7
```

If you do not set this variable, and the tool detects a design with low utilization, the tool sets the `placer_max_cell_density_threshold` variable dynamically, based on the utilization at that stage:

- For utilization less than 40 percent, the tool sets the cell density threshold to 0.7 during compile and incremental compile
- For utilization between 40 percent to 60 percent, the tool sets the cell density threshold to 0.6 during compile and sets it to 0.7 during incremental compile
- For utilization between 60 percent to 65 percent, the tool sets the cell density threshold to -1 (the default) during compile and sets it to 0.7 during incremental compile
- For utilization more than 65 percent, the tool sets the cell density threshold to -1 (the default) during compile and incremental compile

The tool restores the variable back to its default (-1) when the command completes.

The `placer_max_cell_density_threshold` value is not transferred to the IC Compiler tool. You must set it to be consistent with Design Compiler Graphical.

As with Design Compiler Graphical, the IC Compiler user-specified `placer_max_cell_density_threshold` value takes precedence even if the design has low utilization and the user-specified value is set lower than 0.5; however, the tool issues a warning.

Improving Placement of Wide Cells

You can improve the placement of wide cells in a design. Wide cells are cells with a width that exceeds half the distance between adjacent M1 power and ground (PG) straps. When wide cells are placed too close to each other, cells are displaced during legalization, which might affect QoR and congestion.

The following variables improve the placement of wide cells:

- `placer_wide_cell_use_model`
Distributes wide cells evenly to minimize displacement of cells during legalization and reduce congestion. The default is `true`.
- `placer_wide_cell_pg_strap_distance`
Specifies the pitch of the PG straps in the design. The default is 0.

To improve the placement of wide cells in a region, set these variables together as shown in the following example:

```
dc_shell-topo> set_app_var placer_wide_cell_use_model true  
dc_shell-topo> set_app_var placer_wide_cell_pg_strap_distance 0.3
```

Clustering Logic Modules to Minimize QoR Variations

You can perform initial placement during compile in a mode that minimizes QoR variations for subsequent compile runs that are based on small design changes by using the `spg_place_enable_precluster` variable.

By default (`true`), the placer is more aware of logic modules and clusters groups of logic in the netlist, as follows:

- The relative placement of modules and logic clusters are more predictable from run to run. Small changes in your netlist or constraints do not determine where modules are placed.
- Logic modules are more likely to be placed near fixed connections, such as ports and macros.
- Logic modules and logic clusters are more likely to be placed in one physical area instead numerous areas. However, some logic modules might be composed of several loosely connected clusters. These loosely connected modules are divided if it helps quality of results.

You can disable this behavior by setting the `spg_place_enable_precluster` variable to `false`.

Specifying Design Constraints and Power Settings

Before performing design implementation, you must first specify the design QoR objectives for synthesis. You need to apply the design constraints and power settings that will be used to perform mapping and placement-driven optimization in Design Compiler topographical mode.

In the physical guidance synthesis flow, it is especially important that you fully specify your design goals before you run the first `compile_ultra` command step in order to achieve good QoR results that correlate to IC Compiler.

Keeping in mind that Design Compiler physical guidance information is used as seed placement for the `place_opt` command in IC Compiler, the following design objectives should be consistent so the placement is aligned and the timing context is the same between both tools:

- [Design-Specific Settings](#)
- [Physical Constraints](#)
- [Power Optimization Settings](#)

Design-Specific Settings

Align your design settings in Design Compiler with the settings used in IC Compiler. For example, if your back-end flow disables the use of certain metal layers for routing or uses any commands that affect net parasitic (resistance and capacitance) estimation, you need to specify the same design settings in Design Compiler.

The following list shows the type of design settings that should be consistent between Design Compiler and IC Compiler:

- Use the `set_ignored_layers` command to disable the routing resources on certain metal layers. Ignoring metal layers affects Design Compiler placement-driven net capacitance and resistance computation. The ignored metal layer is not considered part of the routing resource; its effect on the resistance and capacitance is not considered during optimization and congestion estimation in topographical mode.
- Use the `set_delay_estimation_options` command to specify scaling factors for derived resistance (R) and capacitance (C) from the library or override the per-unit R and C values of any metal layers specified in the physical library. A scaling factor within 5% to 10% is usually sufficient.
- Use the `set_ahfs_options` command to fine-tune automatic high-fanout synthesis options. By default, Design Compiler performs placement-based automatic high-fanout synthesis for high-fanout nets (with fanout above 100) such as reset and scan-enable, with the exception of `dont_touch` nets, ideal nets, and DRC-disabled clock and constant nets. Placement of fanout influences buffering. As a result, you need to allow Design Compiler to perform the default automatic high-fanout synthesis and only configure automatic high-fanout synthesis options to the same options as IC Compiler for consistent buffering behavior. IC Compiler automatically performs incremental automatic high-fanout synthesis as needed when you run the `place_opt` command.
- Use the `dont_touch` and `size_only` attributes to control cell mapping and optimization. The usage of these attributes limits optimization flexibility and affects QoR and

correlation. Thus, you should use the same sets of `dont_touch` and `size_only` attributes in both tools.

- Use the same set of `set_ideal_network` constraints that are defined during `place_opt` in IC Compiler to ensure correlation.

You should only use these commands when they are being used in your back-end flows because they affect placement, parasitic estimation, mapping, and placement-based optimization in topographical mode the same way as in IC Compiler. Using consistent commands in both tools results in improved correlation for physical implementation.

Physical Constraints

In the physical guidance flow, you should drive placement-based optimization in synthesis with the same set of physical constraints used in IC Compiler to preserve placement consistency and improve correlation with IC Compiler.

Design Compiler supports a number of physical constraints. The physical constraints are extracted from your design floorplan DEF file with the `extract_physical_constraints` command, or they are written out from the Tcl output when you use the `write_floorplan` command in IC Compiler, and they are read into Design Compiler topographical mode when you use the `read_floorplan` command. For more information about the physical constraints supported in Design Compiler topographical mode, see [Using Floorplan Physical Constraints](#).

In addition to supporting physical constraints, Design Compiler also supports constraints for specifying the following:

- Net shapes or preroutes
- Relative placement
- Voltage area
- Tracks

You should consider the following physical constraint specification and usability guidelines for physical guidance flows and comply with the minimum set of floorplan requirements before you run the `compile_ultra` command:

- Use the same floorplan and set of physical constraints in Design Compiler as in IC Compiler. It is recommended that you drive synthesis with the same floorplan used for physical implementation in IC Compiler. However, as a minimum, you must specify the die area before proceeding with mapping and placement-based optimization using the `compile_ultra -spg` command.

It is recommended that you specify the macro and block abstraction locations, the site arrays, and the port locations for all primary I/Os.

To ensure that you have provided the required floorplan information, explicit physical guidance usability checks are automatically performed at the onset of the `compile_ultra -spg` command run in topographical mode. You can also run the `compile_ultra -check_only -spg` command to get information about any missing physical constraints.

- Use the same floorplan information between the initial `compile_ultra` step and subsequent incremental compilation steps with the exception of creating new test ports and specifying locations for the new test ports. If you do not provide the floorplan information before the initial `compile_ultra` step, you might see warning messages.

If you need to modify the floorplan, you can use the Design Compiler Graphical floorplan exploration capability. For more information about using Design Compiler Graphical floorplan exploration, see [Floorplan Exploration Floorplanning With IC Compiler](#).

Design Compiler Graphical also allows you to make floorplan changes interactively on the command line between the initial `compile_ultra -spg` step and subsequent `-incremental -spg` compilation steps without updating the floorplan constraints with the `write_floorplan` command. In this case, you must use the supported floorplan modification commands described in [SolvNet article 1502496, “Making Floorplan Changes After Running the compile_ultra -spg Command.”](#) However, the tool issues a warning message, indicating that it detected floorplan changes.

Power Optimization Settings

Design Compiler Graphical supports power prediction and optimization in the physical guidance flow, which provides accurate power estimation and correlates post-synthesis power numbers with the numbers obtained after place and route. The power optimization and prediction settings are used during the initial or incremental physical guidance runs. For more information, see [Power Correlation](#).

Design Compiler Graphical automatically performs leakage power optimization, which reduces the leakage power of your design whenever possible without affecting the performance. To reduce the overall leakage power of the design, leakage power optimization is performed on paths that are not timing-critical. When the target libraries are characterized for leakage power and contain cells characterized for multiple threshold voltages, Design Compiler Graphical uses the library cells with appropriate threshold voltages to reduce the leakage power of the design.

Dynamic power optimization and low-power placement are also supported in Design Compiler Graphical when you use the following commands:

- `set_dynamic_optimization true`
- `power_low_power_placement true`

When you enable the low-power placement feature, the tool optimizes the dynamic power by placing nets with high switching activity close to each other. Because the dynamic power

saving is based on the switching activity of the nets, annotate the switching activity by using the `read_saif` command. Then, synthesize the design using the `compile_ultra -spg` command. [Example 20-2](#) shows how to enable and use the low-power placement feature.

Example 20-2 Enabling and Using Low-Power Placement

```
prompt> set_dynamic_optimization true
prompt> set_power_low_power_placement true
prompt> read_saif -input s1.saif -instance_name inst_1
prompt> compile_ultra -spg
prompt> report_power
```

It is recommended, but not required, to read in the RTL SAIF prior to optimization. If the RTL SAIF is not available, the tool will use the defaults, static probability of 0.5 and toggle rate of 0.1. If you want to annotate your own values, use the `set_switching_activity` command.

When you enable dynamic power optimization, you must use multiple threshold voltage libraries. For accurate results, set the dynamic power before running the `compile_ultra -spg` command. You can also optimize dynamic power incrementally to provide better QoR and reduce the runtime.

To enable clock-gating optimization, use the `-gate_clock` option with the `-spg` option of the `compile_ultra` command. The tool automatically inserts, modifies, or deletes a clock-gating cell unless you have marked the cell or its parent hierarchical cell with the `dont_touch` attribute.

If you enable power correlation by using the `set_power_prediction` command, the tool performs clock tree estimation during the last phase of the `compile_ultra` or `compile_ultra -incremental` command run. Any subsequent incremental compilations cause clock tree estimation to run again.

To provide a list of library cells to be used for clock tree estimation, use the `-ct_references` option with the `set_power_prediction` command. Using the same clock buffers that are used in IC Compiler clock tree synthesis helps ensure that the predicted power is further correlated with IC Compiler.

If you enable power prediction, the `report_power` command reports the correlated power if the design is mapped to technology-specific cells. If power prediction is disabled, the `report_power` command reports only the total power, static power, and dynamic power used by the design without accounting for the estimated clock-tree power.

See Also

- [Power Correlation](#)
Provides information about configuring power optimization and power prediction
- [Leakage Power and Dynamic Power Optimization](#)

Provides information about leakage power optimization using multithreshold voltage libraries

- The *Power Compiler User Guide*

Provides information about dynamic power optimization and low-power placement

Using Layer Optimization to Increase the Accuracy of Net Delay Estimation

Design Compiler Graphical layer-aware optimization models parasitic variation across metal layers in a way that benefits optimization. This on-by-default optimization helps remove excess pessimism, leading to better area and power.

Soft constraints are passed from the Design Compiler Graphical tool to the IC Compiler II tool as needed to maintain good correlation. To read the soft constraints into the IC Compiler II tool, run the `write_icc2_files` command to write out the constraints, and then source the top-level script generated by the `write_icc2_files` command in the IC Compiler II tool.

You can also control layer optimization by using a net-pattern approach or by setting specific constraints. This section provides an overview. For more information about layer-optimization commands, related commands, and the layer optimization flow, see [SolvNet article 037946, “How to Implement Net Layer Optimization With Design Compiler Graphical.”](#)

Layer optimization is supported only in the Design Compiler Graphical tool, and it is performed automatically when you use the `-spg` option with the `compile_ultra` command.

In addition to the default layer-aware optimization, you can also specify net constraints for layer optimization using the following methods:

- A net pattern

Use the `create_net_search_pattern` command to define a pattern to identify nets, and then use the `set_net_search_pattern_delay_estimation_options` command to define which layers to use for a specific pattern.

- User constraints

Use the `set_net_routing_layer_constraints` command to specify which nets will have specific layer assignment constraints applied to them.

The net-pattern and user-constraints layer optimization methods might affect runtime.

Note:

If layer optimization and nondefault routing rules both apply to a net, the optimization cost factor considers them both.

Enabling Net Promotion to Multiple Layers

You can enable the promotion of nets to multiple layers (more than the top two layers) by setting the `spg_enable_zroute_layer_promotion` variable to `true` in the Design Compiler Graphical tool. When you enable the variable, the tool

- Chooses a range of layers for promotion.
- Chooses candidate nets based on length and timing.
- Uses global routing to determine the best set of nets to promote and the exact range of layers for each net.
- Optimizes buffering for the nets.

You can also use pattern-based matching to control the candidate nets. To use pattern-based layer promotion, perform the following steps before running the `compile_ultra` or `compile_ultra -incremental` command:

1. Set the `spg_enable_zroute_layer_promotion` variable to `true`.
2. Create patterns to identify candidate nets for promotion by using the `create_net_search_pattern` command:

```
set pat1 [create_net_search_pattern -setup_slack_upper_limit "-1.0" \
    -connect_to_port]
set pat2 [create_net_search_pattern -setup_slack_upper_limit "-2.0" \
    -net_length_lower_limit 50 ]
```
3. Apply the patterns using the `set_net_search_pattern_delay_estimation_options` command:

```
set_net_search_pattern_delay_estimation_options -pattern $pat1 \
    -min_layer_name M7 -max_layer_name M10
set_net_search_pattern_delay_estimation_options -pattern $pat2 \
    -min_layer_name M9 -max_layer_name M10
```

This determines the net candidates. Global routing determines the exact promotions.

Enabling the `spg_enable_zroute_layer_promotion` variable uses Zroute for global routing; therefore, it might have a significant impact on runtime. By default, the variable is `false`.

Working With Nondefault Routing Rules

You can define nondefault routing rules to define stricter (wider) wire width and spacing requirements for certain nets, such as long nets and timing-critical signal nets. Using wider metal and increased spacing can improve timing and reduce crosstalk for these nets.

For information about working with nondefault routing rules, see the following topics:

- [Defining Nondefault Routing Rules](#)
- [Applying Nondefault Routing Rules](#)
- [Reporting Nondefault Routing Rules](#)
- [Removing Nondefault Routing Rules](#)
- [Managing Nondefault Routing Rules in the Design Flow](#)

Defining Nondefault Routing Rules

You define nondefault routing rules by using the `define_routing_rule` command. When you define a nondefault routing rule, you must specify a name for the nondefault routing rule. If you attempt to redefine a nondefault routing rule that already exists, the tool issues an error. To change an existing rule, use the `remove_routing_rules` command to remove the rule and then use the `define_routing_rule` command to redefine the rule.

The `define_routing_rule` command defines a named rule but does not apply it to the design; rules are applied to the design using separate commands.

Design Compiler Graphical supports a subset of the nondefault routing rules (width and spacing) used in the IC Compiler tool. To see if a particular option is supported, see the option description in the `define_routing_rule` man page.

A single routing rule definition can contain multiple requirements, such as width and spacing requirements together.

The following topics describe the tasks associated with defining nondefault routing rules.

Defining Minimum Wire Width Rules

You can use a nondefault routing rule to define minimum wire width rules that are stricter (wider) than the minimum width rules defined in the Milkyway technology file.

To define a minimum wire width rule, use the `define_routing_rule` command. You can specify the minimum width by specifying a multiplier that is applied to the default width for each layer, or by specifying the minimum width in microns for each layer, or both.

- To use a multiplier to specify the minimum width, use the following syntax:

```
define_routing_rule rule_name
    [-default_reference_rule | -reference_rule_name ref_rule]
    [-multiplier_width layer_width]
```

The default width for each layer is determined from the reference rule, which is either the default routing rule or the reference rule specified in the `-reference_rule_name` option.

- To specify the minimum width values for each layer, use the following syntax:

```
define_routing_rule rule_name
[-default_reference_rule | -reference_rule_name ref_rule]
[-widths {layer1 width1 layer2 width2 ... layern widthn}]
```

Specify the routing layers by using the layer names from the technology file. You can define a single width value per layer. By default, the tool uses the wire width from the reference rule, which is either the default routing rule or the reference rule specified in the `-reference_rule_name` option, for any layers not specified in the `-widths` option.

- If you specify both the `-multiplier_width` option and the `-widths` option, the tool uses the `-widths` option to determine the base width, and then applies the multiplier to that value to determine the minimum width requirement.

For example, to define a nondefault routing rule named `new_width_rule` that uses the default routing rule as the reference rule and defines nondefault width rules for the METAL1 and METAL4 layers, use the following command:

```
dc_shell-topo> define_routing_rule new_width_rule \
    -widths {METAL1 0.8 METAL4 0.9}
```

To define a nondefault routing rule named `relative_width_rule` that defines nondefault widths of two times the width defined in the default routing rule for all layers, use the following command:

```
dc_shell-topo> define_routing_rule relative_width_rule \
    -multiplier_width 2.0
```

Defining Wire Spacing Rules

You can use a nondefault routing rule to define minimum wire spacing rules that are stricter (more widely spaced) than the minimum spacing rules defined in the Milkyway technology file.

To define a minimum wire spacing rule, use the `define_routing_rule` command. You can specify the minimum spacing by specifying a multiplier that is applied to the default spacing for each layer, by specifying the minimum spacing in microns for each layer, or both.

- To use a multiplier to specify the minimum spacing, use the following syntax:

```
define_routing_rule rule_name
[-default_reference_rule | -reference_rule_name ref_rule]
[-multiplier_spacing layer_spacing]
```

The default spacing for each layer is determined from the reference rule, which is either the default routing rule or the reference rule specified in the `-reference_rule_name` option.

- To specify the minimum spacing values for each layer, use the following syntax:

```
define_routing_rule rule_name
  [-default_reference_rule | -reference_rule_name ref_rule]
  [-spacings {layer1 spacing1 layer2 spacing2 ... layern spacingn}]
```

Specify the routing layers by using the layer names from the technology file. You can define a single spacing value per layer. By default, the tool uses the wire spacing from the reference rule, which is either the default routing rule or the reference rule specified in the `-reference_rule_name` option, for any layers not specified in the `-spacings` option.

- If you specify both the `-multiplier_spacing` option and the `-spacing` option, the tool uses the `-spacings` option to determine the base spacing, and then applies the multiplier to that value to determine the minimum spacing requirement.

For example, to define a nondefault routing rule named `new_spacing_rule` that uses the default routing rule as the reference rule and defines nondefault spacing rules for the METAL1 and METAL4 layers, use the following command:

```
dc_shell-topo> define_routing_rule new_spacing_rule \
  -spacings {METAL1 0.8 METAL4 0.9}
```

To define a nondefault routing rule named `relative_spacing_rule` that defines nondefault spacings of two times the spacing defined in the default routing rule for all layers, use the following command:

```
dc_shell-topo> define_routing_rule relative_spacing_rule \
  -multiplier_spacing 2.0
```

To define a nondefault routing rule with single-side spacing, which limits spacing to one side of the net, use the `-single_side_spacing` option. For example, the following command defines single-side spacing rules on metal layers METAL2, METAL3, and METAL4:

```
dc_shell-topo> define_routing_rule NDR_1 -spacings { METAL2 1.3 \
  METAL3 1.4 METAL4 1.5 } -single_side_spacing
```

Applying Nondefault Routing Rules

To apply a nondefault routing rule to one or more nets, use the `set_net_routing_rule` command. It is recommended that you set nondefault routing rule constraints after compiling your design, before running an incremental compile. Otherwise, the logic associated with the nets might get optimized away during compile, and, as a result, the nets would be optimized away and the nondefault routing rules would be lost.

To assign a nondefault routing rule called `WideMetal` to nets named `DATA*`, use the following command:

```
dc_shell-topo> set_net_routing_rule -rule WideMetal [get_nets DATA*]
```

To apply a nondefault routing rule to nets that meet a pattern previously defined with the `create_net_search_pattern` command, use the following command:

```
dc_shell-topo> set_net_search_pattern_delay_estimation_options \
    -pattern $pattern_id -rule WideMetal
```

This method is particularly useful in a synthesis flow where exact net names are not yet available during initial synthesis or net names change during optimization.

Zroute does not consider nondefault routing rules applied with this method; it only considers rules applied by net name.

Note:

If layer optimization and nondefault routing rules are both applied to a net, the optimization cost factor considers them both.

Reporting Nondefault Routing Rules

To report the nondefault routing rules for specific nets, use the `report_net_routing_rules` command.

```
dc_shell-topo> report_net_routing_rules [get_nets *]
```

To output a Tcl script that contains the `define_routing_rule` commands used to define the nondefault routing rules for the specified nets, use the `-output` option when you run the `report_net_routing_rules` command.

To report the design-specific nondefault routing rules defined by the `define_routing_rule` command, use the `report_routing_rules` command. By default, this command reports all the nondefault routing rules for the current design. To limit the report to a specific nondefault routing rule, specify the rule name as an argument to the command.

```
dc_shell-topo> report_routing_rules rule_name
```

To output a Tcl script that contains the `define_routing_rule` commands used to define the specified nondefault routing rule or all nondefault routing rules for the design if you do not specify a routing rule, use the `-output` option when you run the `report_routing_rules` command.

Removing Nondefault Routing Rules

To remove the nondefault routing rules for specific nets, use the `set_net_routing_rule` command to reset the routing rule for the nets to the default routing rule.

```
dc_shell-topo> set_net_routing_rule -rule default [get_nets my_ndr_nets]
```

Managing Nondefault Routing Rules in the Design Flow

Nondefault routing rules are saved when the design is written to a binary .ddc file. They are restored only when the .ddc file is read back into a Design Compiler Graphical session. They are not restored when the .ddc file is read back into the IC Compiler tool. They are not restored when reading an ASCII netlist back into either tool.

To ensure correlation between the tools, respecify the nondefault routing rules in the IC Compiler tool.

You can use the `report_routing_rules -output` command to generate a Tcl script that can be read into the Design Compiler Graphical or IC Compiler tool.

Automatic Assignment of Nondefault Routing Rules on Nets

Instead of manually applying nondefault routing rules to define stricter wire width and spacing requirements for certain nets, you can allow the tool to choose candidate nets based on slack and length and use timing-driven optimization to finalize nondefault routing rule net promotion.

You can enable the automatic assignment of nondefault routing rules on nets in the Design Compiler Graphical tool as follows:

1. Define the nondefault routing rules by using the `define_routing_rule` command.
2. Set the `optimize_ndr_user_rule_names` variable to a prioritized list of nondefault routing rules that you want to use for automatic net assignment, for example:

```
dc_shell-topo> define_routing_rule "ndr_1w2s" -multiplier_spacing 2
dc_shell-topo> define_routing_rule "ndr_2w3s" -multiplier_spacing 3 \
    -multiplier_width 2
dc_shell-topo> set optimize_ndr_user_rule_names { ndr_1w2s ndr_2w3s }
dc_shell-topo> compile_ultra -spg
```

You can also use the following variables to further control the automatic assignment of nondefault routing rules on nets:

- To set a limit on the maximum number of nets to be assigned nondefault routing rules during optimization, use the `optimize_ndr_max_nets` variable.
For example, if you set the variable to 500, the tool automatically generates nondefault routing rules for up to 500 nets in the design. The default is 2000.

```
dc_shell-topo> set_app_var optimize_ndr_max_nets 500
```

- To set a minimum length threshold on the nets considered for nondefault routing rules during optimization, use the `spg_auto_ndr_net_length_threshold` variable.

The value you specify is multiplied by the minimum height of the unit tile. The default is 100; therefore, by default the minimum length of the nets considered for nondefault routing rule assignment is 100 times the minimum height of the unit tile.

- To choose nets within a specific slack range for nondefault routing rules during optimization, use the `optimize_ndr_critical_range` variable.

The nets that fall within the following range are considered for nondefault routing rules:

```
[WNS, WNS*(1-optimize_ndr_critical_range)]
```

By default, the `optimize_ndr_critical_range` variable is 0.008. Therefore, if the worst negative slack (WNS) is -1000, the range is [-1000, -992] where $-992 = -1000 * (1-0.008)$. If the delay value falls into this range, it is considered critical.

The tool chooses the candidate nets when you run the `compile_ultra -spg` or `optimize_netlist -area` command.

To write out the routing rules and nondefault routing rule specifications for use in the IC Compiler II tool, run the `write_icc2_files` command. The Design Compiler Graphical tool creates the following files in the `<design_name>.floorplan` directory:

- `routing_rule.tcl`—Includes the routing rules
- `routing_rule.net.tcl`—Includes the nondefault routing rule specifications

The `-rule_is_user` option setting (`true` or `false`) indicates whether the nondefault routing rule is user-specified or tool-generated.

The IC Compiler II tool treats user-specified constraints as hard constraints and tool-generated constraints as soft constraints.

Note:

Some `define_routing_rule` command options are not supported. See the man page for details.

Enabling the Physical Guidance Flow

To enable physical guidance in Design Compiler, use the `compile_ultra` command with the `-spg` option. You must specify the die area at a minimum to ensure floorplan consistency between Design Compiler and IC Compiler. It is recommended that you also specify the macro and block abstraction locations, the site arrays, and the port locations for all primary I/Os.

Design Compiler performs explicit physical guidance usability checks the first time you run the `compile_ultra -spg` command in topographical mode to ensure that you have provided the required floorplan information, and error messages and warning messages are issued for any missing floorplan information. For example, Design Compiler issues a warning message if a macro location is missing during a physical guidance flow; however, the `compile_ultra -spg` command assigns the macro location automatically.

If a port, macro, or block abstraction location is missing, you can proceed with synthesis. However, you must provide the location information in IC Compiler. Otherwise, correlation could be affected. Alternatively, you can use Design Compiler Graphical floorplan exploration to place the ports, macros, or block abstractions. After you specify the physical constraints, update the floorplan in floorplan exploration. The `compile_ultra -spg` command can move ports incrementally to enhance timing if the port locations have not been defined.

The IC Compiler `restore_spg_placement` command restores the port locations created by the Design Compiler `compile_ultra -spg` command for ports that were not constrained by the `create_terminal` command or defined in the DEF file. The `restore_spg_placement`

command also restores the standard cell placement created by the `compile_ultra -spg` command. For more information, see the IC Compiler documentation.

In the physical guidance flow, the tool uses both RTL congestion optimization and congestion-aware cell placement to consider cell density and provide a more accurate net model for optimization in topographical mode. This helps to ensure that timing is consistent between the endpoint in Design Compiler and the startpoint in IC Compiler. In addition, placement and placement-based optimization in physical guidance enhances the delay optimization effort in Design Compiler so it is consistent with IC Compiler behavior.

At the end of the `compile_ultra -spg` command run, Design Compiler creates signatures of the physical constraints that were used. The signatures are saved in binary format, in either the .ddc file or the Milkyway CEL, to be later matched in IC Compiler against the physical constraints used for the `place_opt -spg` command run. This allows you to ensure that the physical constraints used in Design Compiler match the physical constraints used in IC Compiler.

At this point in the flow, the mapped top-level design is marked with a read-only `dct_spg_flow_done` attribute to signify that the design was implemented with physical guidance. You can query the attribute to help determine if your design .ddc file or Milkyway CEL was generated using physical guidance.

The derived placement for standard cells and the design floorplan information used to drive synthesis are retained on the in-memory implemented design. You can save the physical guidance information in a .ddc file by using the `write_file` command or to a Milkyway CEL view by using the `write_milkyway` command. In the ASCII flow, you can save the physical guidance information in a .def file by using floorplan exploration.

Enabling the Physical Guidance Incremental Flow

Design Compiler Graphical also provides an incremental physical guidance flow. The information in the full `compile_ultra -spg` command section, [Enabling the Physical Guidance Flow](#), also applies to the `compile_ultra -incremental -spg` command. In addition, the following applies to the incremental flow:

- When you run the `compile_ultra -incremental -spg` command on an input .ddc file, Design Compiler Graphical performs an explicit check for the presence of the read-only `dct_spg_flow_done` attribute to ensure that the design was initially implemented in physical guidance mode.

The tool issues an error message when you run subsequent `compile_ultra -incremental -spg` commands on a .ddc file that was not generated using physical guidance.

- When Design Compiler Graphical confirms that the .ddc file that is being read was generated using physical guidance, any subsequent placement-based optimization using the `compile_ultra -incremental` command uses physical guidance regardless of whether you include the `-spg` option. That is, the `compile_ultra -incremental` and `compile_ultra -incremental -spg` commands have the same effect. At this point, the tool issues a warning message informing you that you are in physical guidance mode and that the design was compiled with physical guidance optimizations.
- You can set the `spg_congestion_placement_in_incremental_compile` variable to `true` to enable congestion-driven placement during incremental compile to improve congestion while preserving quality of results. This variable also improves congestion correlation results, comparing post-incremental compile in Design Compiler and post-placement optimization in IC Compiler. You do not need to enable Zroute to use the variable. If Zroute is enabled, it runs congestion-driven placement in incremental compile by default.
- When you run the `compile_ultra -incremental -spg -scan` command, Design Compiler Graphical reorders and repartitions the scan elements during incremental compile to further reduce congestion. To disable reordering and repartitioning, set the `test_enable_scan_reordering_in_compile_incremental` variable to `false`.

Scan chain reordering and repartitioning are enabled in binary and ASCII flows. For more information about performing scan chain reordering and repartitioning using an ASCII flow, see [Incremental ASCII Flow With a Third-Party DFT Flow Example](#).

The physical guidance flow accepts any of the input source formats that are supported by Design Compiler. The following describes the expected behavior for the `compile_ultra -incremental -spg` command on various input source types:

- .ddc file generated with the `-spg` option

The `compile_ultra -incremental -spg` command proceeds with placement-based optimization in physical guidance mode.

- .ddc file generated without the `-spg` option

The `compile_ultra -incremental -spg` command issues a DCT-054 error message. You must first use `compile_ultra -spg` to map the design.

- ASCII mapped netlist

If you provide standard cell placement information by using the `-standard_cell spg` option with the `extract_physical_constraints` command, the `compile_ultra -incremental -spg` command proceeds with placement-based optimization in physical guidance mode. The tool does not support a netlist that contains both mapped and unmapped cells in this flow.

If you provide standard cell placement information by using the `-standard_cell topo` option with the `extract_physical_constraints` command, the `compile_ultra`

`-incremental -spg` command issues a DCT-054 error message because the design was not compiled with Design Compiler topographical physical guidance optimizations.

If you do not provide standard cell placement information, the `compile_ultra -incremental -spg` command runs limited mapping optimizations and then performs placement-based optimization in physical guidance mode.

If you use other mapping commands, such as `optimize_registers` or netlist editing commands that perform various degrees of design modification, it is recommended that you run an explicit incremental compilation flow afterward with the `-spg` option. However, it is not required. The `place_opt -spg` command in IC Compiler can handle cells that do not have physical guidance defined.

It is recommended, but not required, that you run an incremental compilation flow after running the following commands:

- `optimize_registers`
- `insert_dft`
- `insert_buffer`
- `remove_buffer`
- `balance_buffer`
- `change_link`

The `compile_ultra -incremental -spg` command can move ports incrementally to enhance timing if the port locations have not been defined.

Using Physical Guidance in IC Compiler

To enable physical guidance in IC Compiler, use the `-spg` option with the `place_opt` command. When you run the `place_opt -spg` command, the stored physical guidance information from Design Compiler is used as the initial seed placement.

For more information about using physical guidance in IC Compiler, see [SolvNet article 031198, “Design Compiler and IC Compiler Physical Guidance Technology Application Note.”](#)

Using the Design Compiler Graphical and IC Compiler Hierarchical Flow

You can use physical guidance with a hierarchical flow in Design Compiler Graphical and IC Compiler. If the hierarchical blocks in Design Compiler correspond to physical blocks, use IC Compiler block abstractions to model the physical blocks at the top level in both tools. This ensures optimal correlation and alignment between the tools.

If IC Compiler hierarchical models are not available, Design Compiler physical blocks (full netlist .ddc files or Design Compiler block abstractions) can be used at the top level in Design Compiler Graphical. However, IC Compiler does not support Design Compiler physical blocks, so when you run the top-level design in IC Compiler, you must generate IC Compiler block abstractions.

The design that is written out from Design Compiler Graphical after completing top-level synthesis will contain physical guidance data only for the top-level logic. Physical guidance information will not be included for the physical blocks. Each physical block should be passed to IC Compiler separately to complete the physical guidance flow for each block.

If the hierarchical blocks in Design Compiler do not correspond to physical blocks, do not use block abstractions for those blocks when performing top-level synthesis. You should read in the block-level .ddc file at the top level without using the `set_physical_hierarchy` command on those blocks. The logic in blocks that are not physical blocks will be placed during top-level synthesis and will be included in the physical guidance information at the top level.

In the hierarchical Design Compiler Graphical and IC Compiler flow using physical guidance, specify the `compile_ultra -spg` command in Design Compiler Graphical, and specify the `place_opt -spg` command in IC Compiler. For more information about running a hierarchical flow, see [Performing a Bottom-up Hierarchical Compile](#).

Improved Timing Correlation Between Design Compiler Graphical and IC Compiler II

To improve timing correlation between the Design Compiler Graphical and IC Compiler II tools, use the `set_compile_spg_mode ICC2` command. When you specify the `ICC2` value with the `set_compile_spg_mode` command, the Design Compiler Graphical settings are aligned to the timer settings in the IC Compiler II tool. For example, the `ICC2` mode

- Removes long net pessimism
- Enables placement pre-clustering
- Enables recovery removal checks in the Design Compiler Graphical tool
- Enables layer-aware optimization

The command settings take effect when you run the `compile_ultra`, `compile_ultra-incremental`, or `optimize_netlist -area` command.

When you run the `set_compile_spg_mode` command,

- The tool reports a table with the updated tool settings.

You can also get a report of the updated settings by using the `report_compile_spg_mode` command.

- The tool settings that you have already set are preserved; they are not overwritten.
- Be sure to measure the results after running the `place_opt` command.

The Design Compiler Graphical tool stores these enhanced physical guidance settings in the `.ddc` file. Use the `write_icc2_files` command to write out the settings, along with other user settings that are supported in the IC Compiler II tool.

To read the settings into the IC Compiler II tool, source the top-level script generated by the `write_icc2_files` command in the IC Compiler II tool. You must enable physical guidance to use this improved timing correlation flow.

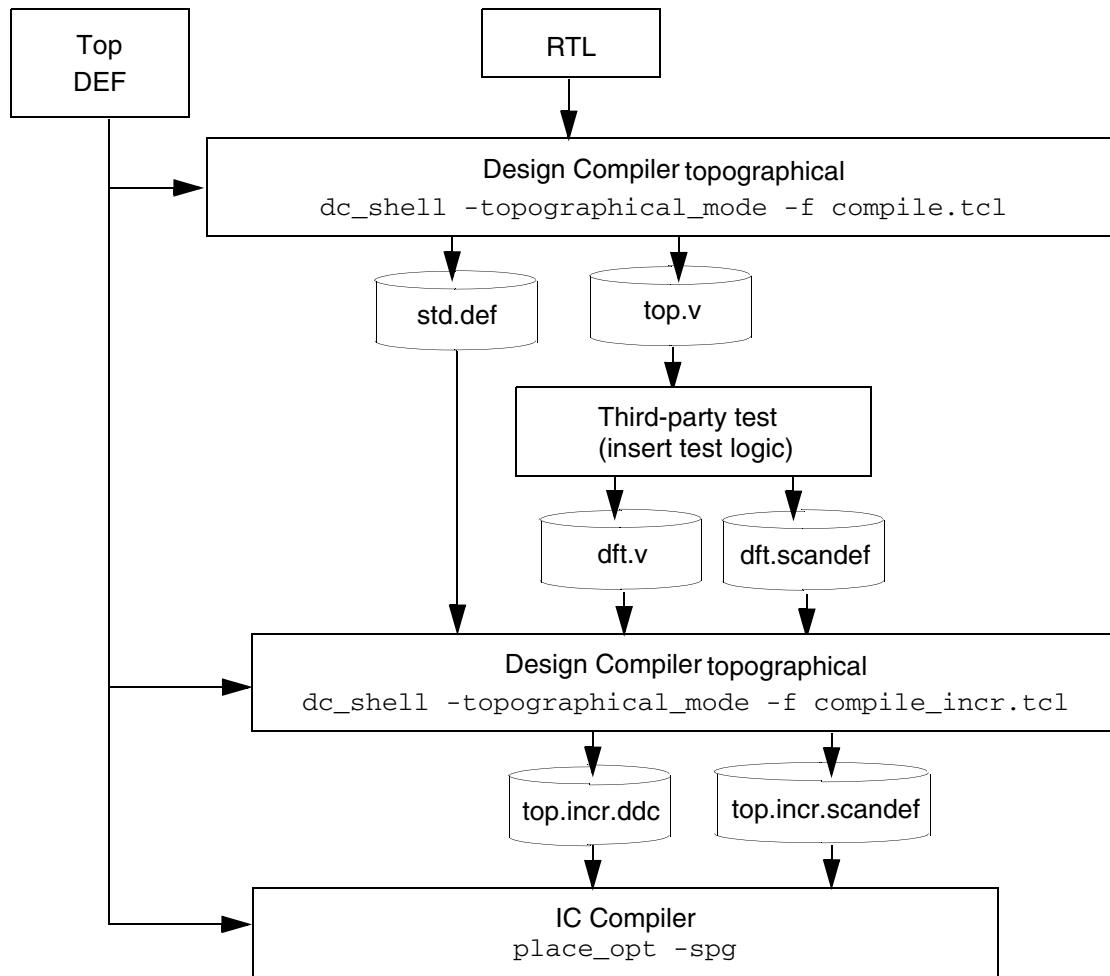
Incremental ASCII Flow With a Third-Party DFT Flow Example

Physical guidance information is automatically saved with your design when you use the `.ddc` or Milkyway file format. However, some flows require you to use a Verilog or VHDL (ASCII) netlist output format. Verilog and VHDL formats are not suitable for physical guidance storage. Therefore, you need to save the physical guidance information in a separate DEF file along with the gate-level Verilog or VHDL netlist. This section provides an overview of the incremental ASCII flow using a third-party DFT flow as an example.

Saving the physical guidance information in DEF format allows you to retain the placement of the standard cells after the initial `compile_ultra -spg` command run. The standard cell placement is restored for the netlist coming back from test insertion and is used in the incremental compilation when you run the `compile_ultra-incremental -spg` command. During the incremental step, Design Compiler uses the placement information that was saved in the DEF file as the seed placement, allowing better correlation between the `compile_ultra -spg` and `compile_ultra-incremental -spg` steps. You must use a fully mapped netlist in this flow.

[Figure 20-5](#) shows the physical guidance flow for third-party DFT flows.

Figure 20-5 Physical Guidance Third-Party Test Flow



For more information about using the incremental ASCII flow using a third-party flow as an example, see [Incremental ASCII Flow With a Third-Party DFT Flow Example](#).

Exporting the Design

You can save standard cell placement in Design Compiler in binary or ASCII format, as described:

- [Saving in Binary Format](#)
- [Saving in ASCII Format](#)
- [Saving in ASCII Format for IC Compiler II](#)

Saving in Binary Format

You can save standard cell placement in .ddc format by using the `write_file -format ddc` command, or save it in Milkyway CEL format by using the `write_milkyway` command.

In regular topographical mode, the standard cell placement can only be used by Design Compiler for further optimization. IC Compiler does not use it. However, in the physical guidance flow, the standard cell placement information is propagated to IC Compiler through the .ddc file or the Milkyway CEL.

The physical constraints and floorplan information used for synthesis are not passed from Design Compiler to IC Compiler. However, physical constraint signatures are created from the physical constraints at the end of the `compile_ultra -spg` and the `compile_ultra -incremental -spg` command steps in the physical guidance flow. These signatures are saved in the .ddc file or Milkyway CEL view. IC Compiler reads the signatures when the physical guidance information is restored and matches them to the physical guidance information in IC Compiler to ensure consistency between the floorplan used in synthesis and the floorplan used for physical implementation.

All binary netlists that are generated in the physical guidance synthesis flow, whether a .ddc file or a Milkyway CEL, also contain a read-only `dct_spg_flow_done` attribute to signify that the design was implemented with the physical guidance flow. You can query the attribute to determine if your design .ddc file or Milkyway CEL was generated using the physical guidance flow.

Saving in ASCII Format

When you save the design in ASCII format, either in a Verilog or a VHDL netlist, the file that is created does not contain standard cell placement information or physical constraint signatures.

You can save the standard cell placement information in a DEF file by using the `write_def` command in Design Compiler or in floorplan exploration. You can then read the DEF file, along with the ASCII netlist, back into Design Compiler to be optimized with the `compile_ultra -incremental -spg` command, or you can read it into IC Compiler before the `place_opt -spg` step.

Design Compiler Graphical requires additional licensing to write out a DEF file. If you do not meet these licensing requirements, the DEF file that is written out cannot be read by third-party tools. For details about licensing, contact your Synopsys representative.

Note:

The `write_def -scan` option is not supported in Design Compiler. Use the `write_scan_def` command to capture the scan chain information.

Saving in ASCII Format for IC Compiler II

You can generate all the files needed to load a design into IC Compiler II by using the `write_icc2_files` command.

The command writes the design files and Tcl scripts for the current design into the directory specified by the `-output` option. The files can include the following data:

- Netlist in Verilog format
- Propagated switching activity information (backward SAIF file)
- Tcl floorplan and DEF files
- Design's power intent as a UPF command script
- Scan chain information in SCANDEF format
- Constraints in SDC format
- Cell expansion data
- Timing contexts (scenarios)

These scenarios are written in a format that can be read into IC Compiler II to create the same set of scenarios as specified in Design Compiler.

To write these design data files, the `write_icc2_files` command uses the `write_file`, `write_saif`, `write_floorplan`, `save_upf`, and `write_timing_context` commands.

In addition, the `write_icc2_files` command writes the following scripts:

- The Design Compiler settings in IC Compiler II format
- The top-level script to load all data generated in IC Compiler II

The scripts generated by the `write_icc2_files` command do not include scripts for library creation in IC Compiler II. You need to create a library that is consistent with the library used in Design Compiler and then source the top-level script.

When used with the `set_host_options` command, the `write_icc2_files` command uses up to the user-specified number of CPU cores on the same computer for parallel execution. See the description of the `-max_cores` option in the `set_host_options` man page for more information.

The `write_icc2_files` command uses the `write_floorplan -format icc2` command to write out the physical constraints. For more information, see [Saving Physical Constraints in IC Compiler II Format](#).

Writing Standard Cell Placement Only

You can write out the standard cell placement information exclusively by using the `-golden_floorplan` option with the `write_icc2_files` command:

```
dc_shell-topo> write_icc2_files -output icc2_files \
    -golden_floorplan my_golden.floorplan.tcl
```

All other floorplan information is excluded from the output files. For example, routing optimization performed by the Design Compiler Graphical tool, including layer promotion and nondefault routing rule definitions are excluded. You must set these floorplan constraints in the golden floorplan file to include them.

When you source the top-level script generated by the `write_icc2_files` command in the IC Compiler II tool, as shown in the following `icc2_script.tcl` file, the tool reads the golden floorplan file and the DEF file with the Design Compiler Graphical standard cell placement information to the IC Compiler II floorplan file:

```
#####
# Read golden floorplan
#####
if {[file exists my_golden.floorplan.tcl]} {
    source -continue_on_error my_golden.floorplan.tcl
}
#####
# Read Design Compiler DEF (std cells)
#####
if {[file exists ${_DCG_ICC2_DIR_}/my_design.floorplan/floorplan.def]} {
    read_def ${_DCG_ICC2_DIR_}/my_design.floorplan/floorplan.def
}
```

Examples

The following examples show how to use the `write_icc2_files` command in Design Compiler Graphical to write the design files and Tcl scripts for the current design and then how to use the top-level script in IC Compiler II to load the design:

Example 20-3 Writing the Design Information From Design Compiler in IC Compiler II Format

```
# Perform synthesis and DFT insertion
dc_shell-topo> compile_ultra -scan -gate_clock -spg
dc_shell-topo> insert_dft
dc_shell-topo> compile_ultra -scan -spg -incremental
dc_shell-topo> optimize_netlist -area

# Change names if needed
dc_shell-topo> change_names -rules verilog -hierarchy

# Save the file for IC Compiler II using the write_icc2_files command
dc_shell-topo> write_icc2_files -output ./icc2_files
dc_shell-topo> quit
```

Example 20-4 Reading the Design into the IC Compiler II Tool

```
# Perform library setting in IC Compiler II  
icc2_shell> source -echo -verbose icc2_lib_setting.tcl  
  
# Load the design using script created by the write_icc2_files command  
icc2_shell> source -echo -verbose icc2_files/ORCA.icc2_script.tcl  
  
# Continue with the flow  
icc2_shell> commit_upf  
icc2_shell> place_opt
```

If you run the `write_icc2_files` command and warnings or errors occur, the tool issues an information message that points to the location of the log file that contains the warning or error message information:

```
dc_shell-topo> write_icc2_files -output ./icc2_files  
Information: During the execution of write_icc2_files error messages were  
issued, please check ./icc2_files/write_icc2_files.log files. (DCT-228)
```

Limitations

The `write_icc2_files` command currently has the following limitations:

- Hierarchical flows are not supported.
- Constraints set on library cells are not included except for the following:
 - SDC constraints: `set_timing_derate` and `set_driving_cell`
 - UPF constraints: `map_isolation_cell`, `map_retention_cell`, and `map_level_shifter_cell`
- The following commands are partially supported:
 - `set_congestion_options`
The `-max_util` option is included only for global settings, not for regional settings.
 - `set_multi_vth_constraint`
The setting is passed only for threshold-voltage groups defined in the library; the settings in user-defined groups are not included.
- The following variables are partially supported:
 - `power_default_toggle_rate_type`
The equivalent IC Compiler II variable only supports the `fastest_clock` value. The absolute setting is not passed.
 - `placer_channel_detect_mode`

The equivalent IC Compiler II variable only supports the `true` and `false` values. The `auto` value is not passed.

- `bus_naming_style`

The setting support is limited to delimiters defined in IC Compiler II: [], { }, (), and < >.

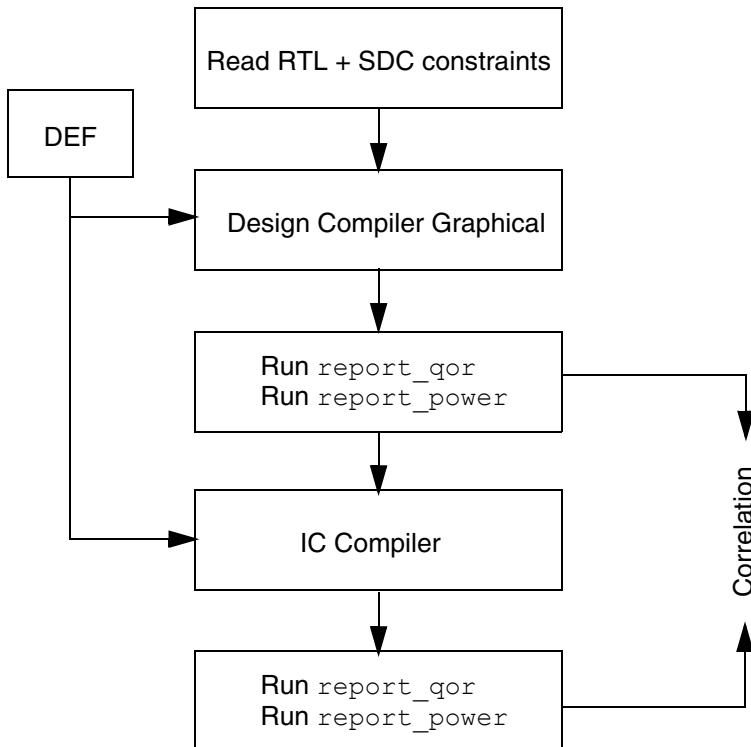
Reporting Physical Guidance Information

The physical guidance information is stored with the design in binary format. It is not available as a standalone file. To view the physical guidance information, you must run IC Compiler and execute the `place_opt -spg` command.

For correlation result comparison between non-physical guidance and physical guidance flows, you should compare post-synthesis results against respective post-placement results for each flow. However, you should measure and compare post-placement results instead of post-synthesis results when looking at overall QoR comparison between non-physical guidance and physical guidance flows.

[Figure 20-6](#) shows how correlation results are measured between Design Compiler and IC Compiler.

Figure 20-6 Correlation Measurement Between Design Compiler and IC Compiler



Physical Guidance Limitations

Physical guidance has the following limitations:

- IC Compiler does not support block abstractions created by Design Compiler. Therefore, in the hierarchical flow between Design Compiler and IC Compiler, you must use block abstractions created by IC Compiler.
- Design Compiler does not support the `-top` option with the `compile_ultra` command in the physical guidance flow.

Floorplan Exploration Floorplanning With IC Compiler

Design Compiler Graphical floorplan exploration allows you to use the IC Compiler floorplanning tools in the IC Compiler layout window. The following topics describe how to create, modify, and analyze floorplans in IC Compiler using Design Compiler Graphical.

- [Floorplan Exploration Overview](#)
- [Enabling Floorplan Exploration](#)
- [Running Floorplan Exploration](#)
- [Using the Floorplan Exploration GUI](#)
- [Creating and Editing Floorplans](#)
- [Analyzing the Data Flow for Macro Placement](#)
- [Saving the Floorplan or Discarding Updates](#)
- [Saving the Floorplan into a Tcl Script File or DEF File](#)
- [Exiting the Session](#)
- [Incremental or Full Synthesis After Floorplan Changes](#)
- [Using Floorplan Exploration With a dc_shell Script](#)
- [Black Box Support](#)
- [Handling Physical Hierarchies and Block Abstractions](#)

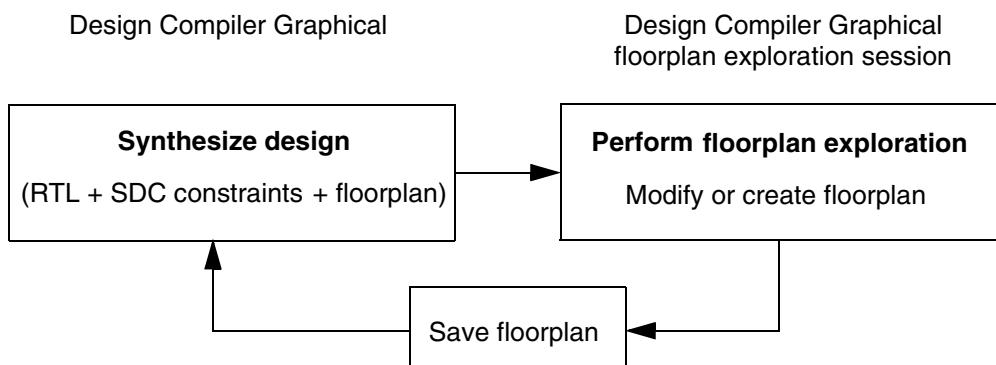
Floorplan Exploration Overview

Floorplan exploration allows you to perform floorplanning tasks, including floorplan analysis, floorplan creation, and floorplan modification from within the synthesis environment. Design Compiler Graphical floorplan exploration uses the IC Compiler floorplanning tools in the

IC Compiler layout window. Although you use the IC Compiler layout window, the interface between floorplan exploration in Design Compiler Graphical and the IC Compiler layout window is transparent, allowing you to move seamlessly between the Design Vision and IC Compiler layout windows.

[Figure 20-7](#) shows a typical floorplan exploration flow. You read an RTL file, specify the physical and logic libraries, define the Synopsys design constraints, specify a floorplan (if you have one), and synthesize the design in Design Compiler Graphical. After synthesis, you evaluate the QoR results and use floorplan exploration to create a new floorplan or improve an existing floorplan, as needed.

Figure 20-7 Floorplan Exploration Flow



Enabling Floorplan Exploration

Before you begin floorplan exploration, you must have the following:

- A synthesized design

Design Compiler Graphical issues an error message if it finds any unmapped logic.
- Both the physical and logic libraries

All cells in the design must link to the physical and logic libraries.

After you have a synthesized design that links to the physical and logic libraries, you must set the design planning options, as described in the following steps:

1. In Design Compiler Graphical, open the GUI by entering the `gui_start` command.
2. In the GUI, open the layout window by choosing `Window > New Layout Window`.
3. Choose `Floorplan > Set Design Planning Options`.

The Set Design Planning Options dialog box appears, as shown in [Figure 20-8](#).

Figure 20-8 Set Design Planning Options Dialog Box



4. In the Work Directory box, enter the name of your working directory with the relative path from the current directory.

This is where the floorplanning scripts, the floorplan, and any other necessary files are stored. If you do not provide the path to a specific directory, the files are stored in the directory created at the \$TMPDIR/dcg_unique_string location if you specified a location with the TMPDIR UNIX environment variable. If you did not specify a location with the TMPDIR variable, the files are stored in the /tmp directory location.

5. In the ICC Executable box, enter the name and location of the IC Compiler executable.

By default, Design Compiler uses the `icc_shell` executable specified by the `$path` variable.

6. In the File Name Prefix box, enter the prefix you want to use in the file name for any generated files, such as the floorplanning scripts and floorplan files.

The default file prefix naming style is `%s_%d` where `%s` is the design name and `%d` is the process ID.

7. (Optional) Select the Keep Files option if you want to retain the files that are created during and after the floorplan exploration session, such as the floorplanning scripts, floorplan files, and so on.

This option is deselected by default, which means that the generated files are removed when the floorplan exploration session is closed.

8. Click OK.

You can set the design planning options at the command line by using the `set_icc_dp_options` and `start_icc_dp` commands. For more information, see [Using Floorplan Exploration With a dc_shell Script](#).

Running Floorplan Exploration

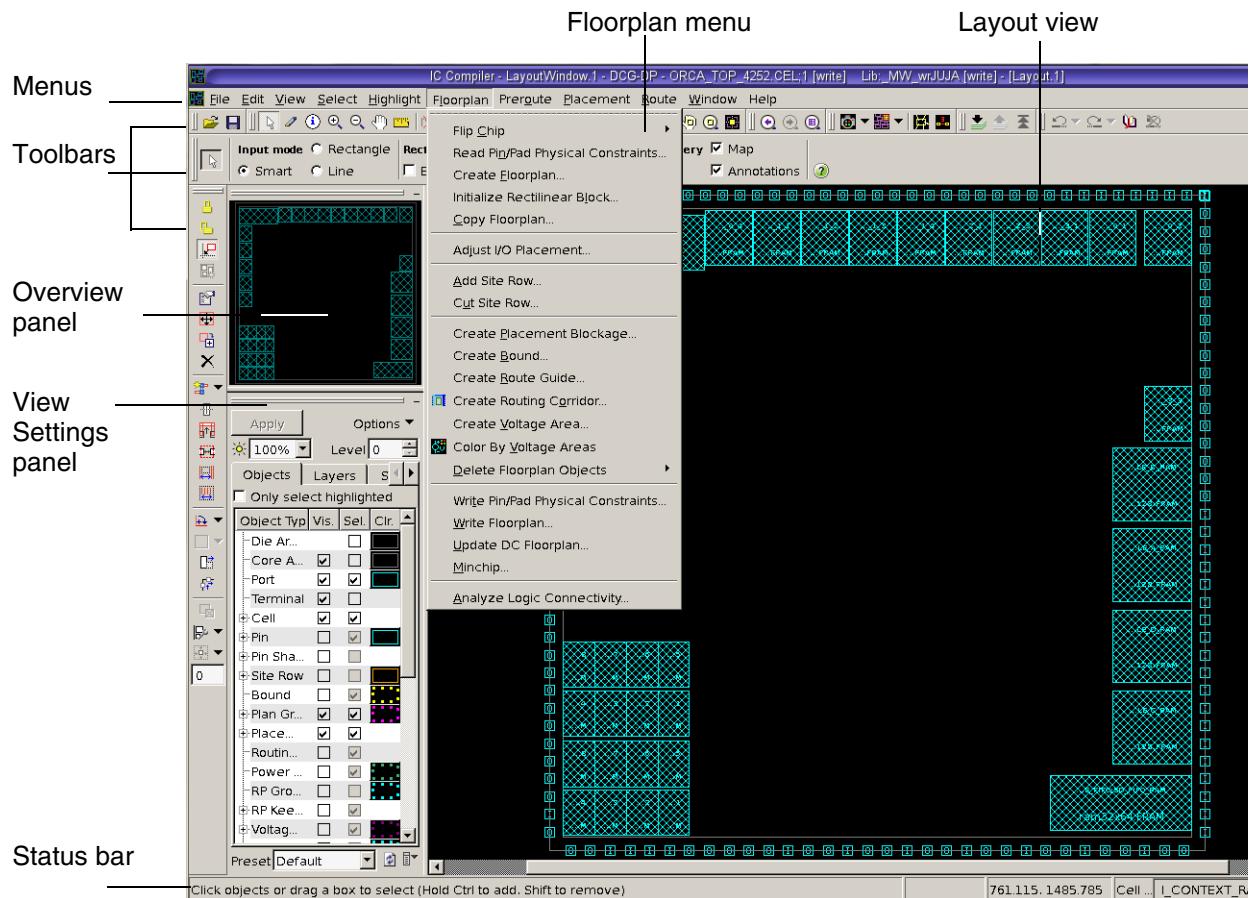
After you set the design planning options as described in [Enabling Floorplan Exploration](#), you can start a floorplan exploration session by choosing Floorplan > Start Design Planning in the GUI. When you do this, the following actions are performed automatically:

- Design Compiler Graphical invokes floorplan exploration with simplified floorplanning menus in the IC Compiler layout window.
- Design Compiler Graphical transfers the design and its floorplan to the IC Compiler layout window.
- Design Compiler Graphical transfers all SDC constraints, attributes, and library setup files to IC Compiler for floorplanning.
- All Design Compiler Graphical windows and the Design Compiler Graphical shell are disabled until you exit the IC Compiler floorplanning session.

Using the Floorplan Exploration GUI

Floorplan exploration in Design Compiler Graphical uses the IC Compiler layout window. By default, when you enable floorplan exploration, Design Compiler Graphical configures the IC Compiler layout window so that it includes only the menus and menu commands you need to perform Design Compiler Graphical floorplanning. [Figure 20-9](#) shows the simplified floorplanning menu structure.

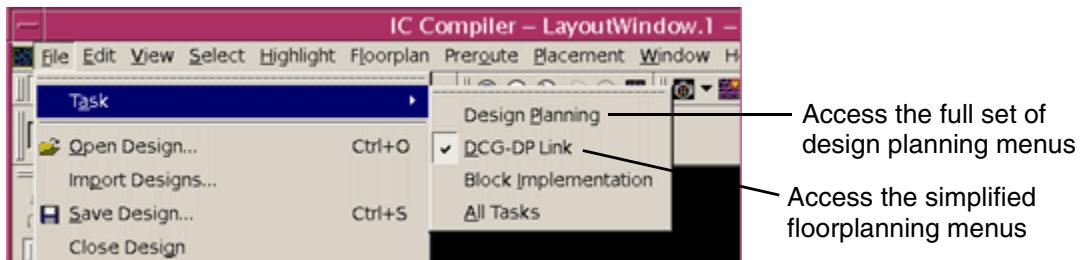
Figure 20-9 IC Compiler Simplified Layout Window Features



If you are an experienced IC Compiler user, you can use the full set of design planning menus.

You can access the full set of IC Compiler design planning menus by choosing File > Task > Design Planning, as shown in [Figure 20-10](#). To switch from the full design planning menu structure back to the simplified menu structure, choose File > Task > DCG-DP Link.

Figure 20-10 Switching Between Simplified and Full Design Planning Menu Structures



Any work you complete using the full set of IC Compiler design planning menus is available when you switch back to the simplified floorplanning menus.

For general information about working in the IC Compiler layout window, see the IC Compiler Help system and the *IC Compiler Design Planning User Guide*.

Creating and Editing Floorplans

Whether you use the simplified Design Compiler Graphical floorplanning menus or the full set of IC Compiler design planning menus, the IC Compiler layout window provides interactive tools and commands that you can use to create or modify your floorplan. You can

- Use editing tools to move, resize, copy, split, reshape, align, distribute, spread, or remove objects.
- Use editing commands to rotate or expand objects or to change cell orientations.

Shortcut keys are available for the most frequently performed editing operations. Online Help pages are provided for each of the editing tools.

In addition, you can use commands on the Floorplan menu to create physical objects such as placement blockages, bounds, and routing keepouts. When you create an object, you can specify its coordinates or draw the object in the layout view.

For more information about working with editing tools and commands in the IC Compiler layout window, see the IC Compiler Help system. For sample flows that show how to create a floorplan using Design Compiler Graphical floorplan exploration and how to modify a floorplan to improve timing or congestion, see [SolvNet article 030613, "Design Compiler Graphical Floorplan Exploration Application Note."](#)

Analyzing the Data Flow for Macro Placement

Floorplan exploration allows you to analyze and improve the data flow through the physical design using the IC Compiler design planning data flow analysis tools. Information about the data flow can help you to decide what to do next to improve the placement of hard macros in your design.

After performing your initial macro placement in IC Compiler, you should analyze your floorplan to find areas of high congestion. If your macro placement appears to be causing congested hot spots, you can use the data flow analysis tools to examine hard macro placement and connectivity and to edit the macro placement.

To open the data flow analyzer window, choose Floorplan > Analyze Logic Connectivity. Alternatively, you can use the `analyze_logic_connectivity` command.

For information about using the data flow analysis tools, see the IC Compiler Help system and the *IC Compiler Implementation User Guide*.

Saving the Floorplan or Discarding Updates

You can save or discard floorplan updates in Design Compiler Graphical from the IC Compiler floorplanning session at any time using the “Update DC Floorplan” command. To save or discard the floorplan, perform the following steps:

1. Choose Floorplan > Update DC Floorplan.

The Update DC Floorplan dialog box appears, as shown in [Figure 20-11](#).

Figure 20-11 Update DC Floorplan Dialog Box



The “Update DC Floorplan” command is available only in the simplified floorplanning menu structure for Design Compiler Graphical. For information about switching from the full set of design planning menus to the simplified floorplanning menu structure, see [Using the Floorplan Exploration GUI](#).

2. Choose one of the following options:

- To save the current floorplan in Design Compiler Graphical, select the “Save current floorplan for DC” option.

- If you have previously saved a floorplan, the tool overwrites it.
- To remove a previously saved floorplan, select the “Discard DC floorplan updates” option.
3. Click OK.

Saving the Floorplan into a Tcl Script File or DEF File

You can also save the floorplan into a Tcl script file or a DEF file during the floorplan exploration session. However, the Tcl or DEF file is not automatically read in Design Compiler Graphical.

To save the floorplan into a Tcl script file, choose Floorplan > Write Floorplan, and specify the required options in the Write Floorplan dialog box.

Alternatively, you can use the `write_floorplan` command with its required options at the tool prompt.

To save the floorplan in a DEF file, choose File > Export > Write DEF, and specify the required options in the Write DEF dialog box.

Alternatively, you can use the `write_def` command with its required options at the tool prompt.

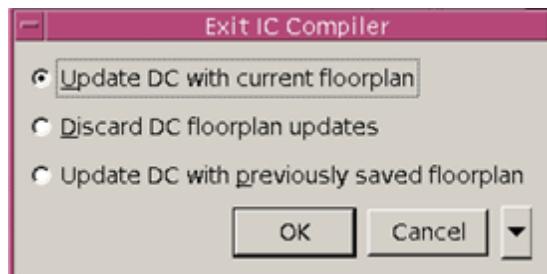
Exiting the Session

To exit the floorplan exploration session and save your floorplan for synthesis or discard the floorplan, perform the following steps:

1. Choose File > Exit in the menu.

The Exit IC Compiler dialog box appears, as shown in Figure 20-12.

Figure 20-12 Exit IC Compiler Dialog Box



2. Choose one of the following options:

- Select the “Update DC with current floorplan” option to save the current floorplan in Design Compiler Graphical and exit the floorplan exploration session.
- To exit the floorplan exploration session and discard the floorplan, including any previously saved floorplans, select the “Discard DC floorplan updates” option.
- To keep the floorplan that you saved previously in Design Compiler Graphical, discard any subsequent changes that you have not saved, and exit the floorplan exploration session, select the “Update DC with previously saved floorplan” option.

3. Click OK.

When you exit the floorplan exploration session, the Design Compiler Graphical windows and shell interface become active.

If you update the floorplan, the tool closes the original Design Vision layout window. To see the floorplan changes, open a new Design Vision layout window.

Incremental or Full Synthesis After Floorplan Changes

The floorplan changes affect the design during synthesis. After you create a floorplan or modify an existing floorplan using floorplan exploration, you need to update the floorplan constraints by using the `write_floorplan` command, as shown, before continuing with any additional synthesis runs:

```
dc_shell-topo> write_floorplan -all DESIGN.fp
```

Then, use the `read_floorplan` command to read the floorplan in Design Compiler for synthesis. After you read the RTL file, the SDC constraints, and the new floorplan file, you can perform incremental synthesis by using the `compile_ultra -incremental` command for a quick assessment of the floorplan’s QoR benefits. Incremental synthesis might not meet the requirements for achieving good QoR results or the requirements for a final implementation flow. In these cases, perform full synthesis using the `compile_ultra` command.

Design Compiler Graphical also allows you to make floorplan changes interactively on the command line between the initial `compile_ultra -spg` step and subsequent `-incremental -spg` compilation steps without updating the floorplan constraints with the `write_floorplan` command. In this case, you must use the supported floorplan modification commands described in [SolvNet article 1502496, “Making Floorplan Changes After Running the compile_ultra -spg Command.”](#) However, the tool issues a warning message, indicating that it detected floorplan changes.

See Also

- [Using Floorplan Physical Constraints](#)

Provides information about the `read_floorplan` and `write_floorplan` commands

- [Overview of Topographical Technology](#)

Provides information about synthesizing the design in topographical mode

Using Floorplan Exploration With a dc_shell Script

You can perform floorplan exploration from a script. Scripts allow you to create and modify floorplans on the command line.

Use the following commands in Design Compiler Graphical to perform floorplan exploration with a script:

1. Use the `set_icc_dp_options` command to specify the IC Compiler executable and other setup requirements.

The following options are available with the `set_icc_dp_options` command:

- `-work_dir directory`

Specifies the directory where the temporary files are created and stored.

If you do not provide the path to a specific directory, the files are stored in the directory created at the `$TMPDIR/dcg_unique_string` location if you specified a location with the `TMPDIR` UNIX environment variable. If you did not specify a location with the `TMPDIR` variable, the files are stored in the `/tmp` directory location.

- `-icc_executable executable`

Specifies the path to `icc_shell`.

- `-check`

Performs permission checks and reports any errors that would prevent the `start_icc_dp` command from successfully launching the floorplan exploration session.

- `-file_name_prefix file_name`

Specifies the prefix for the name of any generated files.

- `-keep_files`

Specifies to keep the temporary files.

2. (Optional) Use the `report_icc_dp_options` command to report the options specified by the `set_icc_dp_options` command.

If you do not specify any options using the `set_icc_dp_options` command, `report_icc_dp_options` reports the default settings.

The following options are available with the `report_icc_dp_options` command:

- `-check`

Performs permission checks and reports any errors that would prevent the `start_icc_dp` command from successfully launching the floorplan exploration session.

- `-verbose`

Prints verbose messages.

3. Use the `start_icc_dp` command to launch the floorplan exploration session.

The `start_icc_dp` command uses the IC Compiler executable specified by the `set_icc_dp_options` command. To source a script, use the `-f` option with the `start_icc_dp` command:

```
dc_shell-topo> start_icc_dp -f ../../scripts/script_file_name.tcl
```

The `start_icc_dp` command sources the file specified with the `-f` option and runs the script in floorplan exploration.

The following options are available with the `start_icc_dp` command:

- `-check_only`

Performs permission checks and reports any errors that would prevent the `start_icc_dp` command from successfully launching the floorplan exploration session.

- `-verbose`

Prints verbose messages.

- `-f file_name`

Specifies the path to the script file to be sourced.

Note:

If you run floorplan exploration from a batch script, by using the `-f` option on the command line when you start the tool, the `start_icc_dp` command closes the GUI automatically before starting the floorplan exploration session.

To ensure that IC Compiler saves the floorplan to Design Compiler Graphical when exiting the floorplan exploration session, you must include the `update_dc_floorplan` command in the IC Compiler floorplanning script after the floorplan creation or modification steps. After

the `update_dc_floorplan` command, you must include the `exit` command in the IC Compiler floorplanning script to return to the Design Compiler Graphical shell interface:

```
# After many IC Compiler floorplanning script commands:  
update_dc_floorplan  
exit
```

The following example shows a Design Compiler Graphical script used in a floorplan exploration session. It sources an IC Compiler floorplanning script called `iccdp_script.tcl`.

```
compile_ultra -scan  
set_icc_dp_options -icc_executable \  
"/global/apps5/icc_2010.03-SP1-1/bin/icc_shell"  
start_icc_dp -f iccdp_script.tcl  
compile_ultra -scan -incremental
```

Black Box Support

Design Compiler Graphical supports floorplan exploration with netlists that contain black box cells. If a design has black box cells, you do not need to perform any additional steps or modify the floorplan exploration flow. The black boxes that are defined or created in Design Compiler Graphical are transferred to IC Compiler design planning automatically. Using IC Compiler, you can edit the floorplan with black boxes during the floorplanning session and transfer the modifications back to Design Compiler Graphical.

See Also

- [Handling Black Boxes](#)
-

Handling Physical Hierarchies and Block Abstractions

Design Compiler Graphical transfers Design Compiler topographical physical hierarchies and Design Compiler topographical block abstractions to IC Compiler design planning automatically during floorplan exploration. Both physical hierarchies and block abstractions are transferred as macros with quick timing models in the block. For multicorner-multimode designs, only the current scenario timing information is transferred to IC Compiler.

When transferring Design Compiler topographical physical hierarchies and block abstractions to IC Compiler for design planning, Design Compiler Graphical automatically sets the `dct_physical_hier` attribute to `true` on the instances.

Querying Hierarchical Blocks and Pin Connections

To return all the Design Compiler topographical hierarchical blocks, including physical hierarchies and block abstractions, use the `dct_physical_hier` attribute as shown:

```
icc_shell> get_cells -hierarchical -filter dct_physical_hier==true  
{GPRs Multiplier}
```

The Design Compiler topographical physical hierarchies and block abstractions are preserved when brought back to Design Compiler Graphical.

To query the nets connected to the pin of a hierarchical block, use the `-boundary_type lower` or `-boundary_type upper` option with the `get_nets` command for the net inside or outside the hierarchical block, respectively. To query both nets, specify the `both` keyword. If you do not specify the `-boundary_type` option, the command returns the net outside the hierarchical block. When using this option, you must specify a pin by using the `-of_objects` option.

For example, the following two commands return the topnet net connected to the in pin outside the u2 hierarchical block:

```
dc_shell-topo> get_nets -of_objects u2/in  
{topnet}  
dc_shell-topo> get_nets -of_objects u2/in -boundary_type upper  
{topnet}
```

The following command returns the u2/in net connected to the in pin inside the u2 hierarchical block:

```
dc_shell-topo> get_nets -of_objects u2/in -boundary_type lower  
{u2/in}
```

See Also

- [Using Hierarchical Models](#)

Floorplan Exploration Floorplanning With IC Compiler II

Design Compiler Graphical allows you to perform floorplan exploration using the IC Compiler II design planning tools in the IC Compiler II layout window. After synthesis, you can evaluate the quality of results and use floorplan exploration to create a floorplan or improve an existing floorplan in IC Compiler II. The interface between floorplan exploration in Design Compiler and the IC Compiler II layout window is seamless.

- [License Requirements](#)
- [Prerequisites for Floorplan Exploration](#)
- [Command Summary](#)

- [Running the Floorplan Exploration Flow](#)
- [Data Transfer to IC Compiler II](#)

See Also

- [Floorplan Exploration Floorplanning With IC Compiler](#)
- [SolvNet article 030613, “Design Compiler Graphical Floorplan Exploration Application Note”](#) for sample flows that show how to create a floorplan using Design Compiler Graphical floorplan exploration and how to modify a floorplan to improve timing or congestion

License Requirements

Running the floorplan exploration flow using the `start_icc2` command requires an IC Compiler II license in addition to a Design Compiler Graphical license. For licensing details, contact your Synopsys representative.

Prerequisites for Floorplan Exploration

Before performing floorplan exploration in IC Compiler II, you must perform the following tasks:

- Synthesize the design
 - You must run IC Compiler II on a synthesized (mapped) design.
- Generate IC Compiler II reference libraries
 - Because the reference libraries are not automatically converted by Design Compiler, you must generate the libraries using the IC Compiler II Library Manager.
- Specify the golden UPF if the golden UPF flow is enabled in Design Compiler

Command Summary

The following table summarizes the commands to use floorplan exploration in IC Compiler II.

Table 20-1 Commands to Use Floorplan Exploration in IC Compiler II

Command	Description
<code>set_icc2_options</code>	Specifies the IC Compiler II executable, IC Compiler II reference libraries, and other setup requirements.
<code>report_icc2_options</code>	Reports the options specified by the <code>set_icc2_options</code> command.
<code>start_icc2</code>	Launches a design planning session in IC Compiler II.
<code>update_dc_floorplan</code>	Saves the floorplan created or edited in the IC Compiler II design planning session to Design Compiler Graphical. Note: The <code>update_dc_floorplan</code> command works only in an IC Compiler II session invoked using <code>start_icc2</code> .

When you run the `start_icc2` command, the files needed to invoke IC Compiler II are written to the `$TMPDIR/dcguuid` working directory if you specified a location with the `TMPDIR` UNIX environment variable. If you did not specify a location with the `TMPDIR` variable, the files are stored in the `/tmp` directory location. To change the default directory, specify the `-work_dir` option with the `set_icc2_options` command.

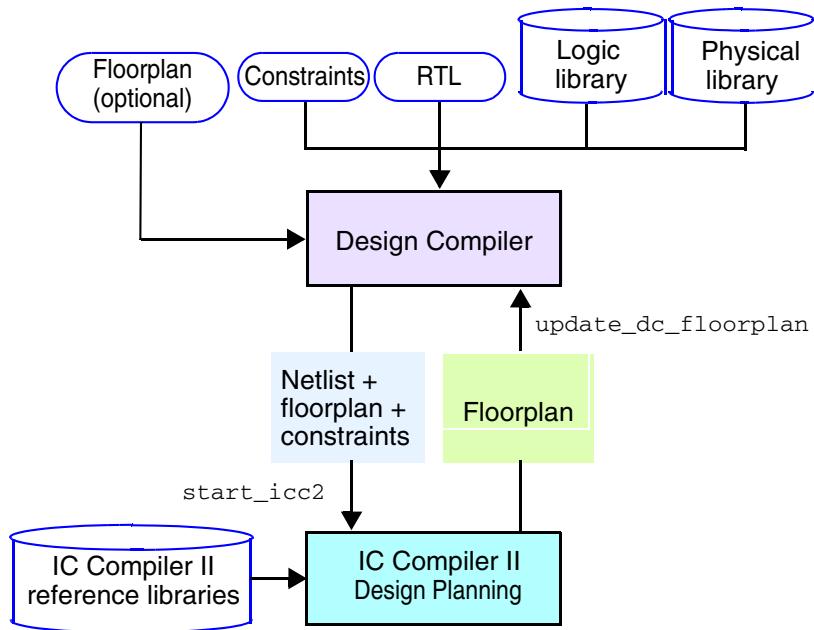
Note:

Do not use the `start_icc2` and `start_icc_dp` commands in the same Design Compiler session.

Running the Floorplan Exploration Flow

The following figure shows an overview of floorplan exploration from Design Compiler Graphical to design planning in IC Compiler II.

Figure 20-13 Flow From Design Compiler Graphical to Floorplan Exploration in IC Compiler II



Before running floorplan exploration, you must complete the following tasks:

1. Synthesize the design by using the `compile_ultra` command.
2. Specify IC Compiler II reference libraries by using the `set_icc2_options` command.

For example,

```
dc_shell-topo> set_icc2_options -icc2_executable ../../icc2_shell \
    -ref_libs "lib1.ndm lib2.ndm"
```

When you create a DEF file in Design Compiler NXT or Design Compiler Graphical to hand off a design while invoking IC Compiler II, the site name in the DEF file is from the Milkyway reference library. When you read this DEF file in the IC Compiler II link, mismatch between the site name in the Milkyway reference library and the tile name in technology file might occur resulting in the following DEFR-004 error message:

```
Error: Cannot find site 'unit'. (DEFR-004)
```

To avoid the tool from issuing the DEFR-004 error message due to site name mismatch, use the `-convert_sites {from_site to_site}` option as shown in the following example:

```
prompt> set_icc2_options -ref_libs "lib1.ndm" -technology tech.tf \
-converst_sites {unit CORE}
```

In this example,

- `from_site` (unit) is the site name in the Milkyway reference library
- `to_site` (Core) is the tile name in the technology file that is passed to the IC Compiler II tool

The `set_icc2_options -convert_sites` command converts the site name from "unit" to "CORE" when the tool reads the DEF file after invoking IC Compiler II and issues the following message:

```
Information: Converted site definition 'unit' to 'CORE'. (DEFR-031)
```

The following sections describe the floorplan exploration steps in interactive mode and in batch mode using a `dc_shell` script.

In Interactive Mode

To start the floorplan exploration flow,

1. In Design Compiler Graphical, start an IC Compiler II design planning session by using the `start_icc2` command.

The `start_icc2` command opens the IC Compiler II layout window and the `icc2_shell>` prompt is displayed.

2. In IC Compiler II, perform floorplan exploration.

You can create a floorplan or edit an existing floorplan using the design planning tools in the IC Compiler II layout window.

3. Save the updated floorplan by using the `update_dc_floorplan` command in `icc2_shell`.

The floorplan data in IC Compiler II is transferred to Design Compiler Graphical. If you do not perform this step, the floorplan changes performed in IC Compiler II are not updated in Design Compiler Graphical and are lost.

4. (Optional) To preserve the floorplan files in step 3, specify the `-keep_files` option with the `set_icc2_options` command.

5. End the design planning session by entering the `exit` command in `icc2_shell`.

You are returned to the `dc_shell>` prompt.

In Batch Mode

To run floorplan exploration from a batch script, specify the `-f script_file` option with the `start_icc2` command when you start Design Compiler Graphical. In batch mode, the `start_icc2` command invokes IC Compiler II and executes the specified script, but it does not open the GUI.

The following script shows an example of the general floorplan exploration flow with IC Compiler II:

```
#Perform synthesis using compile_ultra
compile_ultra -scan -gate_clock -spg

#Set the IC Compiler II options
set_icc2_options -icc2_executable ../icc2_shell \
-ref_libs "lib1.ndm lib2.ndm"

#Launch IC Compiler II session in batch mode
start_icc2 -f my_icc2.tcl
```

my_icc2.tcl batch script example:

```
#This example script moves the macro I_RAM by 10 microns
set_placement_status placed [get_cells I_RAM]
move_objects -delta {10 10} [get_cells I_RAM]
set_placement_status fixed [get_cells I_RAM]

#Update the floorplan changes before exit
update_dc_floorplan
exit
```

Data Transfer to IC Compiler II

When you launch IC Compiler II within Design Compiler Graphical, design data is automatically transferred to IC Compiler II. The following table shows which design and environment settings are and are not transferred from Design Compiler Graphical to IC Compiler II.

Table 20-2 Design and Environment Settings That Are and Are Not Transferred to IC Compiler II

Transfer	Design and environment settings
Transferred	<p>Verilog netlist</p> <p>Timing constraints (in a format compatible with IC Compiler II)</p> <p>Floorplan constraints using a DEF file and a Tcl floorplan file that is compatible with IC Compiler II</p> <p>Technology file</p> <p>TLUPlus file</p> <p>UPF file</p> <p>Attributes:</p> <ul style="list-style-type: none"> • dont_touch • size_only • dont_use on library cells • Note: Other attributes need to be reapplied in IC Compiler II <p>Layer-related constraints</p> <ul style="list-style-type: none"> • Net routing layer constraints • Preferred routing directions • Ignored layer settings
Not transferred	<p>Hierarchical models</p> <ul style="list-style-type: none"> • Design Compiler and IC Compiler block abstractions and physical hierarchies are treated as unresolved references in icc2_shell. <p>Estimations of physical black boxes</p> <ul style="list-style-type: none"> • These physical cells are created by the <code>estimate_fp_black_boxes</code> command. <p>Relative placement constraints</p> <ul style="list-style-type: none"> • These constraints are created in Design Compiler.

Working With Via Ladders

A via ladder, also known as a via pillar, is a stacked via that starts from the pin layer and extends into an upper layer where the router connects to it. Via ladders are used to improve timing QoR and electromigration robustness.

To learn how to work with via ladders in the Design Compiler Graphical tool, see the following topics:

- [Reading In Via Ladders Defined in the Technology File](#)
 - [Creating, Querying, Removing, and Reporting Via Ladders](#)
 - [Performing What-If Analysis With Via Ladders](#)
 - [Performing Via-Ladder-Aware Optimization](#)
-

Reading In Via Ladders Defined in the Technology File

Via ladders are usually defined in the technology file and read in by the tool. If via ladders are not defined in the technology file, you can create them with Tcl commands.

The following example shows a via ladder defined in the technology file:

```
ViaRule      "VL3" {  
    cutLayerNameTbl = (VIA1,VIA2,VIA3)  
    cutNameTbl     = (Vs,Vs,Vs)  
    numCutRowsTbl  = (2,2,1)  
    numCutsPerRowTbl = (1,2,2)  
}
```

Creating, Querying, Removing, and Reporting Via Ladders

If via ladders are not defined in the technology file, you can use the following commands to create, query, report, and remove via ladders:

- `create_via_rule`—Creates new via ladders.
- `get_via_rules`—Returns the via rules created by the `create_via_rule` command and the via rules read in from the technology file.
- `report_via_rules`—Reports the via ladder rules created by the `create_via_rule` command and the via rules read in from technology file.
- `remove_via_rules`—Removes via rules created by the `set_via_ladder_rules` command.

There are two flows for using via ladders:

- The “what-if analysis” flow in which you manually associate a pin with a via ladder and check its impact on timing
- The “via-ladder-aware optimization” flow in which you allow the tool to infer via ladders during synthesis to improve timing QoR or electromigration robustness.

Performing What-If Analysis With Via Ladders

In the what-if analysis flow, you manually associate a pin with a via ladder and check its impact on timing.

Before starting the what-if analysis flow,

1. Confirm that via ladders are defined in the technology file or create them with Tcl commands.
2. Enable the use of enhanced timing models during RC estimation by setting the `spg_enhanced_timing_model` variable to `true`.
3. Enable via resistance support for the driver pin of the cells by setting the `physopt_enable_root_via_res_support` variable to `true`.
4. Compile the design with the `compile_ultra -spg` command.

Follow these steps to perform what-if analysis with via ladders:

1. Set a via ladder on a pin of a cell instance or on a pin of a library reference by using the `set_via_ladder_constraints` or `set_via_ladder_rules` command.
2. (Optional) Report the via ladder settings by using the `report_via_ladder_constraints` or `report_via_ladder_rules` command.
3. (Optional) Remove the via ladder settings by using the `remove_via_ladder_constraints` or `remove_via_ladder_rules` command.
4. Re-estimate the timing of the design or timing path using the `extract_rc -estimate` command.
5. Check the timing impact on the design or path using the `report_timing` and `report_qor` commands. A “v” symbol in the report indicates via ladders.

Performing Via-Ladder-Aware Optimization

In the via-ladder-aware optimization flow, you allow the tool to infer via ladders during synthesis.

Before starting the via-ladder-aware flow,

1. Confirm that via ladders are defined in the technology file or create them with Tcl commands.
2. Enable the use of enhanced timing models during RC estimation by setting the `spg_enhanced_timing_model` variable to `true`.
3. Enable via resistance support for the driver pin of the cells by setting the `physopt_enable_root_via_res_support` variable to `true`.
4. Enable via-ladder optimization by setting the `spg_enable_via_ladder_opto` variable to `true`.

Follow these steps to perform via-ladder-aware optimization:

1. Specify the via ladder candidates on library pins by using the `set_via_ladder_candidate` command.
2. (Optional) Report the via ladder candidates by using the `report_via_ladder_candidates` command.
3. (Optional) Remove the via ladder candidates by using the `reset_via_ladder_candidate` command.
4. (Optional) Specify the `is_em_via_ladder_required` attribute on the library pin if electromigration via ladders are required to improve electromigration robustness.
5. Run synthesis in physical guidance mode.
6. (Optional) Report via ladders in the design after optimization by using the `report_via_ladder_constraints` command.
7. Provide via ladders and settings to the IC Compiler II tool by using the `write_icc2_files` command.

The `report_timing` report displays a “v” to indicate where there are via ladders on pins. You can also report the via ladders added during optimization by using the `report_via_ladder_constraints` command.

For more details, see [SolvNet article 2624185 “Via Ladder Support in Design Compiler Graphical.”](#)

Assigning of Via Ladders on Nets

To control the assignment of via ladders on nets, use the following variables:

- To set a minimum length threshold on the nets considered for via ladder assignment during optimization, use the `optimize_via_ladder_net_length_threshold` variable.

The value you specify is multiplied by the minimum height of the unit tile. For example, if you set the variable to 125, the minimum length threshold for nets considered for via ladder assignment is 125 times the minimum height of the unit tile. The default is 100.

```
dc_shell-topo> set_app_var optimize_via_ladder_net_length_threshold  
125
```

- To choose via ladders on driver pins in a timing path within a specific slack range during optimization, use the `optimize_via_ladder_critical_range` variable.

The driver pins in the timing path that fall within the following range are considered for via ladder insertion:

```
[WNS, WNS*(1 - optimize_via_ladder_critical_range)]
```

By default, the `optimize_via_ladder_critical_range` variable is 0.5. Therefore, if the worst negative slack (WNS) is -1000, the range is [-1000, -500] where $-500 = -1000 * (1 - 0.5)$. If the delay value falls into this range, it is considered critical.

The tool chooses the driver pins for via ladder insertion when you run the `compile_ultra -spg` or `optimize_netlist -area` command.

Working With Must-Join Pins

Some standard cells have multiple output terminals that must be connected together. When these must-join pins have a `pattern_must_join` attribute, the IC Compiler II router uses a predefined pattern to join the pin shapes together. The Design Compiler Graphical tool can recognize `pattern_must_join` attributes on library pins in the Milkyway FRAM view and perform RC extraction taking the `pattern_must_join` attributes into consideration.

To enable RC estimation based on `pattern_must_join` attributes in the Design Compiler Graphical tool,

1. Make sure the Milkyway library has pins with the `pattern_must_join` attribute.
2. Enable the following variables. The variables are `false` by default.
 - a. Enable the enhanced timing model:


```
dc_shell-topo> set_app_var spg_enhanced_timing_model true
```
 - b. Enable via resistance support for the driver pin of the cells:

```
dc_shell-topo> set_app_var physopt_enable_root_via_res_support true
```

- c. Enable via ladder optimization and pattern_must_join attribute recognition:

```
dc_shell-topo> set_app_var spg_enable_via_ladder_opto true
```

- 3. Synthesize the design in physical guidance mode:

```
dc_shell-topo> compile_ultra -scan -spg
```

When you enable the variables, the tool performs RC estimation based on the pattern_must_join attributes during optimization. If a library pin has both a pattern_must_join attribute and via ladder candidates, the via ladder has higher priority if the estimated delay based on the pattern_must_join attribute and the via ladder is the same.

The pattern_must_join attributes are stored in the .ddc file and available in future Design Compiler Graphical sessions.

See Also

- [Querying and Reporting the pattern_must_join Attribute](#)
- [Checking the Inconsistency of pattern_must_join Attributes on Library Pins](#)

Querying and Reporting the pattern_must_join Attribute

You can query the pattern_must_join attribute as shown:

- dc_shell-topo> get_attribute [get_pins */BUFF/Z] pattern_must_join
- dc_shell-topo> get_attribute [get_lib_pins */BUFF/Z] pattern_must_join

You can get a list of pins that have the pattern_just_join attribute as shown:

```
dc_shell-topo> get_lib_pins my_library/*/* \
    -filter {pattern_must_join == true}
```

You can report the cells that have a pattern_must_join attribute as shown:

```
dc_shell-topo> get_cells -of_objects [get_pins -hierarchical \
    -filter {pattern_must_join == true}]
```

The report_timing report also reports the pattern_must_join attribute. A “v” in the report indicates a cell with the pattern_must_join attribute.

Checking the Inconsistency of pattern_must_join Attributes on Library Pins

You can report inconsistencies among `pattern_must_join` attributes on library pins in the Design Compiler Graphical tool by using the following options with the `report_lib` command:

- `-pattern_must_join_pin`

Reports inconsistencies in `pattern_must_join` attributes that are set on library pins. Inconsistencies that are reported include

- Output pins with multiple shapes on the same metal layer that do not have a `pattern_must_join` attribute set on them
- Output pins with multiple shapes on different metal layers that have a `pattern_must_join` attribute set on them

- `-pattern_must_join_pin_exclusion_list`

Excludes pins from being reported when specified with the `-pattern_must_join_pin` option

Before you use the `report_lib` command to report `pattern_must_join` attribute inconsistencies, you must

1. Ensure that the Library Compiler tool is installed and its executable is located in the Design Compiler installation directory
2. Enable via ladder optimization and `pattern_must_join` attribute recognition by setting the `spg_enable_via_ladder_opto` variable to `true`

The following example reports all inconsistencies among `pattern_must_join` attributes in the specified library:

```
dc_shell-topo> report_lib library -pattern_must_join_pin
*****
Report : library
...
*****
#BEGIN_REPORT_PATTERN_MUST_JOIN_PIN
-----
target library: my_lib
-----
Report of single metal layer multi-terminals ports without
pattern-must-join setting:
Warning: All terminals of the port 'ZN' of cell 'xyz' are on a
single metal layer 'M1'. However, the 'pattern_must_join' attribute
is not set as true on this port, which makes conflicts. (FRAM-075)
Total number of cells with single metal layer multi-terminals port
without pattern-must-join setting: 1 (out of 197)
```

```
Report of pattern-must-join ports with terminals on multiple metal
layers:
Total number of cells with terminals on multiple metal layers but with
pattern must-join ports: 0 (out of 0)

#END_REPORT_PATTERN_MUST_JOIN_PIN
```

The following example excludes the Z pin in the XOR2 library cell from the report:

```
dc_shell-topo> report_lib library -pattern_must_join_pin \
    -pattern_must_join_pin_exclusion_list [get_lib_pins */XOR2/Z]
```

Inbound Cell Optimization

Inbound cells are standard cells that extend beyond PG pins in placement rows so that the PG pins are completely inside the cell boundary. Inbound cells are generally faster than other library cells, but they pose challenges during legalization and placement. For example, same-color edges cannot be abutted, and inbound cells lead to half-row fragmentation during placement because they extend beyond placement rows.

The Design Compiler tool generally uses inbound cells during synthesis for better post-synthesis QoR. However, the QoR can degrade after placement, especially in high-density designs, due to half-row fragmentation and displacement as a result of the legalization constraints. Inbound cell optimization allows you to control the inbound cell percentage in the synthesized netlist to avoid QoR degradation during placement.

To enable inbound cell optimization in the Design Compiler Graphical tool, do the following before running the `compile_ultra`, `compile_ultra -incremental`, or `optimize_netlist -area` command:

1. Get the inbound cell site definition name from the technology file, and specify the inbound cell site definition name in the tool using the same name. For example:

```
dc_shell-topo> set_app_var compile_inbound_sitedef_name ibunit
```
2. Enable inbound cell optimization by setting the following variable to `true` (the default is `false`):

```
dc_shell-topo> set_app_var compile_inbound_cell_optimization true
```
3. (Optional) Limit the number of inbound cells to a specific percentage of cells by setting the following variable (by default, the tool allows up to 10 percent):

```
dc_shell-topo> set_app_var compile_inbound_max_cell_percentage 5.0
```

The percentage value can change based on the design characteristics. However, as a general guideline, limit inbound cell usage to a smaller percentage for high-density designs.

The `write_icc2_files` output includes the inbound cell site definition name for the IC Compiler II tool.

To report inbound cell usage in a design, use the `report_inbound_cell` command:

```
dc_shell-topo> report_inbound_cell
*****
Report : inbound_cell
Design : my_design
...
*****
Inbound Cell Design Summary
-----
| Inbound max cell percentage | 10.00% |
| Number of inbound cells used | 4879   |
| Percentage of inbound cells used | 9.48% |
-----
```

Optimizing Multicorner-Multimode Designs

Multicorner-multimode design optimization is a feature available only in the Design Compiler Graphical tool. This feature enables you to analyze and optimize designs across multiple modes and multiple corners concurrently.

This section describes multicorner-multimode design optimization, in the following subsections:

- [Multicorner-Multimode Concepts](#)
- [Multicorner-Multimode Feature Support](#)
- [Unsupported Features for Multicorner-Multimode Designs](#)
- [Basic Multicorner-Multimode Flow](#)
- [Creating a Scenario](#)
- [Concurrent Multicorner-Multimode Optimization and Timing Analysis](#)
- [Power Optimization in Multicorner-Multimode Designs](#)
- [Setting Up the Design for a Multicorner-Multimode Flow](#)
- [Handling Libraries in the Multicorner-Multimode Flow](#)
- [Scenario Management Commands](#)
- [Reporting Commands for Multicorner-Multimode Designs](#)
- [Supported SDC Commands for Multicorner-Multimode Designs](#)
- [Multicorner-Multimode Script Example](#)
- [Using Block Abstractions in Multicorner-Multimode Designs](#)

Multicorner-Multimode Concepts

Designs must often operate under multiple operating conditions, often called corners, and in multiple modes. Such designs are referred to as multicorner-multimode designs. Design Compiler Graphical extends the topographical technology to analyze and optimize these designs across multiple modes and multiple corners concurrently.

The multicorner-multimode feature also provides ease-of-use and compatibility between flows in Design Compiler and IC Compiler due to the similar user interface. The following terms are commonly used to describe multicorner-multimode technology:

- Corner

A *corner* is defined as a set of libraries characterized for process, voltage and temperature (PVT) variations. Corners are not dependent on functional settings; they are meant to capture variations in the manufacturing process, along with expected variations in the voltage and temperature of the environment in which the chip will operate.

- Mode

A *mode* is defined by a set of clocks, supply voltages, timing constraints, and libraries. It can also have annotation data, such as SDF or parasitics files. Multicorner-multimode designs can operate in many modes such as the test mode, mission mode, standby mode and so forth.

- Scenario

A *scenario* is a combination of modal constraints and corner specifications. In a design, the tool uses a scenario or a set of scenarios as the unit for multimode-multicorner analysis and optimization. Some constraints can be part of both the mode and corner specification. Optimization of multicorner-multimode design involves managing the scenarios of the design. For more details on scenario management, see [Scenario Management Commands](#).

Multicorner-Multimode Feature Support

The multicorner-multimode feature in Design Compiler Graphical provides compatibility between flows in Design Compiler and IC Compiler through the use of a similar user interface.

Keep the following points in mind when running multicorner-multimode optimizations:

- All options of the `compile_ultra` command are supported.
- Leakage power optimization is supported.

See [Power Optimization in Multicorner-Multimode Designs](#).

- Dynamic power optimization is supported.
See [Power Optimization in Multicorner-Multimode Designs](#)
- The UPF flow is supported only for multivoltage designs.

Unsupported Features for Multicorner-Multimode Designs

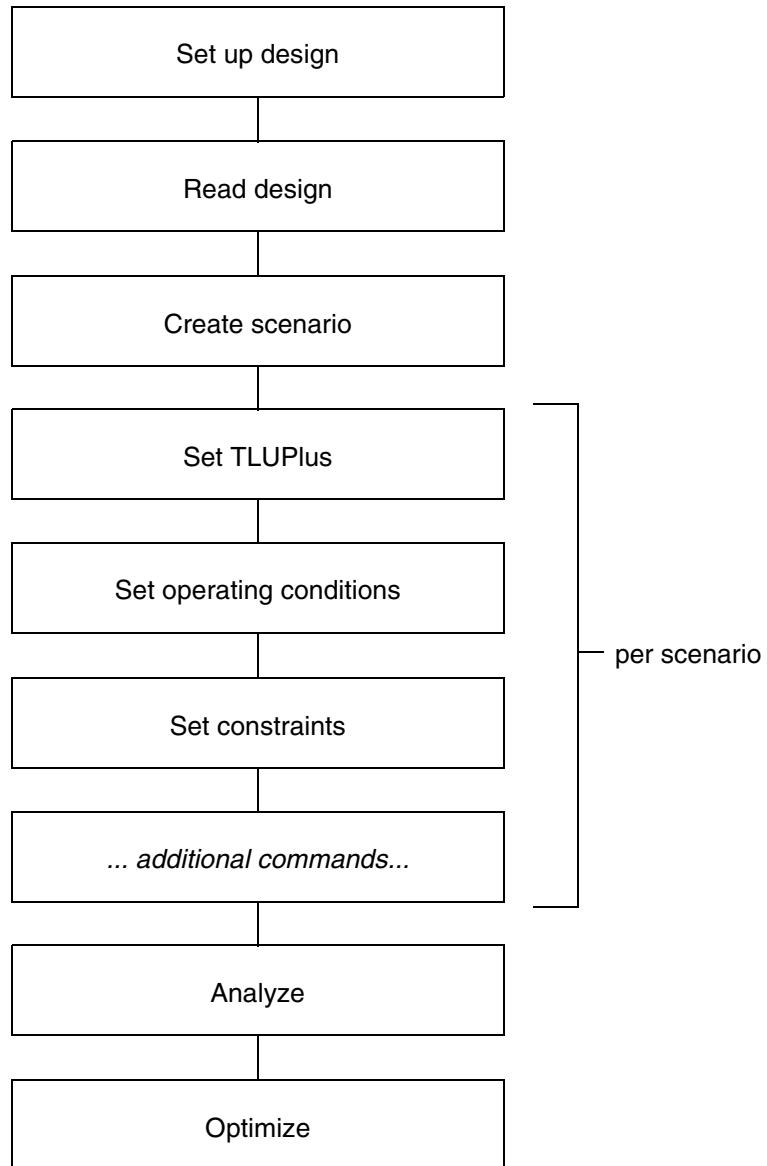
The following features are not supported in Design Compiler Graphical for multicorner-multimode designs:

- Clock tree estimation is not supported.
- k-factor scaling is not supported.
Because multicorner-multimode design libraries do not support the use of k-factor scaling, the operating conditions that you specify for each scenario must match the nominal operating conditions of one of the libraries in the list of the link libraries.
- The `set_min_library` command is not supported for individual scenarios.
The `set_min_library` command applies to all scenarios. If you use `set_min_library` to define one scenario, the tool will use the library for all scenarios.

Basic Multicorner-Multimode Flow

[Figure 20-14](#) shows the basic multicorner-multimode flow. The key step in multicorner-multimode optimization involves creating the scenarios. At a minimum, a scenario definition includes commands that specify the TLUPlus libraries, operating conditions, and constraints. You can create multiple scenarios. For each scenario, you set constraints specific to the scenario mode and you set operating conditions specific to the scenario corner. For an example script, see [Multicorner-Multimode Script Example](#).

Figure 20-14 Basic Multimode Flow



Creating a Scenario

To define modes and corners, use the `create_scenario` command. A scenario definition includes commands that specify the TLUPlus libraries, operating conditions, and constraints, as shown in [Example 20-5](#).

Example 20-5 Basic Scenario Definition

```
create_scenario s1
set_operating_conditions WORST -library stdcell.setup.typ.db:stdcell_typ
set_tlu_plus_files -max_tluplus design.tlup -tech2itf_map layermap.txt
read_sdc s1.sdc
```

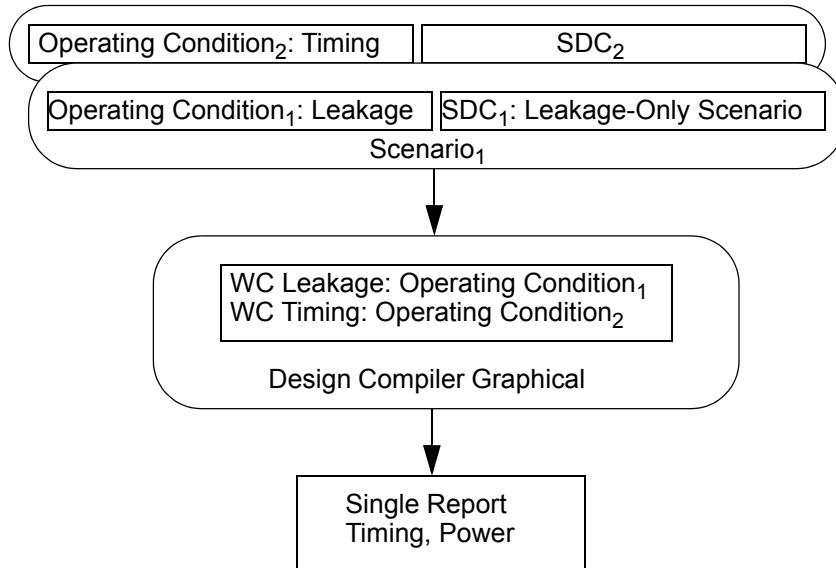
A scenario definition must include the `set_operating_conditions` and `set_tlu_plus_files` commands. The following sections describe these commands along with the associated library setup information that is needed to run multicorner-multimode design optimization.

Other commands can be included in the scenario definition. For example, you can use the `set_scenario_options -leakage true` command to control leakage power on a per-scenario basis or you can use the `read_sdf` command to set the correct net RC and pin-to-pin delay information in the respective scenarios.

After you configure all the scenarios, you can activate a subset of these scenarios by using the `set_active_scenarios` command.

[Figure 20-15](#) shows how to set various constraints on different scenarios of a multicorner-multimode design.

Figure 20-15 Setting Different Constraints on Different Scenarios



Concurrent Multicorner-Multimode Optimization and Timing Analysis

Concurrent multicorner-multimode optimization works on the worst violations across all scenarios, eliminating the convergence problems observed in sequential approaches.

Timing analysis is carried out on all scenarios concurrently, and cost is measured across all scenarios for timing and design rules. As a result, the timing and constraint reports show worst-case timing across all scenarios.

To run timing analysis, use one of the following methods:

- Traditional minimum-maximum analysis

To use this analysis method, define your analysis type as `bc_wc`, for example,

```
set_operating_conditions -analysis_type bc_wc
```

- Early-late analysis

To use this analysis method, define your analysis type as `on_chip_variation`, for example,

```
set_operating_conditions -analysis_type on_chip_variation
```

The PrimeTime tool also uses the on-chip variation (OCV) method.

- Parametric on-chip variation (POCV) analysis

In this mode, the tool computes arrival times, required times, and slack as statistical distributions rather than fixed minimum and maximum values. The timing results are less pessimistic, which provides a better basis for timing optimization, and they have better correlation with the PrimeTime tool when POCV analysis is enabled in that tool.

To use this analysis method, see “Parametric On-Chip Variation Analysis” in Chapter 4, “Operating Conditions,” in the *Synopsys Timing Constraints and Optimization User Guide*.

The PrimeTime tool also supports the POCV analysis method.

Power Optimization in Multicorner-Multimode Designs

Typically, in a multicorner-multimode design, leakage power optimization and timing optimization are done on different corners. Therefore, the worst case leakage corner might be different from the worst case timing corner. To perform leakage power optimization on specific corners, set the leakage power option on specific scenarios of the multicorner-multimode design by using the `set_scenario_options` command as follows:

```
set_scenario_options -scenarios S1 \
    -setup false \
```

```
-hold false \
-leakage_power true
```

Note:

The `get_dominant_scenarios` command is not supported in Design Compiler Graphical.

When you optimize for leakage power in multicorner-multimode designs,

- Define the leakage power option on specific scenarios targeted for leakage power optimization.
- Do not use the `set_leakage_optimization` command inside a scenario. This command is not supported for multicorner-multimode designs.

Leakage and timing optimizations can be performed concurrently across multiple scenarios. The worst case leakage corner is different from the worst case timing corner.

Note:

The `set_dynamic_optimization` command enables dynamic power optimization on all scenarios of a multicorner-multimode design. Dynamic power optimization is scenario-independent.

If no leakage scenario is defined, the average leakage value of all the scenarios is used for leakage optimization.

When you use the `set_multi_vth_constraint` command, you must specify a leakage corner using the `set_scenario_options -scenarios` command.

The following example shows how to enable leakage power on a multicorner-multimode design by specifying the `-leakage_power` option to `true`. In this example, leakage power optimization is performed on only two scenarios, `scenario_1` and `scenario_3`:

```
prompt> set_scenario_options -scenarios {scenarios_1, scenarios_3} \
          -leakage_power true
prompt> set_scenario_options -scenarios {scenarios_2, scenarios_4} \
          -leakage_power false
```

[Example 20-6](#) shows how to create a scenario and set the leakage power option on the scenario:

Example 20-6 Leakage Power Optimization in a Multicorner-Multimode Design

```
read_verilog top.v
current_design top
link
create_scenario s1
set_operating_conditions WCCOM -library slow.db:slow
set_tlu_plus_files -max_tluplus max.tlu_plus -tech2itf_map tech.map
read_sdc ./s1.sdc
set_switching_activity -toggle_rate 0.25 \
    -base_clock p_Clk -static_probability 0.015 -type inputs
set_scenario_options -scenarios s1 -setup false -hold false \
    -leakage_power true

create_scenario s2
set_operating_conditions BCCOM -library fast.db:fast
set_tlu_plus_files -max_tluplus max.tlu_plus -tech2itf_map tech.map
read_sdc ./s2.sdc

create_scenario s3
set_operating_conditions TCCOM -library typ.db:typ
set_tlu_plus_files -max_tluplus max.tlu_plus -tech2itf_map tech.map
read_sdc ./s3.sdc

create_scenario s4
set_operating_conditions NCCOM -library typ2.db:typ2
set_tlu_plus_files -max_tluplus max.tlu_plus -tech2itf_map tech.map
read_sdc ./s4.sdc
set_scenario_options -scenarios s4 -setup false -hold false \
    -leakage_power true

report_scenarios
compile_ultra -scan -gate_clock
report_power -scenarios [all_scenarios]
report_timing -scenarios [all_scenarios]
report_scenarios
report_qor
report_saif
```

See Also

- [Leakage Power and Dynamic Power Optimization](#)

Setting Up the Design for a Multicorner-Multimode Flow

To setup a design for a multicorner-multimode flow, you must specify the TLUPlus files, operating conditions, and Synopsys Design Constraints for each scenario. Design Compiler uses the nominal process, voltage, and temperature (PVT) values to group the libraries into

different sets. Libraries with the same PVT values are grouped into the same set. For each scenario, the PVT of the maximum operating condition is used to select the appropriate set. Setup considerations are described in the following sections:

- [Specifying TLUPlus Files](#)
- [Specifying Operating Conditions](#)
- [Specifying Constraints](#)

Specifying TLUPlus Files

Use the `set_tlu_plus_files` command to specify the TLUPlus files for each scenario, as shown in [Example 20-7](#).

Example 20-7 Specifying TLUPlus Files for a Scenario

```
create_scenario s1
set_operating_conditions WORST -library stdcell.setup.typ.db:stdcell_typ
set_tlu_plus_files -max_tluplus design.tlup -tech2itf_map layermap.txt
read_sdc s1.sdc
```

If you do not specify a TLUPlus file, the tool returns an error message similar to the following:

```
Error: tlu_plus files are not set in this scenario s1.
      RC values will be 0.
```

If a TLUPlus library is not correct, the tool issues the following error message:

```
Error: TLU+ sanity check failed (OPT-1429)
```

If you want to enable temperature scaling, the TLUPlus files must contain the `GLOBAL_TEMPERATURE` and `CRT1` variables. The `CRT2` variable is optional. The following example is an excerpt from a TLUPlus file:

```
TECHNOLOGY = 90nm.lib
GLOBAL_TEMPERATURE = 105.0
CONDUCTOR metal18 {THICKNESS= 0.8000
    CRT1=4.39e-3 CRT2=4.39e-7
...

```

Specifying Operating Conditions

You must define the operating condition for each scenario. You specify different operating conditions for different scenarios using the `set_operating_conditions` command, as shown in [Example 20-8](#).

Example 20-8 Specifying Operating Conditions for a Scenario

```
create_scenario s1
set_operating_conditions SLOW_95 -library max_vmax_v95_t125
set_tlu_plus_files -max_tluplus design.tlup -tech2itf_map layermap.txt
read_sdc s1.sdc
```

If you do not define an operating condition for a scenario, the tool issues MV-020 and MV-021 warnings.

Specifying Constraints

For each scenario, you must specify the design constraints specific to that scenario, as shown in [Example 20-9](#).

Example 20-9 Specifying SDC Constraints for a Scenario

```
create_scenario s1
set_operating_conditions WORST -library stdcell.setup.typ.db:stdcell_typ
set_tlu_plus_files -max_tluplus design.tlup -tech2itf_map layermap.txt
read_sdc s1.sdc
```

The tool discards any previous scenario-specific constraints after you execute the `create_scenario` command and reports an MV-020 warning, as shown in [Example 20-10](#).

Example 20-10 Tool Removes Previous Constraints

```
dc_shell-topo> create_scenario s1
Warning: Any existing scenario-specific constraints
         are discarded. (MV-020)
dc_shell-topo> report_timing
Warning: No operating condition was set in scenario s1 (MV-021)
```

Handling Libraries in the Multicorner-Multimode Flow

The following sections discuss how to handle libraries in multicorner-multimode designs:

- [Using Link Libraries That Have the Same PVT Nominal Values](#)
- [Using Unique PVT Names to Prevent Linking Problems](#)
- [Unsupported k-factors](#)
- [Automatic Detection of Driving Cell Library](#)
- [Defining Minimum Libraries](#)

Using Link Libraries That Have the Same PVT Nominal Values

The link library lists all the libraries that are to be used for linking the design for all scenarios. Furthermore, because several libraries are often intended for use with a particular scenario, such as a standard cell library and a macro library, Design Compiler automatically groups the libraries in the link library list into sets and identifies which set must be linked with each scenario.

The tool groups libraries according to the PVT value of the library. Libraries with the same PVT values are grouped into the same set. The tool uses the PVT value of a scenario's maximum operating condition to select the appropriate set for the scenario.

If the tool finds no suitable cell in any of the specified libraries, an error is reported as shown in the following example,

```
Error: cell TEST_BUFBuf_BUFA/Z (inx4) is not characterized
      for 0.950000V, process 1.000000,
      temperature -40.000000. (MV-001)
```

You should verify the operating conditions and library setup. You must fix this error before you can optimize your design.

Link Library Example

To understand how library linking works, consider [Table 20-3](#).

[Table 20-3](#) shows the libraries in the link library list, their nominal PVT values, and the operating condition, if any, that is specified in each library. The design has instances of combinational, sequential, and macro cells.

Table 20-3 Link Libraries With PVT and Operating Conditions

Link library (in order)	Nominal PVT	Operating conditions in library (PVT)
Combo_cells_slow.db	1/0.85/130	WORST (1/0.85/130)
Sequentials_fast.db	1/1.30/100	None
Macros_fast.db	1/1.30/100	None
Macros_slow.db	1/0.85/130	None
Combo_cells_fast.db	1/1.30/100	BEST (1/1.3/100)
Sequentials_slow.db	1/0.85/130	None

To create the scenario, s1, with cell instances linked to the Combo_cells_slow, Macros_slow, and Sequential_slow libraries, you run

```
dc_shell-topo> create_scenario s1
dc_shell-topo> set_operating_conditions -max WORST -library slow
```

where the library “slow” is defined within the Combo_cells_slow.db file. Note that using the `-library` option with the `set_operating_conditions` command helps the tool identify the correct PVT for the operating conditions. The PVT of the maximum operating condition is used to find the correct matches in the link library list during linking.

Using this library linking method, you can link libraries that do not have operating condition definitions. The method also enables you to have multiple library files. For example, you can have one library file for standard cells, another for macros, and so forth.

Inconsistent Libraries Warning

When you use multiple libraries, if library cells with the same name are not functionally identical or do not have identical sets of library pins with the same name and order, the tool issues a warning stating that the libraries are inconsistent.

You should run the `check_library` command before running a multicorner-multimode flow, as shown in the following example,

```
set_check_library_options -mcmm
check_library -logic_library_name {a.db b.db}
```

When you use the `-mcmm` option with the `set_check_library_options` command, the `check_library` command performs multicorner-multimode specific checks, such as determining operating condition or power-down inconsistencies. When inconsistencies are detected, the tool generates a report that lists the inconsistencies. In addition, the tool issues the following summary information message:

```
Information: Logic library consistency check FAILED for MCMM.
(LIBCHK-360)
```

When you get a LIBCHK-360 message, check the report to identify the cause of the problem and fix the library inconsistencies. The LIBCHK-360 man page describes possible causes for the library inconsistencies.

Setting the `dont_use` Attribute on Library Cells in the Multicorner-Multimode Flow

When you set the `dont_use` attribute on a library cell, the multicorner-multimode feature requires that all characterizations of this cell have the `dont_use` attribute. Otherwise, the tool might consider the libraries as inconsistent. You can use the wildcard character to set the `dont_use` attribute as follows:

```
set_dont_use */AN2
```

When the library cells that have a `dont_use` attribute have a pin order that does not match exactly in the libraries of various corners, the tool continues with the flow without any error or warning messages. If you remove the `dont_use` attribute on these cells, the tool issues the MV-087 error message.

Note that you do not have to issue the command multiple times to set the `dont_use` attribute on all characterizations of a library cell.

Using Unique PVT Names to Prevent Linking Problems

To prevent linking problems, make sure your PVT operating conditions have unique names. If the maximum libraries associated with each scenario do not have distinct PVT values, your cell instances might be incorrectly linked, resulting in incorrect timing values. This happens because the nominal PVT values that are used to group the link libraries into sets group the maximum libraries of different corners into one set. Consequently, the cell instances are linked to the first cell with a matching type in that set, for example, the first AND2_4 cell, even though the `-library` option is specified for each of the scenario-specific `set_operating_conditions` commands. That is, the `-library` option locates the operating condition and its PVT but not the library to link.

The following paragraphs describe a linking problem due to non-unique PVT names. For the conditions given in [Table 20-4](#), [Table 20-5](#), and [Example 20-11](#), the tool groups the Ftyp.db and TypHV.db libraries into a set with Ftyp.db as the first library in the set. Therefore, the cell instances in scenario s2 are not linked to the library cells in TypHV.db, as intended. Instead, they are incorrectly linked to the library cells in the Ftyp.db library, assuming that all the libraries include the library cells required to link the design.

[Table 20-4](#) shows the libraries in the link library, listed *in order*; their nominal PVT; and the operating condition that is specified in each library.

Table 20-4 Link Libraries With PVT and Operating Conditions

Link library (in order)	Nominal PVT	Operating conditions in library (PVT)
Ftyp.db	1/1.30/100	WORST (1/1.30/100)
Typ.db	1/0.85/100	WORST (1/0.85/100)
TypHV.db	1/1.30/100	WORST (1/1.30/100)
Holdtyp.db	1/0.85/100	BEST (1/0.85/100)

[Table 20-5](#) shows the operating condition specifications for each of the scenarios; [Example 20-11](#) shows the corresponding scenario creation script.

Table 20-5 Scenarios and Their Operating Conditions

Scenarios				
	s1	s2	s3	s4
Max Opcond (Library)	WORST (Typ.db)	WORST (TypHV.db)	WORST (Ftyp.db)	WORST (Typ.db)
Min Opcond (Library)	None	None	None	BEST (HoldTyp.db)

Example 20-11 Linking Problem Due to Non-Unique PVT Name

```
create_scenario s1
set_operating_conditions WORST -library Typ.db:Typ
create_scenario s2
set_operating_conditions WORST -library TypHV.db:TypHV
create_scenario s3
set_operating_conditions WORST -library Ftyp.db:Ftyp
create_scenario s4
set_operating_conditions \
    -max WORST -max_library Typ.db:Typ \
    -min BEST -min_library HoldTyp.db:HoldTyp
```

Ambiguous Libraries Warning

The tool issues a warning if your design uses any libraries containing cells with the same name and same nominal PVT. The warning states that the libraries are ambiguous and identifies which libraries are being used and which are being ignored.

Unsupported k-factors

Multicorner-multimode design libraries do not support k-factor scaling. Therefore, the operating conditions that you specify for each scenario must match the nominal operating conditions of one of the libraries in the link library list.

Automatic Detection of Driving Cell Library

In multicorner-multimode flow, the operating condition setting is different for different scenarios. To build the timing arc for the driving cell, different libraries are used for different scenarios. You can specify the library using the `-library` option of the `set_driving_cell` command. But specifying the library is optional because the tool can automatically detect the driving cell library.

When you specify the library using the `-library` option of the `set_driving_cell` command, the tool searches for the specified library in the link library set. If the specified library exists, it is used. If the specified library does not exist in the link library, the tool issues the UID-993 error message as follows:

```
Error: Cannot find the specified driving cell in memory. (UID-993)
```

When you do not use the `-library` option of the `set_driving_cell` command, the tool searches all the libraries for the matching operating conditions. The first library in the link library set that matches the operating condition is used. If no library in the link library set matches the operating condition, the first library in the link library set that contains the matching library cell is used. If no library in the link library set contains the matching library cell, the tool issues the UID-993 error message.

Defining Minimum Libraries

Minimum libraries are usually defined with the `set_operating_conditions` command. You can use the `set_min_library` command, but it is not scenario-specific. If you use `set_min_library` to define a minimum library for a scenario, the tool uses that library as the minimum library for all scenarios, even if you do not define that library in all your scenarios. If you want to define different minimum libraries for each scenario, use the `set_operating_conditions` command.

Table 20-6 Unsupported Multiple Minimum Library Configuration

Scenarios	
s1	s2
Max library	Slow.db
Min library	Fast_0yr.db
	Fast_10yr.db

For example, you could not relate two different minimum libraries – say, `Fast_0yr.db` and `Fast_10yr.db` – with the maximum library, `Slow.db`, in two separate scenarios. The first minimum library you specify would apply to both scenarios. [Table 20-6](#) shows the *unsupported* configuration.

Note, however, that a minimum library can be associated with multiple maximum libraries. As shown in [Table 20-7](#), the minimum library Fast_0yr.db is paired with both the maximum library Slow.db of scenario 1 and the maximum library SlowHV.db of scenario 2.

Table 20-7 Supported Min-Max Library Configuration

Scenarios		
	s1	s2
Max library	Slow.db	SlowHV.db
Min library	Fast_0yr.db	Fast_0yr.db

Scenario Management Commands

Use the following commands to create and manage scenarios:

- `create_scenario`
- `current_scenario`
- `all_scenarios`
- `all_active_scenarios`
- `set_active_scenarios`
- `set_scenario_options`
- `set_preferred_scenario`
- `check_scenarios`
- `remove_scenario`
- `report_scenarios`
- `report_scenario_options`

The following subsections describe how you use these commands to manage scenarios:

- [Creating Scenarios](#)
- [Defining Active Scenarios](#)
- [Scenario Reduction](#)
- [Specifying Scenario Options](#)
- [Removing Scenarios](#)

Creating Scenarios

You use the `create_scenario` command to create a new scenario. When the first scenario is created, all previous scenario-specific constraints are removed from the design and the following warning is issued:

```
dc_shell-topo> create_scenario s1
Warning: Any existing scenario-specific constraints
         are discarded. (MV-020)
Current scenario is: s1
```

Use the `current_scenario` command to specify the name of the current scenario. Without arguments, the command returns the name of the current scenario. Note that the `current_scenario` command merely specifies the focus scenario and that when you define more than one scenario, the tool performs concurrent analysis and optimization across all scenarios, independent of the current scenario setting.

Use the `set_tlu_plus` command to set the TLUPlus files in the scenario specified by the `current_scenario` command. Note that each scenario must have a set of TLUPlus files specified, or the following error is reported:

```
Error: tlu_plus files are not set in this scenario <name>.
RC values will be 0.
```

Defining Active Scenarios

During concurrent analysis and optimization, you can significantly reduce memory usage and runtime by limiting the number of active scenarios to those that are “essential” or “dominant.” An essential or dominant scenario has the worst slack among all the scenarios for at least one of its constrained objects. (Constrained objects can include delay constraints associated with a pin, the design rule constraints for a net, leakage power, and so on.) Any scenario for which one of its constrained objects is the worst slack value is a dominant scenario.

You use the `set_active_scenarios` command to define active scenarios. Other commands related to active scenarios are `all_scenarios` and `all_active_scenarios`.

Scenario Reduction

Topographical mode automatically performs scenario reduction on the current set of active scenarios to reduce memory and runtime. In general, restricting concurrent analysis and optimization to a subset of dominant scenarios does not lead to a significant difference in QoR. The tool analyzes all the active scenarios and automatically determines a set of dominant scenarios, based on the number of violating endpoints, and optimizes them for timing, power, and design rule. In the current version, topographical mode does not support the `get_dominant_scenarios` command, which IC Compiler supports. However, you can choose a preferred scenario by using the `set_preferred_scenario` command. When you

set the preferred scenario, the tool treats it as the most constraining scenario. It still performs scenario reduction to determine the dominant scenarios but treats the user-specified preferred scenario as the most constraining one.

Specifying Scenario Options

To define specific constraint options, such as leakage power, that you want optimized in a scenario, use the `set_scenario_options` command. You can apply the constraint option to more than one scenario at a time by using the `-scenarios` option. The options can be specified on both active and inactive scenarios. If you do not specify any scenario, the options are applied only to the current scenario.

To enable or disable the leakage power optimization, use the `-leakage_power` option. The default for the `-leakage_power` option is `false`. Set the `-leakage_power` option with the `set_scenario_options` command to `true` to enable leakage power optimization on specific scenarios in a multicorner-multimode design. The following command shows how to enable leakage power optimization on specific scenarios only.

```
dc_shell-topo> set_scenario_options -leakage_power true \
    -scenarios scenario_list
```

To enable or disable the dynamic power optimization, use the `-dynamic_power` option. The default for the `-dynamic_power` option is `false`.

To enable or disable the setup or maximum delay optimization for specified scenarios, use the `-setup` option. The default for the `-setup` option is `true`, so maximum delay optimization is enabled by default. The following example shows how to disable maximum delay optimization on specific scenarios.

```
dc_shell-topo> set_scenario_options -setup false -scenarios
    scenario_list
```

To enable or disable the hold or minimum delay optimization for the specified scenarios, use the `-hold` option. The default for the `-hold` option is `true`. When you set the `-hold` option to `false`, the tool ignores the hold or the minimum delay violations in the specified scenarios.

```
dc_shell-topo> set_scenario_options -reset_all true \
    -scenarios [all_scenarios]
```

To report the scenario options, use the `report_scenario_options` command.

Removing Scenarios

To remove the specified scenarios, use the `remove_scenario` command. All scenario-specific constraints defined in the removed scenario are deleted. For example,

```
dc_shell-topo> remove_scenario
s1 s2
```

```
dc_shell-topo> remove_scenario -all
Removed scenario 's2'
Removed scenario 's1'
dc_shell-topo> all_scenarios
```

Reporting Commands for Multicorner-Multimode Designs

This section describes the commands that you can use for reporting multicorner-multimode designs in Design Compiler Graphical:

- [report_scenarios Command](#)
- [report_scenario_options Command](#)
- [Reporting Commands That Support the -scenario Option](#)
- [Commands That Report the Current Scenario](#)
- [Reporting Examples](#)

report_scenarios Command

The `report_scenarios` command reports the scenario setup information for multicorner-multimode designs. This command reports all the defined scenarios. The scenario specific information includes the logic library used, the operating condition, and TLUPlus files.

The following example shows a report generated by the `report_scenarios` command:

```
*****
Report : scenarios
Design : DESIGN1
scenario(s) : SCN1
...
*****
All scenarios (Total=4): SCN1 SCN2 SCN3 SCN4
All Active scenarios (Total=1): SCN1
Current scenario      : SCN1

Scenario #0: SCN1 is active.
Scenario options:
Has timing derate: No
Library(s) Used:
  logic library name (File: library.db)

Operating condition(s) Used:
  Analysis Type      : bc_wc
  Max Operating Condition: library:WCCOM
  Max Process        : 1.00
```

```

Max Voltage      : 1.08
Max Temperature: 125.00
Min Operating Condition: library:BCCOM
Min Process     : 1.00
Min Voltage     : 1.32
Min Temperature: 0.00

Tlu Plus Files Used:
  Max TLU+ file: tlu_plus_file.tf
  Tech2ITF mapping file: tf2itf.map

```

report_scenario_options Command

Use the `report_scenario_options` command to report the scenario options set by the `set_scenario_options` command. You can specify a list of scenarios to be reported by using the `-scenarios` option. By default, this command reports scenario options for the current, active scenario.

To illustrate the report generated by the `report_scenario_options` command, consider [Example 20-12](#) where the `set_scenario_options -leakage_power false` command is executed. This command sets the `-leakage_power` option to `false`. With the `-leakage_power` option set to `false`, the tool will not optimize the current, active scenario for leakage power. This status is shown by the `report_scenario_options` report in [Example 20-12](#).

Example 20-12

```

dc_shell-topo> set_scenario_options -leakage_power false
dc_shell-topo> report_scenario_options
*****
Report : scenario options
Design : TEST03
...
*****
Scenario: MODE1 is active.

setup      : true
hold       : true
leakage_power : false
dynamic_power : true

```

Reporting Commands That Support the `-scenario` Option

Some reporting commands support the `-scenario` option to report scenario-specific information. You can specify a list of scenarios to the `-scenario` option, and the tool reports scenario details for the specified scenarios.

The following reporting commands support the `-scenario` option:

- `report_timing`
- `report_timing_derate`
- `report_power`
- `report_clock`
- `report_path_group`
- `report_extraction_options`
- `report_tlu_plus_files`
- `report_constraint`

Commands That Report the Current Scenario

The following reporting commands report scenario-specific details for the current scenario. The header section of the report contains the name of the current scenario. No additional options are required to report the scenario-specific details of the current scenario.

- `report_net`
- `report_annotated_check`
- `report_annotated_transition`
- `report_annotated_delay`
- `report_attribute`
- `report_case_analysis`
- `report_ideal_network`
- `report_internal_loads`
- `report_clock_gating_check`
- `report_clock_tree`
- `report_delay_calculation`
- `report_delay_estimate_options`
- `report_transitive_fanout`
- `report_disable_timing`
- `report_latency_adjustment_options`

- report_net
- report_power_calculation
- report_noise
- report_signal_em
- report_timing_derate
- report_timing_requirements
- report_transitive_fanin
- report_crpr
- report_clock_timing

Reporting Examples

This section contains sample reports for some of the multicorner-multimode reporting commands.

report_qor Command

The `report_qor` command reports by default the QoR details for all the scenarios in the design. The following example shows a report generated by the `report_qor` command:

```
*****
Report : qor
Design : DESIGN1
*****
Scenario 's1'
Timing Path Group 'reg2reg'
-----
Levels of Logic:          33.00
Critical Path Length:    694.62
Critical Path Slack:     -144.52
Critical Path Clk Period: 650.00
Total Negative Slack:   -4533.01
No. of Violating Paths:  136.00
-----
Scenario 's2'
Timing Path Group 'reg2reg'
-----
Levels of Logic:          33.00
Critical Path Length:    393.61
Critical Path Slack:     61.18
Critical Path Clk Period: 500.00
Total Negative Slack:   0.00
No. of Violating Paths:  0.00
-----
```

report_timing -scenario

This command reports timing results for the active scenarios in the design. You can specify a list of scenarios with the `-scenario` option. When the `-scenario` option is not specified only the current scenario is reported.

```
*****
Report : timing
    -path full
    -delay max
    -max_paths 1
Design : DESIGN1
...
*****
* Some/all delay information is back-annotated.

# A fanout number of 1000 was used for high fanout net computations.

Startpoint: TEST_BUFBEn
            (input port clocked by clk)
Endpoint: TEST1/TEST2_SYN/latch_3
            (non-sequential rising-edge timing check clocked by clk)
Scenario: s1
Path Group: clk
Path Type: max
Point           Incr      Path      Lib:OC
-----
clock clk (rise edge)          0.00      0.00
clock network delay (propagated) 0.00      0.00
input external delay           450.00   450.00 f
TEST_BUFBEn (in)              0.00      450.00 f stdcell_typ:WORST
TEST_BUFBEn_BUFB1/Z (inx4)     9.75      459.75 r stdcell_typ:WORST
U468/Z (inx10)                10.21     469.96 f stdcell_typ:WORST
TEST_BUFBEn_BUFB/Z (inx11)     8.74      478.70 r stdcell_typ:WORST
U293/Z (inx11)                9.30      488.00 f stdcell_typ:WORST
TEST1/TEST2_SYN/U74963/Z (nr2x4) 12.78     500.78 r stdcell_typ:WORST
U31662/Z (inx4)               10.58     511.37 f stdcell_typ:WORST
TEST1/TEST2_SYN/U75093/Z (aoi21x6) 18.98     530.34 r stdcell_typ:WORST
U42969/Z (nd2x6)              14.16     544.51 f stdcell_typ:WORST
TEST1/TEST2_SYN/U53046/Z (inx8) 13.35     557.86 r stdcell_typ:WORST
U2765/Z (inx8)                11.48     569.33 f stdcell_typ:WORST
U32442/Z (inx6)               7.61      576.94 r stdcell_typ:WORST
U33615/Z (nd2x3)              18.14     595.09 f stdcell_typ:WORST
U32269/Z (nd2x6)              8.74      603.82 r stdcell_typ:WORST
TEST1/TEST2_SYN/clk_gate/EN (cklan2x1) 0.00      603.82 r stdcell_typ:WORST
data arrival time              603.82
```

```

clock clk (rise edge)           650.00    650.00
clock network delay (propagated) 0.00     650.00
TEST1/TEST2_SYN/clk_gate/CLK (cklan2x1)
                                     0.00     650.00 r
library setup time             -56.25    593.75
data required time              593.75
-----
data required time                593.75
data arrival time                 -603.82
-----
slack (VIOLATED)                  -10.07

```

report_constraint

This command reports constraints for all active scenarios. Each scenario is reported separately. When used with the `-scenario` option, it reports constraints for a specified list of scenarios.

```
*****
Report : constraint
Design : DESIGN1
Scenarios: 0, 1
...
*****

```

Group (max_delay/setup)	Cost	Weight	Weighted Cost		Scenario
			Cost	Scenario	
CLK	10.07	1.00	10.07	s1	
in2out	372.89	1.00	372.89	s1	
in2reg	199.73	1.00	199.73	s1	
reg2out	467.99	1.00	467.99	s1	
reg2reg	171.16	1.00	171.16	s1	
default	0.00	1.00	0.00	s1	
CLK	90.60	1.00	90.60	s2	
in2out	474.97	1.00	474.97	s2	
in2reg	166.88	1.00	166.88	s2	
reg2out	326.46	1.00	326.46	s2	
reg2reg	0.00	1.00	0.00	s2	
default	0.00	1.00	0.00	s2	
max_delay/setup			4404.52		
...					
Constraint			Multi-Scenario Cost		
multiport_net			0.00 (MET)		
min_capacitance			0.00 (MET)		
max_transition			45.28 (VIOLATED)		
max_fanout			150.00 (VIOLATED)		
max_capacitance			0.00 (MET)		
max_delay/setup			4404.52 (VIOLATED)		
critical_range			4404.52 (VIOLATED)		
min_delay/hold			0.00 (MET)		
max_area			714233.56 (VIOLATED)		

report_tlu_plus_files

This command reports the TLUPlus files associations; it shows each minimum and maximum TLUPlus and layer map file per scenario:

```
dc_shell-topo> current_scenario s1
Current scenario is: s1

dc_shell-topo> report_tlu_plus_files
Max TLU+ file: /snps/testcase/s1max.tluplus
Min TLU+ file: /snps/testcase/s1min.tluplus
Tech2ITF mapping file: /snps/testcase/tluplus_map.txt
```

report_power

The `report_power` command supports the `-scenario` option. Without the `-scenario` option, only the current scenario is reported. To report power information for all scenarios, use the `report_power -scenarios [all_scenarios]` command.

Note:

In the multicorner-multimode flow, the `report_power` command does not perform clock tree estimation. The command reports only the netlist power in this flow.

The following example shows the report generated by the `report_power -scenario` command.

```
*****
Report : power
Design : Design_1
Scenario(s): s1
...
*****

Library(s) Used: slow (File: slow.db)

Global Operating Voltage = 1.08
Power-specific unit information :
  Voltage Units = 1V
  Capacitance Units = 1.000000pf
  Time Units = 1ns
  Dynamic Power Units = 1mW      (derived from V,C,T units)
  Leakage Power Units = Unitless

Warning: Could not find correlated power. (PWR-725)

Power Breakdown
-----
          Cell      Driven Net   Tot Dynamic     Cell
          Internal   Switching    Power (mW)    Leakage
Cell        Power (mW)    Power (mW)   (% Cell/Tot)  Power(nW)
-----
Netlist Power           4.8709     1.2889  6.160e+00 (79%) 1.351e+05
Estimated Clock Tree Power  N/A         N/A       (N/A)          N/A
-----
```

Supported SDC Commands for Multicorner-Multimode Designs

[Table 20-8](#) lists the SDC commands supported in the multicorner-multimode flow.

Table 20-8 Supported SDC Commands

Commands	
all_clocks	set_fanout_load
create_clock	set_input_delay
create_generated_clock	set_input_transition
get_clocks	set_latency_adjustment_options
group_path	set_load
set_annotated_delay	set_max_capacitance
set_capacitance	set_max_delay
set_case_analysis	set_dynamic_optimization
set_clock_gating_check	set_max_time_borrow
set_clock_groups	set_max_transition
set_clock_latency	set_min_delay
set_clock_transition	set_multicycle_path
set_clock_uncertainty	set_output_delay
set_data_check	set_propagated_clock
set_disable_timing	set_resistance
set_drive	set_timing_derate
set_false_path	

Multicorner-Multimode Script Example

[Example 20-13](#) shows a basic sample script for the multicorner-multimode flow.

Example 20-13 Basic Script to Run a Multicorner-Multimode Flow

```
#.....path settings.....  
set search_path ". $DESIGN_ROOT $lib_path/dbs \  
    $lib_path/mwlibs/macros/LM"  
set target_library "stdcell.setup.ftyp.db \  
    stdcell.setup.typ.db stdcell.setup.typhv.db"  
set link_library [concat * $target_library \  
    setup.ftyp.130v.100c.db setup.typhv.130v.100c.db \  
    setup.typ.130v.100c.db]  
set_min_library stdcell.setup.typ.db -min_version stdcell.hold.typ.db  
  
#.....MW setup.....  
#.....load design.....  
  
create_scenario s1  
set_operating_conditions WORST -library stdcell.setup.typ.db:stdcell_typ  
set_tlu_plus_files -max_tluplus design.tlup -tech2itf_map layermap.txt  
read_sdc s1.sdc  
set_scenario_options -scenarios s1-setup false -hold false \  
-leakage_power true  
  
create_scenario s2  
set_operating_conditions BEST -library stdcell.setup.ftyp.db:stdcell_ftyp  
set_tlu_plus_files -max_tluplus design.tlup -tech2itf_map layermap.txt  
read_sdc s2.sdc  
  
create_scenario s3  
set_operating_conditions NOM -library stdcell.setup.ftyp.db:stdcell_ftyp  
set_tlu_plus_files -max_tluplus design.tlup -tech2itf_map layermap.txt  
read_sdc s3.sdc  
  
set_active_scenarios {s1 s2}  
report_scenarios  
compile_ultra -scan -gate_clock  
report_qor  
report_constraint  
report_timing -scenario [all_scenarios]  
. . .  
insert_dft  
. . .  
compile_ultra -incremental -scan
```

Using Block Abstractions in Multicorner-Multimode Designs

A block abstraction is a structural model of a circuit that is modeled as a smaller circuit representing the interface logic of the block. The model contains cells whose timing is affected by or affects the external environment of a block. Block abstractions enhance capacity and reduce runtime for the optimization of the top-level design. For block abstraction details, see [Using Hierarchical Models](#).

Block abstractions are compatible with multicorner-multimode scenarios. You can apply multicorner-multimode constraints to a block abstraction and use the block abstraction in a top-level design.

The following requirements apply to the use of block abstractions with multicorner-multimode scenarios:

- For each top-level multicorner-multimode scenario, an identically named scenario must exist in each of the block abstraction blocks used in the design.

If there is a mismatch, use the `select_block_scenario` command to map the scenarios in the block abstraction to the top-level design. Mapping scenarios enables Design Compiler Graphical to support name mismatches between the top-level design and the block abstraction.

To reset user-specified multicorner-multimode scenario mapping, use the `-reset` option.

- A block abstraction can have additional scenarios that are not used at the top level.
- By default, in a top-level design without multicorner-multimode scenarios, only block abstractions without multicorner-multimode scenarios can be used. Using the `select_block_scenario` command to specify scenario mapping allows you to use multicorner-multimode block abstractions even if the top-level design does not have multicorner-multimode scenarios.
- For each TLUPlus file that is used, the block abstraction stores the extraction data and the specified operating condition. In the top-level design, you cannot use additional TLUPlus files or define additional temperature corners for the existing TLUPlus files.
- A top-level design with multicorner-multimode scenarios can be used with block abstractions that do not have any multicorner-multimode scenario definitions. No log message is issued in this case, and the same block data is used in all the top-level scenarios.

Methodology for Using Block Abstractions With Scenarios at the Top Level

Follow these steps to use a block abstraction at the top level when it has scenario information:

1. Set the current design to the top-level design.

2. Remove all scenarios:

```
remove_scenario -all
```

3. Define scenarios for the top-level design.

The scenarios defined in the top-level design must match the scenario definitions in the block abstraction blocks of the design. In the top-level design, all the scenarios must be defined before the next step.

4. Perform optimization:

```
compile_ultra
```

At the beginning of compilation, the `compile_ultra` command performs the following checks to ensure that there are no scenario mismatches between the top-level design and the block abstractions. The compilation is terminated when any of the following mismatches are encountered:

- The number of scenarios in the top-level design does not match the number of scenarios in the block abstraction blocks.

If the tool detects that the top-level design has more scenarios than block abstraction blocks, a TL-70 error message is issued, and compilation is terminated:

```
Error: Scenario S6 is not available in block reference Block1.  
(TL-70)
```

- The scenario information in the top-level design is not consistent with the scenario information in the block abstraction blocks.

If scenarios are not defined in the top-level design and the block abstraction blocks have scenario definitions, a TL-151 error message is issued and compilation is terminated:

```
Error: No scenario data loaded from block reference 'BlockInit'.  
(TL-151)
```

You can also use the `check_scenarios` command to check consistency between scenarios.

21

Using Design Compiler NXT

The Design Compiler NXT tool extends topographical technology by providing

- Better QoR through advanced optimizations
- Enhanced physical guidance to IC Compiler II, with improved RC and timing correlation
- Faster runtime, with improved multithreading synthesis for better scalability
- Plug-and-play user interface and script compatibility with Design Compiler Graphical

For features supported in both Design Compiler NXT and Design Compiler Graphical, see [Using Design Compiler Graphical](#).

The following features are supported only in the Design Compiler NXT tool:

- [Performing Buffering-Aware Placement With the IC Compiler II Link](#)
- [Performing Automatic Timing Control With the IC Compiler II Link](#)
- [Performing Congestion-Driven Restructuring With the IC Compiler II Link](#)
- [Performing RC Extraction With the RC Model](#)
- [Applying Required Settings for High Performance Cores](#)

Performing Buffering-Aware Placement With the IC Compiler II Link

Buffering-aware placement considers high-fanout nets and long nets during the initial placement to improve QoR. To enable the buffering-aware timing-driven placement when you use the `compile_ultra -spg` command, use the `placer_buffering_aware` variable. The default is `false`.

To use the buffering-aware placement feature, you must use the `set_icc2_options` command as follows:

```
dcnxt_shell-topo> set_app_var placer_buffering_aware true  
dcnxt_shell-topo> set_icc2_options -ref_libs "lib1.ndm lib2.ndm" \  
-technology tech.tf
```

When you set the `placer_buffering_aware` variable to `true` during the initial placement with the `compile_ultra -spg` command ,

- Design Compiler NXT enables timing-driven placement with an approximate timing model that estimates the effects of buffering high-fanout nets and long nets. The netlist does not change.
- Before invoking the IC Compiler II tool, Design Compiler NXT issues the following message:

```
Information: Buffering aware placement is enabled. (SPG-129)
```

- After invoking the IC Compiler II tool, the following message is issued in the IC Compiler II log file:

```
Create Placement Options:  
Buffering Aware Timing Driven: true
```

Performing Automatic Timing Control With the IC Compiler II Link

Automatic timing control improves total negative slack (TNS) during direct timing-driven placement. To enable automatic timing control when you use the `compile_ultra -spg` and `compile_ultra -incremental -spg` commands, use the `placer_auto_timing_control` variable. The default is `false`.

Note:

If you set both the `placer_buffering_aware` and `placer_auto_timing_control` variables to `true`, the tool performs both buffering-aware placement and automatic timing control during the initial placement. However, the tool still performs automatic timing control during the direct timing-driven placement.

To use the automatic timing control feature, you must use the `set_icc2_options` command as follows:

```
dcnxt_shell-topo> set_app_var placer_auto_timing_control true
dcnxt_shell-topo> set_icc2_options -ref_libs "lib1.ndm lib2.ndm" \
-technology tech.tf
```

When you set the `placer_auto_timing_control` variable to `true`,

- Design Compiler NXT enables the IC Compiler II direct timing-driven placement along with the Zroute-based congestion-driven placement when you use the `compile_ultra -spg` command.
- Design Compiler NXT enables only the IC Compiler II direct timing-driven placement when you use the `compile_ultra -incremental -spg` command. However, to enable the IC Compiler II Zroute-based congestion-driven placement during the incremental compile, you must set the `spg_congestion_placement_in_incremental_compile` variable to `true`.
- The direct timing-driven placement tries to focus the timing optimization on the most critical timing paths to balance between reducing the worst negative slack (WNS) and reducing the total negative slack (TNS)
- Before invoking the IC Compiler II tool, Design Compiler NXT issues the following message:

Information: Auto timing control is enabled in the placer. (SPG-129)

- After invoking the IC Compiler II tool, the following message is issued in the IC Compiler II log file:

```
Printing options for 'place.coarse.*' (non-default only)
place.coarse.auto_timing_control: true
```

Performing Congestion-Driven Restructuring With the IC Compiler II Link

Congestion in the design can be caused by commutative and associative logic trees, such as OR trees, AND trees, and XOR trees. You can improve congestion by rewiring input signals of the logic trees with the congestion-driven restructuring feature.

To enable congestion-driven restructuring when you use the `compile_ultra -spg` command, use the `placer_cong_restruct` variable. The default is `false`. To use the congestion-driven restructuring feature, you must use the `set_icc2_options` command as follows:

```
dcnxt_shell-topo> set_app_var placer_cong_restruct true
dcnxt_shell-topo> set_icc2_options -ref_libs "lib1.ndm lib2.ndm" \
-technology tech.tf
```

When you set the `placer_cong_restruct` variable to `true`,

- Design Compiler NXT performs restructuring of the net during the initial placement with the `compile_ultra -spg` command
- Before invoking the IC Compiler II tool, Design Compiler NXT issues the following message:

```
Information: Congestion driven restructuring placement is enabled.  
(SPG-131)
```

- After invoking the IC Compiler II tool, the following message is issued in the IC Compiler II log file:

```
Printing options for 'place.coarse.*' (non-default only)  
place.coarse.cong_restruct: on
```

Performing RC Extraction With the RC Model

You can perform RC extraction for improving the RC correlation with the pre `place_opt` results from the IC Compiler II tool. To perform the RC extraction, use the `spg_icc2_rc_correlation` variable to correlate the RC value of Design Compiler NXT with the IC Compiler II tool, which helps in solving timing miscorrelation issues.

This feature is available only

- When you use the `compile_ultra -spg` command
- In the advanced-technology nodes (16 nm and lower-process nodes) when metal layers have a significant difference in resistance value

When you set this variable to `true` (the default is `false`),

- The `extract_rc` command performs RC extraction by using the advanced RC model
- The tool issues the “Turning on advanced RC model” message to indicate that this feature is enabled for optimization and extraction when you use the `compile_ultra`, `compile_ultra - incremental`, and `optimize_netlist` commands and explicitly run the `extract_rc` command

Applying Required Settings for High Performance Cores

You can apply tool settings required for achieving the best QoR for high performance cores by using the `set_technology` and `set_hpc_options` commands. The applied settings affect synthesis, placement, optimization, and timing analysis. The 16, 12, and 7 nanometer technology nodes are supported with the `set_technology` command.

As shown in the following example, you must use

- The `set_technology` command to apply technology-node settings before using the `set_hpc_options` command
- The `set_hpc_options` command before using the `compile_ultra` and `compile_ultra -incremental` commands to set the stage-specific settings

```
# read the setup, design, and constraints
set_technology -node 7
set_hpc_options -core A55 -stage compile
compile_ultra -spg
...
insert_dft
...
set_hpc_options -core A55 -stage compile_inc
compile_ultra -spg -incremental
```

Note:

If you apply any tool settings after using the `set_hpc_options` command, the last setting overrides the previous settings.

To report the core and technology-node settings applied by the tool, use the `-report_only` option of the `set_hpc_options` command.

22

Using Hierarchical Models

Hierarchical models are used in Design Compiler to reduce the number of design objects and the memory requirements when the tool performs top-level optimization on large designs. The term *hierarchical models* refers to block abstractions and physical hierarchies.

To learn about generating and using hierarchical models in Design Compiler, see

- [Overview of Hierarchical Models](#)
- [Block Abstraction Hierarchical Flow](#)

For information about using hierarchical models in a bottom-up compile flow, see [Performing a Bottom-up Hierarchical Compile](#).

Overview of Hierarchical Models

Hierarchical models are used in Design Compiler to reduce the number of design objects and the memory requirements when the tool performs top-level optimization on large designs. For interface logic models (ILMs), the gate-level netlist for a block is modeled by a partial gate-level netlist that contains only the required interface logic of the block and possibly the logic that you manually associate with the interface logic. All other logic is removed. However, Design Compiler does not support ILMs. Instead, Design Compiler uses block abstractions.

Block abstractions are an extension to interface logic models. Unlike an ILM where the internal logic is removed and saved in a separate .ddc file, when you use block abstractions, the full design and the block abstraction information are both saved in the same .ddc file. The interface logic is marked, and only the interface logic is loaded when using the block abstraction.

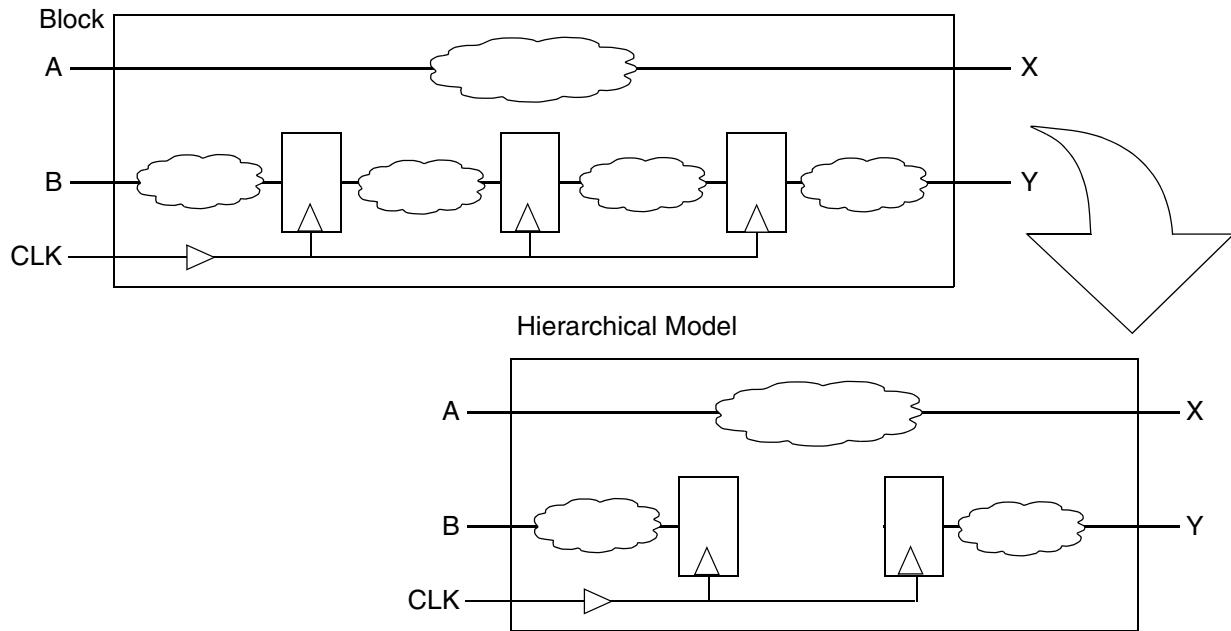
Design Compiler can create block abstractions, and it can use block abstractions created either in Design Compiler or in IC Compiler. However, IC Compiler can only use block abstractions created in IC Compiler. For optimal correlation and alignment between the tools, use IC Compiler block abstractions to model the physical blocks in both tools.

[Figure 22-1](#) shows a block and its hierarchical model, where the logic is preserved between

- The input port and the first register of each timing path
- The last register of each timing path and the output port

Logic associated with pure combinational input-port-to-output-port timing paths (A to X) is also preserved. Clock connections to the preserved registers are kept as well. The register-to-register logic is discarded.

Figure 22-1 A Block and Its Hierarchical Model



Design Compiler supports the use of hierarchical models in a variety of flows, including

- Multicorner-multimode

Design Compiler automatically detects the presence of multiple corners and multiple modes and retains the interface logic involved in the interface timing paths of each scenario.

- Designs with multiple levels of physical hierarchy

You can use nested block abstractions to model designs with multiple levels of physical hierarchy. For more information about nested block abstractions, see [Creating Block Abstractions for Nested Blocks](#).

See Also

- [Information Used in Hierarchical Models](#)
- [Using Hierarchical Models in a Multicorner-Multimode Flow](#)
- [Viewing Hierarchical Models in the GUI](#)

Information Used in Hierarchical Models

By default, hierarchical models include the following netlist objects:

- Leaf cells and macro cells in the four critical timing paths that lead from input ports to output ports (combinational input-to-output paths), input ports to edge-triggered registers, and edge-triggered registers to output ports
- Abstracted clock trees that include clock paths to interface registers, minimum and maximum clock paths that could also be connected to noninterface registers, and the clock tree synthesis exceptions that are a part of these clock paths

Design Compiler treats generated clocks like clock ports. Design Compiler retains interface registers that are driven by a generated clock. However, internal registers driven by generated clocks are not retained. When the design contains generated clocks, the logic along the timing path between the master clock and the generated clock is also retained.

- Clock-gating circuitry—if it is driven by external ports
- Logic along the timing path between a master clock and any generated clocks derived from it
- Side-load cells for all nets

Side-load cells are cells that can affect the timing of an interface path but are not directly part of the interface logic.

Note:

By default, latches are treated as combinational logic.

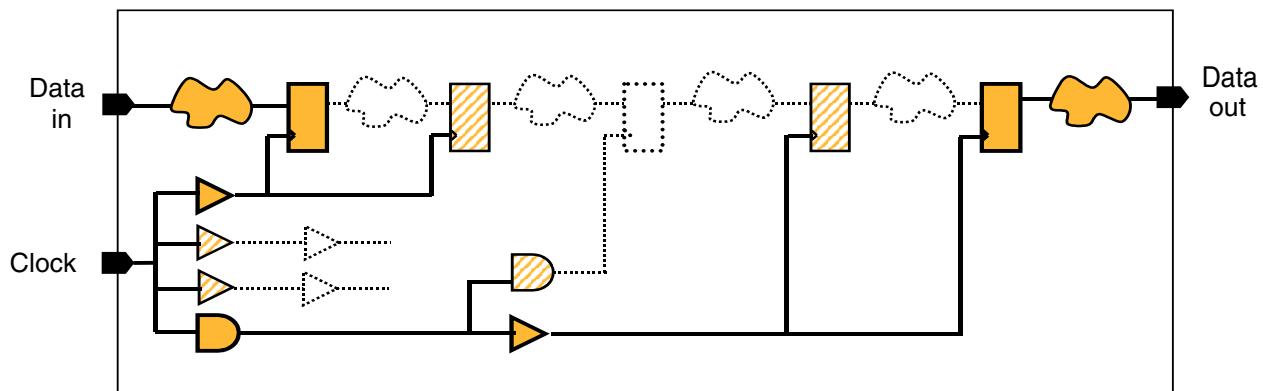
Apart from the netlist objects, the following information is also stored in a hierarchical model:

- Cell or pin location information
- Parasitic information
- Timing constraints
- Pin-to-pin delay for internal nets
- Power consumption

A block abstraction can contain register-to-register paths that are entirely within the block. These paths are the result of combinational logic in the block abstraction that is shared by input-to-register and register-to-output paths. Paths entirely within blocks are optimized during block-level optimization, and ignoring them during top-level optimization enables transparent interface optimization to focus on other paths. For information about ignoring these register-to-register paths, see [Ignoring Timing Paths That Are Entirely Within Block Abstractions](#).

[Figure 22-2](#) shows the elements contained in a hierarchical model, emphasizing the clock tree. The darkly shaded elements are the elements on the clock path; the lightly shaded elements are the side-load cells. Unshaded elements are not included in the model.

Figure 22-2 Hierarchical Model



Using Hierarchical Models in a Multicorner-Multimode Flow

You can apply multicorner-multimode constraints to a hierarchical model and use those constraints in a top-level design.

The following requirements apply to using blocks that are modeled using a hierarchical model with multicorner-multimode scenarios:

- For each top-level multicorner-multimode scenario, an identically named scenario must exist in each of the blocks used in the design.

If there is a mismatch, use the `select_block_scenario` command to map the scenarios in the blocks to the top-level design. A block can have additional scenarios that are not used at the top level.

The syntax for using the `select_block_scenario` command is

```
select_block_scenario
  [-scenarios top_scenarios]
  [-block_references list_design_names]
  -block_scenario block_scenario_name | -reset
```

To reset user-specified multicorner-multimode scenario mapping, use the `-reset` option.

- You can use a top-level design with multicorner-multimode scenarios with blocks that do not have multicorner-multimode scenarios. This does not require the use of the `select_block_scenario` command.

- By default, in a top-level design without multicorner-multimode scenarios, only blocks without multicorner-multimode scenarios can be used.
To use multicorner-multimode blocks with non-multicorner-multimode top-level designs, specify the scenario mapping by using the `select_block_scenario` command.
- For each TLUPlus file, a block stores the extraction data at the specified temperature, which is an operating condition. For accurate results during parasitic extraction at the top level, the TLUPlus files and temperature corners at the top level should match the TLUPlus and temperature corners used during block-level implementation. However, you can use additional TLUPlus and temperatures at the top level with loss in accuracy.

Viewing Hierarchical Models in the GUI

In the Design Compiler GUI, a hierarchical model instance is displayed as a level of physical hierarchy: a rectangular or rectilinear shape with a fill pattern. The View Settings panel provides additional control over the block abstraction display.

- To display the hierarchical models, select the Cell > ILM/Block Abstraction visibility option.
- To display the hierarchical model pins, select the Pin visibility option.
- To expand the hierarchical model to show the leaf cells, pins, nets, and macros within, type or select a positive integer in the “Level” box.
- To reset the display to the default view, the unexpanded view, type or select 0 in the “Level” box.

For more information about using the GUI to view physical hierarchy blocks, see the “Examining Physical Hierarchy Blocks” topic in Design Vision Help.

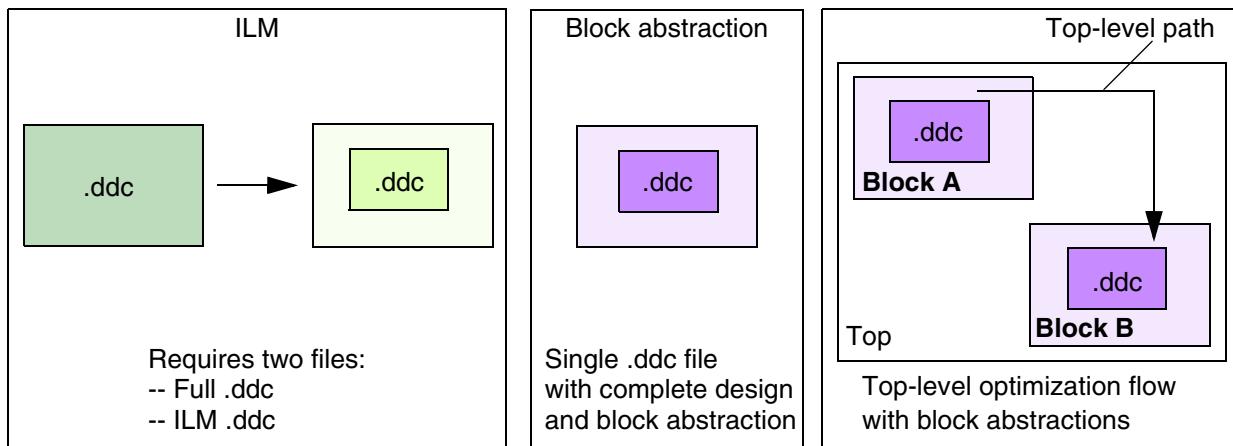
Block Abstraction Hierarchical Flow

A top-level flow using block abstractions allows optimization of only the top-level logic. Block abstractions are an extension to interface logic models (ILMs). Unlike an ILM where the internal logic is removed, all the logic is retained in a block abstraction, but only the interface logic is loaded. This supports faster top-level design closure by allowing optimization of block-level interface logic during top-level synthesis.

When you create and save block abstractions, they are stored in a single .ddc file in the same file that includes the complete design netlist. You do not need to maintain two .ddc files, one for interface logic and the other for the full netlist, as you needed to do for ILMs.

Figure 22-3 illustrates the difference between an ILM and a block abstraction. Design Compiler does not support ILMs. The figure is for comparison only.

Figure 22-3 ILM and Block Abstraction Comparison



By default, block abstractions are not modified or optimized. If you are using Design Compiler in topographical mode, you can optimize the block abstractions using top-level transparent interface optimization. Transparent interface optimization is the process that optimizes the interface logic within the blocks during top-level synthesis to achieve faster timing closure.

Transparent interface optimization provides the following benefits:

- Supports faster top-level design closure by performing concurrent optimization of top-level logic and block-level interfaces.
- Reduces iterations between top-level and block-level implementation.

Design Compiler can create block abstractions and use block abstractions created either in Design Compiler or in IC Compiler. However, IC Compiler can only use block abstractions created in IC Compiler. For information about creating a block abstraction in IC Compiler, see the *IC Compiler Implementation User Guide*.

Note:

You cannot combine block abstractions and ILMs in the same flow.

The following sections describe the block abstraction hierarchical flow:

- [Creating and Saving Block Abstractions](#)
- [Setting Top-Level Implementation Options](#)
- [Resetting Implementation Options](#)
- [Controlling the Extent of Logic Loaded for Block Abstractions](#)

- [Loading Block Abstractions](#)
- [Reporting Implementation Options](#)
- [Reporting Block Abstractions](#)
- [Querying Block Abstractions](#)
- [Checking Block Abstractions](#)
- [Ignoring Timing Paths That Are Entirely Within Block Abstractions](#)
- [Performing Top-Level Synthesis](#)
- [Saving Optimized Block Abstractions After Top-Level Synthesis](#)
- [Limitations](#)

Creating and Saving Block Abstractions

To generate a block abstraction for the current design, use the `create_block_abstraction` command at the end of synthesis. The `create_block_abstraction` command adds the block abstraction information to the design in memory.

To save the block abstraction, you must use the `write_file` command immediately after creating the block abstraction:

```
dc_shell-topo> create_block_abstraction
dc_shell-topo> write_file -hierarchy -format ddc -output ${DESIGN_NAME}.mapped.ddc
```

The .ddc file contains both the complete design and the Design Compiler block abstraction.

You can create block abstractions in either Design Compiler or Design Compiler in topographical mode. However, you must use topographical mode to optimize block abstractions, and only block abstractions created in topographical mode are optimized.

To specify the list of block references that you want to link to block abstractions and integrated with the top-level design, and to optimize the block abstractions using transparent interface optimization, use the `set_top_implementation_options` command as described in [Setting Top-Level Implementation Options](#).

Information Used in Block Abstractions

When you use the `create_block_abstraction` command, the tool

- Identifies the interface logic.

The following netlist objects are part of interface logic:

- All cells, pins, and nets in timing paths from input ports to registers or output ports.
- All cells, pins, and nets in timing paths to output ports from registers or input ports.
- Any logic in the connection from a master clock to generated clocks.
- The clock trees that drive interface registers, including any logic in the clock tree.
- The longest and shortest clock paths from the clock ports.
- The side-load cells of all non-ideal and non-DRC disabled nets.

- Ignores an input or inout port if the percentage of the total registers in the transitive fanout of the port is greater than or equal to the threshold percentage that is specified by using the `abstraction_ignore_percentage` variable.

The following netlist objects are retained for ports that are ignored:

- Connections to the already identified interface logic of the other ports.
- Minimum and maximum critical timing paths.
- Ignores any case analysis settings. Any case analysis settings that are required must be specified at the top level by using the `set_case_analysis` command.
- Assumes all latches found in interface logic are potential borrowers; thus, all logic from I/O ports to flip-flops or output ports are identified as belonging to interface logic.
- Extracts and stores detailed parasitics as part of the abstraction information for each specified TLUPlus file and temperature.
- Calculates power consumption of the design and stores that information in attributes in the design. This information is used by the `report_power` command during the final assembly step of the entire chip.

To include additional leaf cells and nets in the block abstraction, use the `create_block_abstraction` command with the `-include` option. Logical hierarchical cells and pins are ignored.

If either of the following conditions exist, the entire block is included in the block abstraction:

- The number of leaf cells included exceeds 95 percent of the total leaf cells in the block.
- The number of nets included exceeds 95 percent of the total nets in the block.

Block Abstractions for Multicorner-Multimode Usage

The `create_block_abstraction` command automatically detects the presence of multiple scenarios (multiple modes or corners) and determines the interface logic for each scenario. The interface logic identified for each scenario is retained as the interface logic of the block abstraction. If an interface timing path is disabled in one scenario but enabled in another, the path is included in the interface logic.

Setting Top-Level Implementation Options

To use block abstractions at the top level, use the `set_top_implementation_options` command with the `-block_references` option to specify the list of block references that you want to link to the block abstraction. Do this for both Design Compiler block abstractions and IC Compiler block abstractions.

Use the `set_top_implementation_options` command before reading the .ddc file with the block abstraction into memory; otherwise, the full block netlist is loaded. When linking, the tool loads only the interface logic to the top level.

When using IC Compiler block abstractions, use the `set_top_implementation_options` command before running any `link` command in Design Compiler. The IC Compiler block abstractions are automatically read from the list of Milkyway reference libraries while linking the top-level design.

You can use multiple `set_top_implementation_options` commands to specify the options for all of the block abstractions at the top level.

The command settings in [Example 22-1](#) and [Example 22-2](#) are equivalent.

Example 22-1 Specifying Options for Block Abstractions in One Command

```
set_top_implementation_options -block_references {blk1 blk2}
```

Example 22-2 Specifying Options for Block Abstractions in Multiple Commands

```
set_top_implementation_options -block_references {blk1}
set_top_implementation_options -block_references {blk2}
```

To verify the options set by the `set_top_implementation_options` command and ensure that the block abstractions are loaded correctly, see [Reporting Implementation Options](#).

Transparent Interface Optimization

By default, block abstractions are not modified or optimized. If you are using Design Compiler in topographical mode, you can optimize the block abstractions using top-level transparent interface optimization. Transparent interface optimization is a process that optimizes the interface logic within the blocks during top-level synthesis to achieve faster timing closure.

To enable transparent interface optimization, use the `set_top_implementation_options -optimize_block_interface true` command and specify the list of blocks whose interface logic should be optimized during top-level synthesis before you run the `compile_ultra` or `compile_ultra -incremental` command. During top-level synthesis, Design Compiler links the block abstractions to the top-level design, optimizes and modifies each block's interface logic, updates the top-level interface logic, and performs full optimization.

Optimization of block abstractions is limited to cell sizing only; therefore, the `-size_only_mode` option has no effect. Transparent interface optimization is available only in Design Compiler in topographical mode for block abstractions created in topographical mode. Transparent interface optimization cannot be performed on block abstractions created in IC Compiler; these can only be optimized in IC Compiler.

The following example sets the top-level implementation options for the blk1 and blk2 blocks:

```
dc_shell-topo> set_top_implementation_options -block_references {blk1 blk2}
```

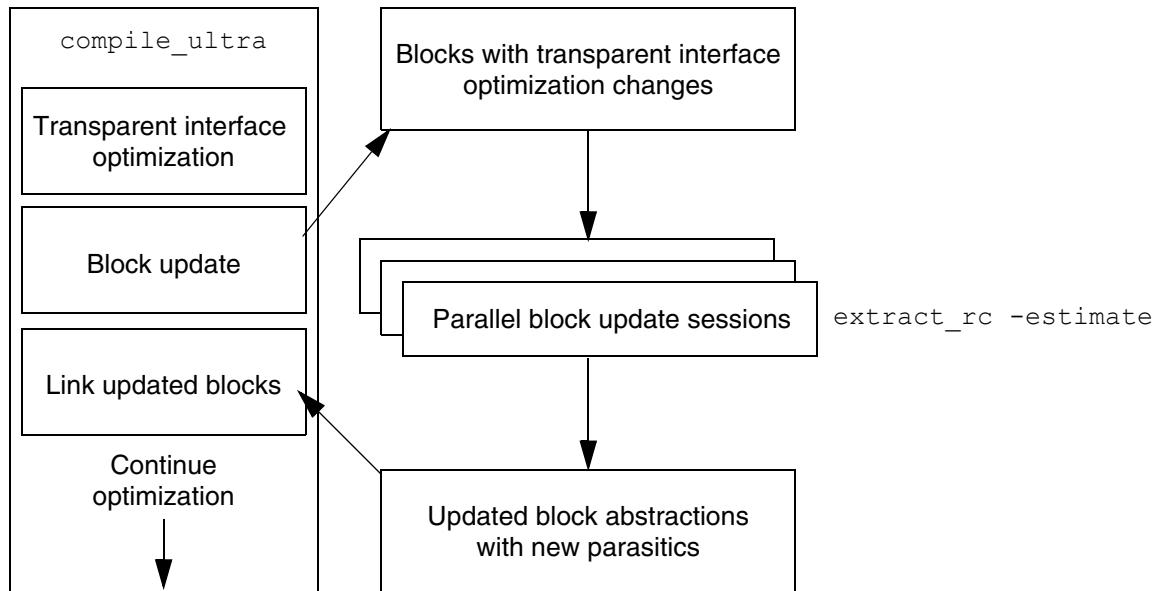
The following example enables transparent interface optimization for the blk1 block:

```
dc_shell-topo> set_top_implementation_options -block_references blk1 \
    -optimize_block_interface true
```

As a part of performing transparent interface optimization, parallel Design Compiler sessions are run during compile in order to update the parasitics of the block abstractions. Multicore optimization is used to run the block update sessions in parallel. You can control the number of cores that are used by using the `set_host_options` command. If multicore optimization is not enabled, all the updates are performed in serial in the same session.

[Figure 22-4](#) shows how this process works in compile. Transparent interface optimization performs sizing of the logic inside the block abstractions. Then, extraction of the updated parasitics is performed on the modified blocks using parallel Design Compiler sessions. Next, the updated block abstractions are reloaded back into the main compile session to continue the optimization.

Figure 22-4 Automatic Block Update in Transparent Interface Optimization



To specify a setup file to be sourced during the block update, use the `-block_update_setup_script` option with the `set_top_implementation_options` command. This is useful for passing variable settings for the block update process.

To specify a script file to run commands to update specific blocks, use the `-block_update_cmd_script` option. If you do not specify this option, Design Compiler automatically runs the `extract_rc -estimate` command to perform RC estimation for the block update. If you specify the option, Design Compiler sources the specified script file and does not run the `extract_rc -estimate` command automatically. Make sure that you include the `extract_rc -estimate` command in your command script file. Design Compiler automatically runs the `create_block_abstraction` and `write_file` commands after sourcing the script file.

The following example specifies the setup file to be sourced during the block update and a script to run the block update. You must specify the absolute paths to both file names. The example also enables multicore optimization to allow parallel processing of the blocks in multiple Design Compiler sessions.

```

dc_shell-topo> set_top_implementation_options -block_references {blkA blkB} \
              -optimize_block_interface true \
              -block_update_setup_script block_update_setup.tcl \
              -block_update_cmd_script block_update_command.tcl \
dc_shell-topo> set_host_options -max_cores 4
  
```

[Example 22-3](#) shows the flow used to automatically update the blocks after transparent interface optimization. [Example 22-4](#) shows the flow used to update the blocks when user-specified setup and command scripts are specified.

Example 22-3 Automatic Block Update Flow

```
set_app_var link_library ...
set_app_var target_library ...
set_app_var search_path ...
open_mw_lib top_MW
read_ddc block.tio.ddc
current_design blockA
link
extract_rc -estimate
create_block_abstraction
write_file -hierarchy -format ddc ...
```

Example 22-4 Block Update Flow Using User-Specified Setup and Command Scripts

```
set_app_var link_library ...
set_app_var target_library ...
set_app_var search_path ...
source -echo -verbose block_update_setup.tcl
open_mw_lib top_MW
read_ddc block.tio.ddc
current_design blockA
link
source -echo -verbose block_update_cmd.tcl
create_block_abstraction
write_file -hierarchy -format ddc ...
```

Each block update iteration creates a new subdirectory where all the parallel block update sessions are run. The directory names use a TIO_current_date prefix and increments a number suffix for each iteration:

```
TIO_20130923/
TIO_20130923.1/
TIO_20130923.2/
```

The compile log contains information about which transparent interface optimization block update directories were created during each compile step:

```
Information: Directory information for distributed jobs:
Main directory:
  .../TIO_20130923.1
Block CORE_A's distributed job's working directory:
  .../TIO_20130923.1/blk_CORE_A.1
Block CORE_B's distributed job's working directory:
  .../TIO_20130923.1/blk_CORE_B.1
```

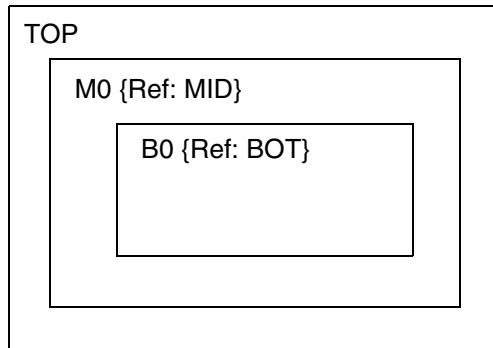
Creating Block Abstractions for Nested Blocks

You can use the `create_block_abstraction` command to create block abstractions for blocks that include nested block abstractions. The command does not create block abstractions with other types of physical hierarchy.

When you use nested block abstractions, you must specify the references of the blocks instantiated in all levels of the hierarchy, as shown:

```
dc_shell-topo> set_top_implementation_options -block_references {MID BOT}
```

Figure 22-5 Block Abstraction With Nested Physical Hierarchy



You can use transparent interface optimization to optimize the interface logic at any level.

For multicorner-multimode designs, you need to specify scenario mapping from the top level to each of the levels by using the `select_block_scenario` command.

Note:

Visibility of nested block abstractions in the layout view is not yet available.

Resetting Implementation Options

You can reset the top implementation options by using the `-reset` option with the `set_top_implementation_options` command. This causes the full design netlist to be loaded for the block instead of loading the block abstraction.

The following example resets the top implementation options for the blk2 block. In this example, the block abstraction for blk1 is loaded, and the full netlist for blk2 is loaded:

```
dc_shell-topo> set_top_implementation_options -block_references {blk1 blk2}
dc_shell-topo> set_top_implementation_options -block_references {blk2} -reset
```

Controlling the Extent of Logic Loaded for Block Abstractions

When loading blocks as block abstractions at the top level, you can control the extent of logic that is loaded, either loading the blocks with a full interface or a compact interface. When you create a block abstraction by using the `create_block_abstraction` command, the tool identifies both full interface logic and compact interface logic at the same time.

When you load blocks with a compact interface, only the min/max and rise/fall interface logic is retained. For blocks with ports that fan out to a large number of boundary registers, using a compact interface can improve runtime and memory usage.

To choose the type of interface logic to load at the top-level, use the `load_logic` option with the `set_top_implementation_options` command as follows:

- To load the entire block abstraction, consisting of all the interface logic, use the `-load_logic full_interface` option setting, which is the default.

When you load the full interface, the following logic is marked as interface logic:

- All the logic from input ports to registers or output ports
- All the logic to output ports from registers or input ports
- To load a compact version of the block abstraction, consisting of only the logic for the timing-critical paths of every input and output port, use the `-load_logic compact_interface` option setting.

For compact block abstractions, the following logic is marked as interface logic:

- The logic that belongs to min/max rise/fall from input ports to registers or output ports
- The logic that belongs to min/max rise/fall to output ports from registers or input ports

When transparent interface optimization is not performed on a block, you can load the block with a compact interface. However, when you perform transparent interface optimization at the top level with the `compile_ultra` command, load the entire block abstraction for the blocks you want to optimize by using the `-load_logic full_interface` option setting.

The following clock-gating circuitry is marked as interface logic whether you load the full interface or the compact interface:

- Any logic in the path from the master clock to the generated clocks
- The clock trees that drive interface registers, including any logic in the clock tree
- The longest and shortest clock paths from the clock ports
- The side-load cells of all non-ideal and non-DRC disabled nets

The tool retains the following IEEE 1801 Unified Power Format (UPF) objects in a block abstraction whether you load the full interface or the compact interface:

- All retention registers, whether or not they are part of the interface logic
- Level-shifter cells and isolation cells that are part of the interface logic
- Control logic for the retained retention registers, isolation cells, and level-shifter cells

When the tool removes a UPF netlist object, it either removes or updates the associated UPF constraint. The tool ensures that the UPF strategies stored in the block abstraction are consistent with the UPF netlist objects.

To propagate the UPF objects in the block abstraction to the top level, run the `propagate_constraints -power_supply_data` command.

See Also

- [Loading Block Abstractions](#)

Loading Block Abstractions

To load Design Compiler block abstractions to be linked to the top-level design, use the `read_ddc` command after you have used the `set_top_implementation_options` command:

```
dc_shell-topo> read_ddc {DesignA.mapped.ddc DesignB.mapped.ddc}
```

The following message indicates that the Design Compiler block abstraction is being loaded while reading in the block .ddc file:

```
Information: Reading block abstraction for design 'blk1' and its hierarchy. (DDC-27)
```

Make sure that your Milkyway reference libraries include the IC Compiler block abstraction designs. IC Compiler block abstractions are automatically loaded from the list of Milkyway reference libraries when linking the top-level design.

The following message indicates that the IC Compiler block abstraction is being loaded while linking the top-level design:

```
Information: Auto loading blk2.CEL (version 1) from Milkyway library blk2.mw ...  
(TL-102)
```

After a block abstraction is loaded, you can perform the tasks discussed in the following sections to ensure the block abstraction is ready for top-level synthesis:

- [Reporting Implementation Options](#)
- [Reporting Block Abstractions](#)
- [Querying Block Abstractions](#)
- [Checking Block Abstractions](#)

Reporting Implementation Options

To verify the options set by the `set_top_implementation_options` command and ensure that the block abstractions are correctly loaded, use the `report_top_implementation_options` command. The `report_top_implementation_options` command reports the options as shown in the following example:

```
dc_shell-topo> report_top_implementation_options
```

```
*****
Report      : Top implementation options
...
*****
```

```
-----
Block       Optimize     Optimize      Size Only   Host
Reference   Interface   Shared Logic  Mode        Options
-----
block1      true        false        off         --
block2      true        false        off         --
```

Reporting Block Abstractions

To report block abstractions that are linked to the current design, use the `report_block_abstraction` command. The command reports the following information:

- Block reference name
- Block instance name
- Block version
- Full path of the library from which the block was loaded
- Date and time of the last modification of the block
- Cell count and compression statistics for each of the blocks
- Cell count and compression statistics for the top-level design with blocks
- TLUPlus settings
- Parasitics data of the block
- User-defined multicorner-multimode scenario mappings between the top level and blocks
- UPF information

The `report_block_abstraction` command reports all levels of the hierarchy for nested block abstractions.

The following example shows a block abstraction report. The report includes a Design Compiler topographical block abstraction (blk1) and an IC Compiler block abstraction (blk2).

```
*****
Report  : Block Abstraction
Design  : top
...
#####
Top-level details
#####

-----
Block Instance          Reference   Orient    Location
-----
sub/blk1                blk1        0          (23.78, 1127.96)
sub/blk2                blk2        0          (809.74, 25.88)
-----

Top design leaf cell count statistics:
-----
Block instance count      : 2
Top only cell count       : 184148
Top + interface logic cell count : 239527
Top + Block cell count    : 320803
Compression percentage     : 25.34

Top-level and block scenario mapping:
-----
There is no user-specified scenario mapping available.

#####
Block reference : blk1
#####

File name: blk1.ddc
Last modified(mm/dd/yyyy hr:min:sec) : 3/6/2012 0:42:46

Leaf cell count statistics:
-----
Interface logic cell count      : 19142
Block cell count                 : 46219
Compression percentage           : 58.58

TLU+ files used:
-----
Block scenario #0: timing (mapped from top-level scenario timing)
Min TLU+ file      : min.tlup
Max TLU+ file      : max.tlup
Tech2ITF mapping file : tech.map

Block scenario #1: leakage (mapped from top-level scenario leakage)
Min TLU+ file      : min.tlup
```

```
Max TLU+ file           : max.tlup
Tech2ITF mapping file   : tech.map

#####
Block reference : blk2
#####

File name: blk2.mw/CEL/blk2:1
Last modified(mm/dd/yyyy hr:min:sec) : 3/5/2012 21:44:9

Leaf cell count statistics:
-----
Interface logic cell count      : 36237
Block cell count                 : 90436
Compression percentage          : 59.93

TLU+ files used:
-----
Block scenario #0: timing (mapped from top-level scenario timing)
Min TLU+ file      : min.tlup
Max TLU+ file      : max.tlup
Tech2ITF mapping file : tech.map

Block scenario #1: leakage (mapped from top-level scenario leakage)
Min TLU+ file      : min.tlup
Max TLU+ file      : max.tlup
Tech2ITF mapping file : tech.map
1
```

Reporting Block Abstraction Area

Design Compiler in topographical mode calculates the area of cells in a block abstraction as a standard cell area. This might result in higher utilization reported by the `report_area` command.

To enable the tool to calculate the area of cells in a block abstraction as a macro cell area, use the `block_abstraction_compute_area_as_macro` variable.

```
dc_shell-topo> set_app_var block_abstraction_compute_area_as_macro true
```

The default is `false`. When you set the variable to `true`, the area of cells in the block abstraction are calculated and reported in the macro portion.

Querying Block Abstractions

To create a collection of block abstractions, use the `get_cells` command:

```
dc_shell-topo> get_cells -hierarchical -filter is_block_abstraction==true
{blk1 blk2}
```

Checking Block Abstractions

Use the `check_block_abstraction` command to check the readiness of a block abstraction before optimization. The `check_block_abstraction` command checks the blocks linked to the top-level design to ensure they are ready for top-level synthesis.

The command checks for the following:

- Scenario consistency in multicorner-multimode blocks.
- The existence of `dont_touch` settings on blocks and cells.

All objects must have the `dont_touch` attribute so they cannot be modified.

The following example checks the blocks in the current design and reports the errors to be resolved:

```
dc_shell-topo> check_block_abstraction
Error: No scenario data loaded from block reference 'blk1'. (TL-151)
Error: dont_touch attribute incorrectly set to false on block design 'blk2'. (TL-121)
0
```

You can also use the `compile_ultra -check_only` command to verify that the design and libraries have all the data that `compile_ultra` requires to run successfully. The `-check_only` option is only available in Design Compiler in topographical mode.

Ignoring Timing Paths That Are Entirely Within Block Abstractions

A block abstraction can contain register-to-register paths that are entirely within the block. These paths are the result of combinational logic in the block abstraction that is shared by input-to-register and register-to-output paths. Paths entirely within blocks are optimized during block-level optimization, and ignoring them during top-level optimization enables transparent interface optimization to focus on other paths.

To ignore register-to-register paths that are entirely within block abstractions, set the `timing_ignore_paths_within_block_abstraction` variable to `true`. The default is `false`. Setting the variable to `true` does not affect paths that begin within a block abstraction, traverse outside, and return to the block abstraction.

Performing Top-Level Synthesis

After you create the block abstraction, specify the settings linking it to the top-level design, enable transparent interface optimization (if you are using topographical mode and want to optimize the block abstraction), run a report to check the top implementation settings, run a report to get details about the block abstraction, and check the readiness of the block abstraction, you can perform top-level synthesis.

The following script shows part of a top-level synthesis flow with transparent interface optimization enabled. The top-level design contains the DesignA and DesignB blocks and their corresponding block abstractions, DesignA.mapped.ddc and DesignB.mapped.ddc.

```
set BLOCK_ABSTRACTION_DESIGNS "DesignA DesignB"
set_top_implementation_options -block_references ${BLOCK_ABSTRACTION_DESIGNS} \
-optimize_block_interface true
... #Read in the top-level RTL
read_ddc {DesignA.mapped.ddc DesignB.mapped.ddc}
current_design top
link
report_block_abstraction
get_cells -hierarchical -filter is_block_abstraction==true
report_top_implementation_options
...# Read in constraints
check_block_abstraction
compile_ultra -check_only
compile_ultra
```

For the complete top-level hierarchical flow using block abstractions, see [Performing a Bottom-up Hierarchical Compile](#).

Saving Optimized Block Abstractions After Top-Level Synthesis

When you save the top-level design after optimization, block abstractions are not written to the top.ddc file. If you enabled transparent interface optimization, you need to save the updated blocks. Use the `write_file` command to save each optimized block abstraction to a separate .ddc file. You cannot write more than one block abstraction design in a single output .ddc file. The `write_file` command merges the changes with the original .ddc design and writes the complete block as a new .ddc file.

The `-hierarchy` and `-output` options are required with the `write_file` command when you write out block abstractions at the top level, and you must provide only the top-level block abstraction design name as an argument.

The following script shows how to save the top-level design and the optimized block abstractions into separate .ddc files:

```
write_file -hierarchy -format ddc -output top.mapped.ddc
foreach design ${BLOCK_ABSTRACTION_DESIGNS} {
    write_file -hierarchy -format ddc \
        -output ${design}.mapped_tio.ddc \
        ${design}
}
```

Limitations

Block abstractions have the following limitations:

- IC Compiler does not support the loading of block abstractions that were created in Design Compiler.
- You cannot view nested block abstractions in the layout view.

23

Incremental ASCII Flow With a Third-Party DFT Flow Example

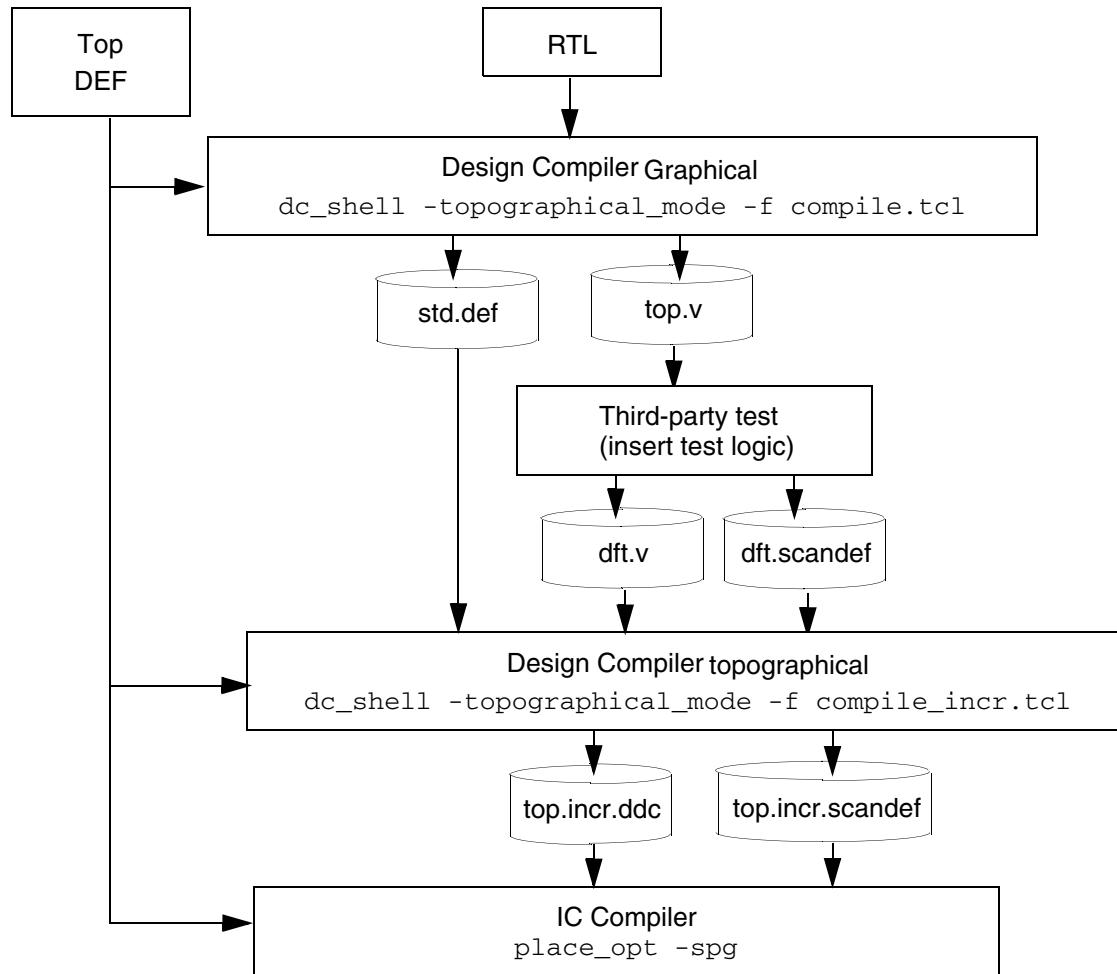
Some flows require you to use a Verilog or VHDL (ASCII) netlist output format. The following topics describe the incremental ASCII flow using a third-party design-for-test (DFT) flow as an example. The flow is described in `compile.tcl` and `compile_incr.tcl` scripts that are typical Tcl scripts that you can use for the initial compilation and incremental compilation flows.

Note:

- A DFTMAX license is required for the flow in this example.
- [Performing Initial Compilation Using the `compile.tcl` Script](#)
- [Performing Incremental Compilation Using the `compile_incr.tcl` Script](#)
- [Using the Netlist SCANDEF Flow](#)

Figure 23-1 shows an example of an incremental ASCII physical guidance flow with a third-party DFT flow.

Figure 23-1 Incremental ASCII Physical Guidance Flow With a Third-Party DFT Flow



Performing Initial Compilation Using the `compile.tcl` Script

[Example 23-1](#) shows a `compile.tcl` script that you can use for a typical initial `compile_ultra` physical guidance flow. It uses Design Compiler Graphical floorplan exploration after the `compile_ultra` step to generate the complete floorplan constraints that contain the standard cell placement. The floorplan exploration step is run with the `start_icc_dp -f icc_dp.tcl` command.

Example 23-1 Initial Compilation Using the `compile.tcl` Script

```
# Analyze and elaborate step
analyze -f verilog top_design.v
elaborate top_design

# Load UPF constraints
load_upf upf_constraints.upf

# Set timing constraints
source timing_constraints.tcl

# Set power settings if required
# set_clock_gating_style
# set_max_leakage_power
# set_max_dynamic_power

extract_physical_constraints -allow_physical_cells top.def

compile_ultra -spg -scan -gate_clock

change_names -rules verilog -hierarchy
set write_sdc_output_net_resistance false
set write_sdc_output_lumped_net_capacitance false
write_sdc top.sdc
save_upf top.upf
write_file -format verilog -hierarchy -output top.v
start_icc_dp -f icc_dp.tcl
```

The `icc_dp.tcl` script generates the standard cell placement so you can reuse it during the `compile_ultra -incremental` step. [Example 23-2](#) shows the commands included in the `icc_dp.tcl` script:

Example 23-2 Commands in `icc_dp.tcl` Script

```
write_def -version 5.7 -component -verbose -output std.def
exit
```

Performing Incremental Compilation Using the `compile_incr.tcl` Script

[Example 23-3](#) shows a `compile_incr.tcl` script that you can use for a typical incremental `compile_ultra` physical guidance flow. The script reads in the Verilog netlist after DFT insertion and reads the placement from the first `compile_ultra` run so that the incremental step can use the standard cell placement as seed placement during placement optimization.

If your initial compilation used physical guidance, as is the case in [Example 23-3](#), you must specify the `-standard_cell spg` option with the `extract_physical_constraints` command to read the standard cell placement in addition to the floorplan constraints. When you run the incremental compilation, specify the `-spg` option.

If your initial compilation did not use physical guidance, use the `-standard_cell topo` option with the `extract_physical_constraints` command to read the standard cell placement in addition to the floorplan constraints. If you do not use the `-standard_cell topo` option, Design Compiler is forced to redo a full placement. This can hurt runtime and correlation because the incremental step cannot use the standard cell placement from the original `compile_ultra` run.

The `compile_incr.tcl` script also reads the SCANDEF file from the DFT insertion step using the `read_scan_def` command and runs the `write_scan_def` command after the `compile_ultra -incremental` command in order to pass the SCANDEF file with all the changes performed during the incremental optimization to IC Compiler. You must use the `-output` option with the `write_scan_def` command if the handoff to IC Compiler is in ASCII format.

For more information about using the netlist SCANDEF flow to import and export scan chain information when the test implementation has been performed outside the Synopsys flow, see [“Using the Netlist SCANDEF Flow” on page 23-5](#).

After you run the `compile_ultra -incremental` command step, you can save the design in .ddc format, along with the SCANDEF file, and proceed with physical implementation in IC Compiler with the `place_opt -optimize_dft` command.

Example 23-3 Incremental Compilation Using the `compile_incr.tcl` Script

```
# Read the DFT netlist
read_verilog top.dft.v

# Load UPF constraints
load_upf top.upf

# Set timing constraints
read_sdc top.sdc

# Restore floorplan constraints and standard cell placement from the
# compile_ultra step
extract_physical_constraints -allow_physical_cells top.def
```

```
extract_physical_constraints -allow_physical_cells \
                             -standard_cell spg std.def

# Read the SCANDEF file from the DFT insertion step
set dct_placement_ignore_scan true
read_scan_def dft.scandef

compile_ultra -incremental -spg

write_scan_def -output top.incr.scandef
write_file -format ddc -hierarchy -output top.incr.ddc
```

Using the Netlist SCANDEF Flow

Design Compiler can import and export scan chain information when the test implementation has been performed outside the Synopsys flow with the netlist SCANDEF flow. In the netlist SCANDEF flow, you use the `read_scan_def` and `write_scan_def` commands to identify and preserve the scan chain structure and write out the new scan chain structure after optimization.

The scan chains are described in a SCANDEF file. The SCANDEF format, which is part of the DEF open format, is used to describe the scan chain elements. You use the `read_scan_def` command to read the SCANDEF file, after reading the netlist, to identify all cells that are part of scan chains. This identification allows the `write_scan_def` command to generate the new SCANDEF file with the same scan chain structure after optimization. You can then pass the SCANDEF file with all the changes performed during the incremental optimization, such as ungrouping, to IC Compiler.

Design Compiler only preserves the scan chains. It does not reorder them. You cannot change or add new scan chains. Therefore, the `insert_dft` command is not supported between the `read_scan_def` and `write_scan_def` commands and errors out. The netlist SCANDEF flow does not support a hierarchical approach; Design Compiler does not read multiple SCANDEF files.

To learn how to use the netlist SCANDEF flow, see the following topics:

- [Identifying the Scan Chain Structure](#)
- [Performing Scan Chain Reordering in Design Compiler Graphical](#)
- [Writing Out the New Scan Chain Structure](#)

Identifying the Scan Chain Structure

Test structure includes scan chains as well as controlling logic. Controlling logic includes the scan enable, scan-in, scan-out, and test mode signals as well as other logic, such as compressors, decompressors, on-chip clocking (OCC), and memory BIST. The netlist SCANDEF flow identifies scan chains only.

Scan chains are identified by SCANDEF files. SCANDEF files can be generated by most DFT tools. [Example 23-4](#) shows an example of a SCANDEF file.

Example 23-4 SCANDEF File Example

```
VERSION 5.5 ;
NAMECASESENSITIVE ON ;
DIVIDERCHAR "/" ;
BUSBITCHARS "[]" ;
DESIGN TOP ;
SCANCHAINS 1 ;
- 1
+ START U20 o
+ FLOATING regA ( IN ti ) ( OUT q )
    regB ( IN ti ) ( OUT q )
    regF ( IN ti ) ( OUT q )
    regE ( IN ti ) ( OUT q )
+ ORDERED regC ( IN ti ) ( OUT q )
    U10 ( IN i_1 ) ( OUT o )
    regD ( IN d ) ( OUT q )
+ PARTITION clk_i_45_45
+ STOP regG ti ;
```

To identify all cells that are part of a scan chain, read the SCANDEF file using the `read_scan_def` command after you read the top-level design. The `read_scan_def` command performs the following tasks automatically:

- Reads the SCANDEF file
- Checks the SCANDEF file for consistency with the netlist

[Example 23-5](#) shows a `read_scan_def` consistency report. If the scan chains pass all structural checks, the Status column reports a V for VALIDATED. If a scan chain fails the structural checks, the Status column reports an F for FAILED. This example shows that the `top.scandef` SCANDEF file is consistent with the netlist.

Example 23-5 read_scan_def Consistency Report

```
dc_shell> read_scan_def top.scandef
Reading DEF file top.scandef
Checking SCANDEF...
Checking for duplication in SCANDEF...
Checking scan cell correspondence between SCANDEF and netlist...
```

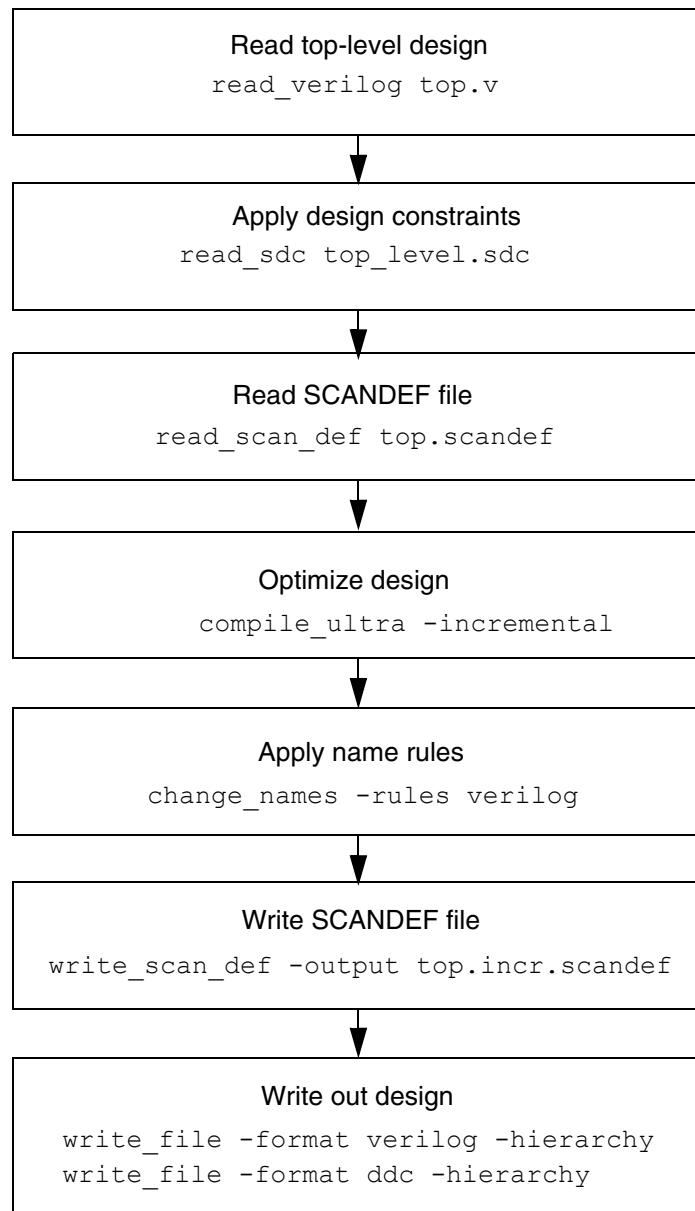
```
Start scan chain checking...
All checks completed.
*****
Report : Scan DEF check
Design : top
...
...
*****
Information from SCANDEF file:
Number of SCANCHAINS: 3
Checking between SCANDEF file and design:
Total SCANCHAINS checked: 3
VALIDATED : 3
FAILED    : 0

Chain name Status #cells PARTITION           Scan IN      Scan OUT
----- ----- ----- -----
1          V     2751  clk_45_45  scanin[1]  scanout[1]
2          V     2751  clk_45_45  scanin[2]  scanout[2]
3          V     2751  clk_45_45  scanin[3]  scanout[3]
```

Make sure that all scan chains are validated before you optimize the design. Any scan chains that fail are ignored and are not read in by the `read_scan_def` command.

Using the netlist SCANDEF flow, you can improve the timing of the functional logic while preserving the scan chain structure. [Figure 23-2](#) shows an example of a netlist SCANDEF flow. It is recommended that you use the `-incremental` option with the `compile_ultra` command. Other options, such as `-spg`, are also supported. Be sure to use the appropriate design constraint commands for flows, such as UPF or multicorner-multimode.

Figure 23-2 Netlist SCANDEF Flow Example



Performing Scan Chain Reordering in Design Compiler Graphical

In a Design Compiler Graphical flow, Synopsys physical guidance technology enables enhanced placement that is consistent with IC Compiler. The `-spg` option of the `compile_ultra` command enables this physical guidance technology.

In the netlist SCANDEF flow, you can perform an incremental compile using this physical guidance technology by using the following command:

```
dc_shell-topo> compile_ultra -scan -spg -incremental
```

In this case, Design Compiler further reduces scan chain wire length and congestion by performing scan chain reordering during the incremental compile, and the tool issues the following message:

```
Information: Performing SCANDEF-based scan chain reordering in the SPG flow. (SPG-127)
```

The reordering process honors any scan order constraints imposed by the input SCANDEF file. Scan reordering is enabled by default during incremental compile when you use the `-spg` and `-scan` options.

When reordering is enabled, repartitioning is also performed unless explicitly disabled with the `set_optimize_dft_options -repartitioning_method none` command.

Before running the incremental compile, you can use the `set_optimize_dft_options` command to configure scan repartitioning, just as you would in IC Compiler. For example,

```
dc_shell-topo> set_optimize_dft_options \
              -repartitioning_method single_directional
```

By default, repartitioning uses the `multi_directional` repartitioning method.

To disable both scan reordering and repartitioning, set the following variable:

```
dc_shell-topo> set_app_var \
              test_enable_scan_reordering_in_compile_incremental false
```

Note the following requirements and limitations:

- Hierarchical SCANDEF is not supported; only a flat DFT flow is supported.
- Post-DFT DRC is not supported after reordering. Perform DRC checking in the ATPG tool to verify scan chain integrity.

Writing Out the New Scan Chain Structure

When you run the `write_scan_def` command, Design Compiler generates a SCANDEF file that contains the same scan chain structure as the input SCANDEF file. It traces the scan chains based on the SCANDEF file that was read by the `read_scan_def` command. If the `write_scan_def` is unable to trace a scan chain because the scan chain element cannot be found in the netlist, Design Compiler issues an error.

In general, you do not need to use the `-output` option with the `write_scan_def` command. If you omit the option, the SCANDEF file is included in the saved .ddc file for handoff to IC Compiler. However, the `-output` option is required if the handoff to IC Compiler is in ASCII format.

The output SCANDEF file contains all changes performed during the incremental optimization and `change_names` command specification steps.

The `write_scan_def` command does not support the `-expand_elements` option in the netlist SCANDEF flow because the `read_scan_def` command does not support hierarchical flows.

24

Analyzing and Resolving Design Problems

Use the reports generated by Design Compiler to analyze and debug your design. You can generate reports both before and after you compile your design. Generate reports before compiling to check that you have set attributes, constraints, and design rules properly. Generate reports after compiling to analyze the results and debug your design.

To learn about analyzing and resolving design problems, see

- [Resolving Bus Versus Bit-Blasted Mismatches Between the RTL and Macros](#)
- [Fixing Errors Caused by New Unsupported Technology File Attributes](#)
- [Using Register Replication to Solve Timing QoR, Congestion, and Fanout Problems](#)
- [Assessing Design and Constraint Feasibility in Mapped Designs](#)
- [Checking for Design Consistency](#)
- [Checking Designs and Libraries Before Synthesis](#)
- [Analyzing Your Design During Optimization Using the Compile Log](#)
- [Analyzing Design Problems](#)
- [Analyzing Area](#)
- [Analyzing Timing](#)
- [Reporting Quality of Results](#)
- [Reporting Infeasible Paths](#)

- [Debugging Cells and Nets With dont_touch](#)
- [Reporting size_only Cells](#)
- [RTL Cross-Probing in the GUI](#)
- [RTL Cross-Probing on the Command Line](#)

Resolving Bus Versus Bit-Blasted Mismatches Between the RTL and Macros

Typically, the RTL pin names and the logical library pin names match. Signals are defined one way in both the RTL and the library. They are defined as buses or the bus is defined by its individual wires. Occasionally, mismatches occur during development when, for example, you port from one technology to another, or when the libraries and the RTL are in flux. When mismatches occur, you can set the `enable_bit_blasted_bus_linking` variable to `true` and read your design into Design Compiler, even though your RTL and libraries do not match with respect to bus versus bit-blasted pin names. When the `enable_bit_blasted_bus_linking` variable is set to `true`, the linker rules are relaxed such that you can read your design when mismatched pin names occur. The default for the `enable_bit_blasted_bus_linking` variable is `false`.

Note that when the `enable_bit_blasted_bus_linking` variable is set to `true`, the tool will match names in accordance with the `bus_inference_style` and `bus_inference_descending_sort` variable settings.

Fixing Errors Caused by New Unsupported Technology File Attributes

Occasionally, IC Compiler adds support for new technology file attributes that are not yet supported in Design Compiler. In these cases, a TFCHK-009 error message is issued in Design Compiler but not in IC Compiler when using the same technology file.

If you get TFCHK-009 errors when reading in your technology file, check the spelling of the attributes and make sure that the attributes are spelled correctly. If you still have TFCHK-009 errors, you can use either of the following two methods to remove the TFCHK-009 errors. Before using either of the following methods, make sure that you can safely remove or ignore the new attributes without affecting the tool functionality. In most cases, new attributes are associated with new routing rules and should not affect the functionality of Design Compiler.

- Remove the new attributes from the specified section in the technology file.
- If the new attributes are safe to ignore, you can set the `ignore_tf_error` variable to `true` which issues TFCHK-009w warnings for the unsupported attributes instead of TFCHK-009 errors.

Using Register Replication to Solve Timing QoR, Congestion, and Fanout Problems

Design Compiler can replicate registers to address timing quality of results (QoR), congestion, and fanout issues. This feature is supported in both Design Compiler wire load mode and topographical mode. In wire load mode, the load of the original replicated register is evenly distributed among the new replicated registers. In topographical mode, register replication is placement-aware, which can help reduce congestion. For more information, see [Using Register Replication](#).

Assessing Design and Constraint Feasibility in Mapped Designs

To help debug missing timing constraints and assess design and constraint feasibility in mapped designs, use the `set_zero_interconnect_delay_mode` command, as shown in [Example 24-1](#). In this example, you set the `set_zero_interconnect_delay_mode` command to `true`, run `report_qor`, and set the `set_zero_interconnect_delay_mode` command back to its default of `false` before proceeding with further optimization commands. When `set_zero_interconnect_delay_mode` is set to `true`, the tool analyzes your design with only cell delays and the capacitance of the pin-load on all the wires in the design to determine if your design meets timing goals. The tool does not consider wire capacitance due to timing paths.

Always set `set_zero_interconnect_delay_mode` back to its default of `false` before running your optimization step. The tool reports warning messages if you use optimization commands when `set_zero_interconnect_delay_mode` is `true`.

Example 24-1 set_zero_interconnect_delay_mode Sample Script

```
...
compile
...
set_zero_interconnect_delay_mode true
report_qor
set_zero_interconnect_delay_mode false
...
```

When `set_zero_interconnect_delay_mode` is set to `true`, the tool reports the following warning when you execute `report_constraint` or `report_qor`:

Warning: Timer is in zero interconnect delay mode. (TIM-177)

Checking for Design Consistency

A design is consistent when it does not contain errors, such as unconnected ports, constant-valued ports, cells with no input or output pins, mismatches between a cell and its reference, multiple driver nets, connection class violations, or recursive hierarchy definitions.

Design Compiler runs the `check_design -summary` command on all designs that are compiled. You can also run the command explicitly to verify design consistency. The command reports a list of warning and error messages as follows:

- It reports an error if it finds a problem that Design Compiler cannot resolve. For example, recursive hierarchy (when a design references itself) is an error. You cannot compile a design that has `check_design` errors.
- It reports a warning if it finds a problem that indicates a corrupted design or a design mistake not severe enough to cause the `compile` command to fail.

The report is displayed in standard output format by default. The report begins with a summary section that shows the type of check and how many violations have been detected. Next, it lists the error and warning messages.

You can specify a report in HTML format by using the `-html_file_name` option with the `check_design` command. When you specify this option and the file name, Design Compiler creates an HTML file in the current directory.

[Figure 24-1](#) shows an example of a `check_design` report in HTML format. The report organizes the information into sections, such as Input/Outputs, Cells, Designs, Nets, and so on. Each section lists the type of check and the number of violations. If there are violations, the number is an HTML link that you can click to display more information. The report in [Figure 24-1](#) shows five black box violations. Clicking the number 5 expands the report to show the five highlighted violations.

Figure 24-1 *HTML check_design Report Example*

Cells	
0	Cells with unconnected inputs (LINT-0)
520	Cells do not drive (LINT-1)
10	Cells do not have output pins (LINT-10)
1999	Connected to power or ground (LINT-32)
1374	Net is fed into multiple inputs (LINT-33)
0	Leaf pins connected to undriven nets (LINT-58)
20	Cells have undriven hier pins (LINT-59)
0	Hier pins without driver and load (LINT-60)
0	Output pin connected to constant (LINT-67)
Designs	
0	Design has no outputs ports (LINT-25)
5	Black box (LINT-55) <ul style="list-style-type: none">Design 'fsl_fler_dasm_instance_g0' does not contain any cells or nets.Design 'dummy_block' does not contain any cells or nets.Design 'pwr_sy' does not contain any cells or nets.Design 'pwr_tf' does not contain any cells or nets.Design 'pwr_mon' does not contain any cells or nets.
Nets	
9	Unloaded nets (LINT-2)

By default, during compilation or execution of the `check_design` command, Design Compiler issues a warning message if a tristate bus is driven by a non-tristate driver. You can have Design Compiler display an error message instead by setting the `check_design_allow_non_tri_drivers_on_tri_bus` variable to `false`. The default is `true`. When the variable is set to `false`, the `compile` command stops after reporting the error; however, the `check_design` command continues to run after the error is reported.

You can use options with the `check_design` command to filter the messages based on the type of check, such as warning messages related to ports, nets, and cells, information messages related to multiply-instantiated designs, and so on. For example, if you specify the `-multiple_designs` option, the report displays a list of multiply-instantiated designs along with instance names and associated attributes, such as `dont_touch`, `black_box`, and `ungroup`. By default, messages related to multiply-instantiated designs are suppressed.

Use the `-no_connection_class` option when you are working on a GTECH design or netlist in which connection class violations are expected. Doing so improves runtime, especially if the GTECH design is large and has several connection class violations.

Checking Designs and Libraries Before Synthesis

Before synthesizing your design, check that all designs and libraries have the necessary data for the compilation to run successfully by using the `compile_ultra` command with the `-check_only` option. The `-check_only` option reports potential problems that could cause the tool to stop during the `compile_ultra` run or to produce unsatisfactory correlation with your physical implementation. Use the `-check_only` option to help debug such problems as

- Missing TLUPlus files and missing physical library cells
- Multiple logic library cells with the same names
- Discrepancies in technology data between multiple physical libraries
- Missing placement location for cells and ports of physical hierarchical modules or block abstractions
- Missing `dont_touch` attributes for physical hierarchical modules or block abstractions
- Missing floorplan information such as core area constraints (through site row definition), port location, macro location, physical hierarchy location, and block abstraction location

Important:

Always execute the `compile_ultra` optimization step with the same set of options that were used when you executed the `compile_ultra -check_only` command.

Analyzing Your Design During Optimization Using the Compile Log

The compile log records the status of the compile run. Each optimization task has an introductory heading, followed by the actions taken while that task is performed. There are three tasks in which Design Compiler works to reduce the compile cost function:

- Delay optimization
- Design rule fixing
- Area optimization

While completing these tasks, Design Compiler performs many trials to determine how to reduce the cost function. For this reason, these tasks are collectively known as the trials phase of optimization.

By default, Design Compiler logs each action in the trials phase by providing the following information:

- Elapsed time

- Design area
- Worst negative slack
- Total negative slack
- Design rule cost
- Endpoint being worked on

You can customize the trials phase output by setting the `compile_log_format` variable. [Table 24-1](#) lists the available data items and the keywords used to select them.

For information about generating a log file in HTML format, see [Compile Log Files](#).

Table 24-1 Compile Log Format Keywords

Column	Column header	Keyword	Column description
Area	AREA	area	Shows the area of the design.
CPU seconds	CPU SEC	cpu	Shows the process CPU time used (in seconds).
Design rule cost	DESIGN RULE COST	drc	Measures the difference between the actual results and user-specified design rule constraints.
Elapsed time	ELAPSED TIME	elap_time	Tracks the elapsed time since the beginning of the current compile or reoptimization of the design.
Endpoint	ENDPOINT	endpoint	Shows the endpoint being worked on. When delay violations are being fixed, the endpoint is a cell or a port. When design rule violations are being fixed, the endpoint is a net. When area violations are being fixed, no endpoint is printed.
Maximum delay cost	MAX DELAY COST	max_delay	Shows the maximum delay cost of the design.
Megabytes of memory	MBYTES	mem	Shows the process memory used (in MB).

Table 24-1 Compile Log Format Keywords (Continued)

Column	Column header	Keyword	Column description
Minimum delay cost	MIN DELAY COST	min_delay	Shows the minimum delay cost of the design.
Path group	PATH GROUP	group_path	Shows the path group of an endpoint.
Time of day	TIME OF DAY	time	Shows the current time.
Total negative slack	TOTAL NEG SLACK	tns	Shows the total negative slack of the design.
Trials	TRIALS	trials	Tracks the number of transformations that the optimizer tried before making the current selection.
Worst negative slack	WORST NEG SLACK	wns	Shows the worst negative slack of the current path group.

Analyzing Design Problems

[Table 24-2](#) shows the design analysis commands provided by Design Compiler.

Table 24-2 Commands to Analyze Design Objects

Object	Command	Description
Design	report_design report_area report_hierarchy report_resources	Reports design characteristics. Reports design size and object counts. Reports design hierarchy. Reports resource implementations.
Instances	report_cell	Displays information about instances.
References	report_reference	Displays information about references.
Pins	report_transitive_fanin report_transitive_fanout	Reports fanin logic. Reports fanout logic.

Table 24-2 Commands to Analyze Design Objects (Continued)

Object	Command	Description
Ports	report_port	Displays information about ports.
	report_bus	Displays information about bused ports.
	report_transitive_fanin	Reports fanin logic.
	report_transitive_fanout	Reports fanout logic.
Nets	report_net	Reports net characteristics.
	report_bus	Reports bused net characteristics.
	report_transitive_fanin	Reports fanin logic.
	report_transitive_fanout	Reports fanout logic.
Clocks	report_clock	Displays information about clocks.

Analyzing Area

Use the `report_area` command to display area information and statistics for the current design or instance. The command reports combinational, non-combinational, and total area. If you have set the current instance, the report is generated for the design of that instance; otherwise, the report is generated for the current design. Use the `-hierarchy` option to report area used by cells across the hierarchy.

[Example 24-2](#) shows a report generated by the `report_area` command:

Example 24-2 report_area Report Example

```
prompt> report_area
*****
Report : area
Design : TEST_TOP
...
*****
Library(s) Used:
test_lib (File: test_lib.db)

Number of ports: 565
Number of nets: 9654
Number of cells: 8120
Number of combinational cells: 6594
Number of sequential cells: 1526
Number of macros: 0
Number of buf/inv: 1439
```

Number of references:	163
Combinational area:	562404.432061
Noncombinational area:	6720840.351067
Net Interconnect area:	undefined (Wire load has zero net area)
Total cell area:	7283244.783128
Total area:	undefined

Analyzing Timing

Use the `report_timing` command to generate timing reports for the current design or the current instance. By default, the command lists the full path of the longest maximum delay timing path for each path group. Design Compiler groups paths based on the clock controlling the endpoint. All paths not associated with a clock are in the default path group. You can also create path groups by using the `group_path` command.

Before you begin debugging timing problems, verify that your design meets the following requirements:

- You have defined the operating conditions.
- You have specified realistic constraints.
- You have appropriately budgeted the timing constraints.
- You have properly constrained the paths.
- You have described the clock skew.

If your design does not meet these requirements, make sure it does before you proceed.

After producing the initial mapped netlist, use the `report_constraint` command to check your design's performance.

[Table 24-3](#) lists the timing analysis commands.

Table 24-3 Timing Analysis Commands

Command	Analysis task description
<code>report_design</code>	Shows operating conditions, wire load model and mode, timing ranges, internal input and output, and disabled timing arcs.
<code>check_timing</code>	Checks for unconstrained timing paths and clock-gating logic.
<code>report_port</code>	Shows unconstrained input and output ports and port loading.
<code>report_timing_requirements</code>	Shows all timing exceptions set on the design.

Table 24-3 Timing Analysis Commands (Continued)

Command	Analysis task description
report_clock	Checks the clock definition and clock skew information.
report_path_group	Shows all timing path groups in the design.
report_timing	Checks the timing of the design.
report_constraint	Checks the design constraints.
report_delay_calculation	Reports the details of a delay arc calculation.

Reporting Quality of Results

You can generate a report on the quality of results (QoR) for the design in its current state by using reporting commands, such as `create_qor_snapshot`, `query_qor_snapshot`, and `report_qor`. The `create_qor_snapshot` command measures and reports the quality of the design in terms of timing, design rules, area, power, congestion, clock tree synthesis, routing, and so on. It stores the quality information in a set of snapshot files. You can later retrieve the snapshot with the `query_qor_snapshot` command and view the information in a categorized timing report. You can also selectively retrieve, sort, and display the information based on your preferences. The `report_qor` command displays QoR information and statistics for the current design.

For information about generating and viewing QoR reports, see

- [Measuring Quality of Results](#)
- [Analyzing Quality of Results](#)
- [Displaying Quality of Results](#)

Measuring Quality of Results

Note:

The `create_qor_snapshot` command is available only in Design Compiler in topographical mode.

To measure the quality of results of the design in its current state and store the quality information in a set of report files, use the `create_qor_snapshot` command. You can capture the QoR information when using different optimization strategies or at different stages of the design and compare the quality of results. For example, you can use the

`create_qor_snapshot` command to create a snapshot after the first time you compile the design, after DFT insertion, and again after an incremental compilation.

The command options let you specify the conditions for analysis, such as zero wire load, maximum paths per timing group, and maximum paths per endpoint.

When you use the `create_qor_snapshot` command to create a snapshot, you must at least specify a name for the snapshot by using the `-name` option. For example,

```
dc_shell-topo> create_qor_snapshot -name snapshot1
```

The report is written to a set of files in a directory called “snapshot” in the current working directory. The set of files act as a database for the snapshot report. You do not need to access these files, and you must not modify them. Later, when you run the `query_qor_snapshot` command and specify the `-name` option with the same name you used to create the snapshot (snapshot1 in this example), the `query_qor_snapshot` command displays the results in text or HTML format.

Analyzing Quality of Results

Note:

The `query_qor_snapshot` command is available only in Design Compiler in topographical mode.

To read a QoR snapshot previously generated by the `create_qor_snapshot` command, analyze the results, and display the information in a categorized timing report, use the `query_qor_snapshot` command. The `query_qor_snapshot` command does not perform any additional analysis of the design but merely processes the report information already saved in the snapshot files.

The `query_qor_snapshot` command can display the categorized timing report in HTML format or in text format. If you open the report in HTML format, you can modify the constraints and generate a new categorized timing report. The text version of the report allows you to view the results but does not allow you to make modifications.

Use the following command to specify a snapshot and specify the name of the generated output file. You can include an .html or .txt file extension to specify the file format. If you do not specify a file format, Design Compiler creates both a text and an HTML output file.

```
dc_shell-topo> query_qor_snapshot -name snapshot1 -output_file file_name
```

Figure 24-2 shows a categorized timing report in HTML format.

Figure 24-2 Categorized Timing Report in HTML Format

<input type="checkbox"/> Create Exceptions <input checked="" type="checkbox"/> Append <input type="checkbox"/> Overwrite								
<input type="checkbox"/> False Paths <input type="checkbox"/> Multicycle Paths <input type="checkbox"/> Max Delay <input type="checkbox"/> Input Delay <input type="checkbox"/> Output Delay								
Start/Endpoints: <input type="checkbox"/> From-To <input type="checkbox"/> Through-To <input type="checkbox"/> From Only <input type="checkbox"/> To Only <input type="checkbox"/> Through Only								
Delays: <input type="checkbox"/> Rising and Falling <input type="checkbox"/> Rising <input type="checkbox"/> Falling								
Add Delay: <input type="checkbox"/> Clock Fall: <input type="checkbox"/>								
Input file: /remote/...								
Filters: -wns ,0 -zero_path ,0 -fanout 40 -logic_levels 50								
And Columns: none								
Sort Column: wns (ascending)								
Number of Paths: 59								
Exceptions	Path Group	Start Point	End Point	WNS	Zero Path	Path Delay	Input Delay	
<input type="checkbox"/> Clear	<input type="checkbox"/> clk	REGS1/c_reg/CK	REGS2/int1_reg/D	-0.8740	-0.2760	0.2760	undefined	
<input type="checkbox"/>	<input type="checkbox"/> clk	REGS1/a_reg/CK	REGS2/int1_reg/D	-0.8730	-0.2760	0.2760	undefined	
<input type="checkbox"/>	<input type="checkbox"/> clk	REGS1/c_reg/CK	REGS2/int1_reg/D	-0.7860	-0.2760	0.2760	undefined	
<input type="checkbox"/>	<input type="checkbox"/> clk		en	REGS1/a_reg/E	-0.6740	-0.6740	0.3240	0.3500
2.0	<input type="checkbox"/> clk		en	REGS1/a_reg/E	-0.6740	-0.6740	0.3240	0.3500

The HTML report lets you quickly find paths with certain problems, such as large fanouts or transition degradation. You can then modify the constraints and generate a new HTML report based on the constraints you specified. To change the constraints and generate a new HTML report, perform the following steps:

1. Select a parameter that you want to change.

For example, in **Figure 24-2**, Input Delay is selected.

2. Enter a new value for a specific path or paths.

In **Figure 24-2**, a new input delay value of 2.0 is set on the enable input pin of the REGS1/a_reg/E register.

3. Click the Create Exceptions button.

Design Compiler opens a new browser window that contains a set of commands based on the constraints you specified. Update your SDC constraints with these commands, and then rerun Design Compiler.

The `query_qor_snapshot` command options let you filter and sort the search results. You can specify which data columns to display. For example, you can generate `set_false_path` constraints for a specified value range, such as WNS, logic level, and so on. (The `set_false_path` command removes timing constraints from specified paths that you know do not affect circuit operation.)

You apply filters to the paths by using the `-filters` option. For example, the following command reports all paths that have a worst negative slack less than zero:

```
dc_shell-topo> query_qor_snapshot -name my_snapshot -filters "-wns ,0"
```

The following example reports all paths that meet any of the specified filter requirements:

```
dc_shell-topo> query_qor_snapshot -name my_snapshot \
    -filters "-wns -4.0 -fanout 20"
```

If you do not specify the `-filters` option, the `query_qor_snapshot` command automatically applies a set of default filters. The default filter settings for the maximum condition are shown in the following example:

```
dc_shell-topo> query_qor_snapshot -name my_snapshot \
    -filters "-wns ,0 -zero_path ,0 -fanout 40 -logic_level 50"
```

For a complete list of filters, see the `query_qor_snapshot` man page.

The following additional examples demonstrate the `query_qor_snapshot` command usage.

Example 24-3 analyzes the snapshot named `my_snapshot1` and reports the paths having a worst negative slack between `-4.0` and `-3.0` time units.

Example 24-3 Reporting Paths With Worst Negative Slack

```
dc_shell-topo> query_qor_snapshot -name my_snapshot1 -filters "-wns
-4.0,-3.0"
...
Path Group      Start Point      End Point      WNS
...            ...              ...          -3.312
...            ...              ...          -3.277
```

Example 24-4 reports the paths having a slack worse than `-1.0` time units and a fanout greater than 40.

Example 24-4 Reporting Paths for Slack and Fanout

```
dc_shell-topo> query_qor_snapshot -name my_snapshot2 \
    -filters "-wns ,-1.0 -fanout 40"
...
```

Path Group	Start Point	End Point	WNS	Large Fanout
...	-3.312	42
...	-3.277	42

[Example 24-5](#) reads the snapshot named `max_logic_level` and specifies the `-filters` option to report all paths between logic levels 2 and 14.

Example 24-5 Reporting All Paths Between Specified Logic Levels

```
dc_shell-topo> query_qor_snapshot -name max_logic_level \
    -filters "-logic_level 2,15"
```

Note that the paths of logic level 2 are included in the report, but the paths of logic level 15 are excluded.

Sometimes the paths with the worst timing start in one hierarchical block and end in another. Identifying and grouping these paths can be a challenging task. The `-hierarchy` option enables the top-level view and automates the process of finding potential path groupings that cross hierarchical boundaries so that the problem paths can be analyzed more efficiently. When you use the `-hierarchy` option, you must specify the `-to`, `-from`, or `-through` option. [Example 24-6](#) finds the violating paths that cross hard macros starting at module A and ending at module B.

Example 24-6 Reporting Timing Violations Across Hierarchical Boundaries

```
dc_shell-topo> query_qor_snapshot -name snap_shot3 -hierarchy \
    -from {A/*} -to {B/*}
```

The `query_qor_snapshot` command automatically generates a report of all violating paths between the extracted timing models (ETMs), block abstractions, and hard macros. You can apply filters or expand the top-level paths by specifying the `-to`, `-from`, and `-through` options. [Example 24-7](#) finds violating paths that pass through modules A, B, or C:

Example 24-7 Reporting Violating Paths That Pass Through Specified Modules

```
dc_shell-topo> query_qor_snapshot -name snap_shot4 -hierarchy \
    -through {A B C}
```

[Example 24-8](#) uses the `-incremental` option, which keeps the previous analysis results (through modules A, B, and C), and applies an additional query about paths starting from module A or B and ending at module A or B.

Example 24-8 Incremental Reporting

```
dc_shell-topo> query_qor_snapshot -hierarchy -incremental \
    -from {A/* B/*} -to {A/* B/*}
```

[Example 24-9](#) reports paths that start from module C and end at module B, having a worst negative slack worse than -3.0 ns or a fanout more than 40.

Example 24-9 Reporting Paths for WNS and Fanout Passing Through Specified Modules

```
dc_shell-topo> query_qor_snapshot -name my_snapshot5 \
```

```
-hierarchy -from top/A/C/* -to top/B/* \
-filters "-wns ,-3.0 -fanout 40"
```

Displaying Quality of Results

To display information about the quality of results and other statistics for the current design, use the `report_qor` command. The command reports information about timing path group details and cell count and current design statistics, including combinational, noncombinational, and total area. The command also reports static power, design rule violations, and compile time details. Note that the `report_qor` command is not part of the `create_qor_snapshot` and `query_qor_snapshot` command set.

[Example 24-10](#) shows a report generated by the `report_qor` command. In the Overall Compile Wall Clock Time section, the reported time is the runtime of the compile command run just before the execution of the `report_qor` command. If the `report_qor` command is run just after reloading the .ddc file in a new Design Compiler session (without running any compile), the reported wall clock time is the runtime of the last compile command that was run in the previous session.

In the Cell Count section, the report shows the number of macros in the design. To qualify as a macro cell, a cell must be nonhierarchical and have the `is_macro_cell` attribute set on its library cell.

Example 24-10 report_qor Report Example

```
prompt> report_qor

*****
Report : qor
Design : TEST_TOP
...
*****

Timing Path Group 'clk'
-----
Levels of Logic:          1.00
Critical Path Length:    0.02
Critical Path Slack:     -0.68
Critical Path Clk Period: 8.00
Total Negative Slack:    -393.61
No. of Violating Paths:  1066.00
Worst Hold Violation:   0.00
Total Hold Violation:   0.00
No. of Hold Violations: 0.00
-----

Cell Count
-----
Hierarchical Cell Count: 7
Hierarchical Port Count: 2360
```

```
Leaf Cell Count:          43978
Buf/Inv Cell Count:      6764
CT Buf/Inv Cell Count:   4
Combinational Cell Count: 37212
Sequential Cell Count:    6773
Macro Count:              0
-----
Area
-----
Combinational Area:     562404.432061
Noncombinational Area:   6720840.351067
Net Area:                0.000000
Net XLength :            2836623.25
Net YLength :            2555007.75
-----
Cell Area:               7283244.783128
Design Area:              7283244.783128
Net Length :             5391631.00

Design Rules
-----
Total Number of Nets:    47207
Nets With Violations:   5
Max Trans Violations:   3
-----
Hostname: machine
Compile CPU Statistics
-----
Resource Sharing:         21.54
Logic Optimization:       182.63
Mapping Optimization:     230.79
-----
Overall Compile Time:     631.32
Overall Compile Wall Clock Time: 288.11
```

Reporting Infeasible Paths

Infeasible timing paths can exist for the following reasons:

- Missing boundary conditions
- Unrealistic timing goals for input or output delays

During optimization, the `compile_ultra` command automatically detects infeasible paths in the design. To control the focus of optimization effort on other paths, the command temporarily sets these infeasible paths as false paths and issues an OPT-1720 information message. At the end of optimization, these false path settings are removed.

After running the `compile_ultra` command, you can report the infeasible paths that were temporarily set as false paths by using either of these methods:

- [Reporting Infeasible Paths in an HTML Categorized Timing Report](#)
- [Reporting Infeasible Paths in a Timing Report](#)

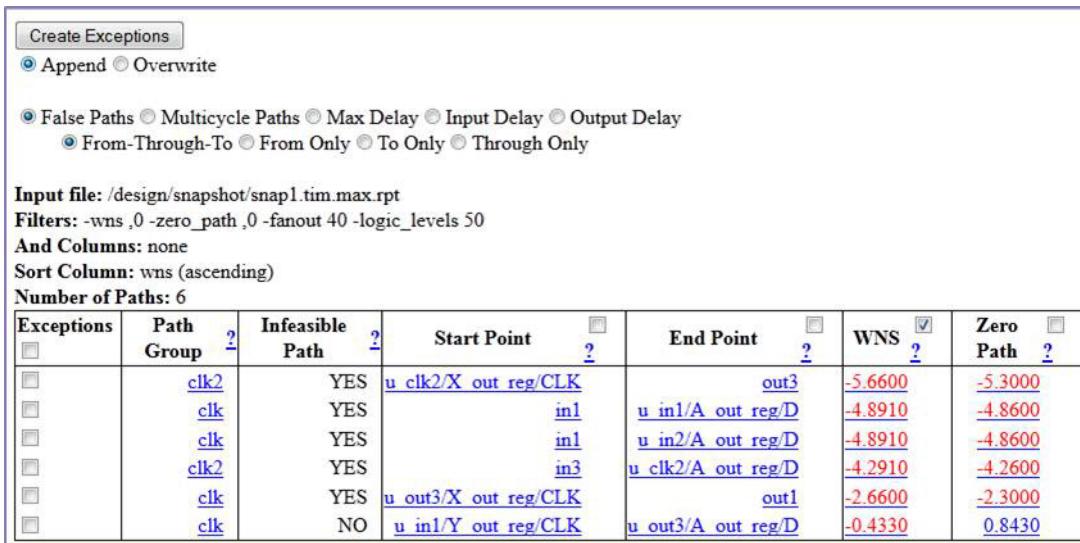
Reporting Infeasible Paths in an HTML Categorized Timing Report

You can view the tool-detected infeasible paths in a categorized timing report by specifying the `-infeasible_paths` option with the `create_qor_snapshot` and `query_qor_snapshot` commands in Design Compiler in topographical mode. You can use the `-infeasible_paths` option with any of the `create_qor_snapshot` and `query_qor_snapshot` options to generate a customized report that indicates which paths are detected by the tool as infeasible. In the following example, the maximum number of paths in the report is limited to 30.

```
dc_shell-topo> compile_ultra
dc_shell-topo> create_qor_snapshot -max_paths 30 -infeasible_paths \
    -name snap1
dc_shell-topo> query_qor_snapshot -name snap1 -infeasible_paths \
    -columns {path_group infeasible_paths startpoint \
    endpoint wns zero_path} -output_file snap1.html
dc_shell-topo> sh firefox snap1.html
```

[Figure 24-3](#) shows the categorized timing report in HTML format generated by the previous command sequence. If a path was detected as infeasible during optimization, it is indicated with a YES under the Infeasible Path column.

Figure 24-3 Categorized Timing Report in HTML Format



The screenshot shows a web-based categorized timing report. At the top, there are several configuration options:

- Create Exceptions** button
- Append** () and **Overwrite** () radio buttons
- False Paths** () and **Multicycle Paths** () radio buttons
- Max Delay** () and **Input Delay** () radio buttons
- Output Delay** () and **From-Through-To** () radio buttons
- From Only** () and **To Only** () radio buttons
- Through Only** () radio button

Below these are input file and filter settings:

- Input file:** /design/snapshot/snap1.tim.max.rpt
- Filters:** -wns ,0 -zero_path ,0 -fanout 40 -logic_levels 50
- And Columns:** none
- Sort Column:** wns (ascending)
- Number of Paths:** 6

A table follows, showing the results of the analysis:

Exceptions	Path Group	Infeasible Path	Start Point	End Point	WNS	Zero Path
	clk2	YES	u_clk2/X out reg/CLK	out3	-5.6600	-5.3000
	clk	YES	in1	u_in1/A out reg/D	-4.8910	-4.8600
	clk	YES	in1	u_in2/A out reg/D	-4.8910	-4.8600
	clk2	YES	in3	u_clk2/A out reg/D	-4.2910	-4.2600
	clk	YES	u_out3/X out reg/CLK	out1	-2.6600	-2.3000
	clk	NO	u_in1/Y out reg/CLK	u_out3/A out reg/D	-0.4330	0.8430

You can also specify exceptions and create constraints from the categorized timing HTML report by following these steps:

1. Click Append.
2. Click False Paths.
3. Select the check box under the Exceptions column of each infeasible path for which you want to create an exception.
4. Click the Create Exceptions button.

A new browser window that contains a set of commands based on the exceptions you specified appears. For example, when you select the first five infeasible paths, the following commands are displayed in the pop-up window:

```
set_false_path -from u_clk2/X_out_reg/CLK -to out3;
set_false_path -from in1 -to u_in1/A_out_reg/D;
set_false_path -from in1 -to u_in2/A_out_regD;
set_false_path -from in3 -to u_clk2/A_out_reg/D;
set_false_path -from u_out3/X_out_reg/CLK -to out1;
```

5. Update your SDC file with these commands.

Reporting Infeasible Paths in a Timing Report

You can view the tool-detected infeasible paths by generating a timing report with the `report_timing -attributes` command in either Design Compiler in topographical mode or wire load mode, as shown in the following example. The `inf` attribute in the report denotes the infeasible paths that were detected during compile. The timing report shows an infeasible path whose startpoint and endpoint are marked with the `inf` attribute.

```
prompt> report_timing -attributes ...
...
Attributes:
  d - dont_touch
  u - dont_use
  mo - map_only
  so - size_only
  i - ideal_net or ideal_network
  inf - infeasible path

  Point          Incr      Path      Attributes
  -----
clock clk2 (rise edge)
clock network delay (ideal)
u_clk2/X_out_reg/CLK (DFFX1-LVT)    0.00    0.10 r      inf
u_clk2/X_out_reg/Q (DFFX1_LVT)      0.36    0.46 f
u_clk2/X_out (unit_1)                0.00    0.46 f
out 3 (out)                         0.00    0.46 f      inf
data arrival time                   0.46
```

...

To exclude all paths that are flagged as infeasible from the report, use the `-ignore_infeasible_paths` option with the `report_timing` command. Similarly, you can use the `-ignore_infeasible_paths` option with the `report_constraint` and `report_qor` commands.

Debugging Cells and Nets With `dont_touch`

If Design Compiler did not optimize or remove a cell or net, it is often due to a `dont_touch` on the object. Explicit `dont_touch` attributes are straightforward to debug. However, it can be difficult to determine why an object has an implicit `dont_touch`, especially when there are multiple overlapping types of implicit `dont_touch` on the object. Even if you remove the `dont_touch` setting on the object, you might still see the object reported as having a `dont_touch`. This happens in cases where the object has a `dont_touch` for multiple reasons.

For information about debugging why an object in a cell or net is marked as `dont_touch`, see

- [Reporting `dont_touch` Cells and Nets](#)
- [Creating a Collection of `dont_touch` Cells and Nets](#)

Reporting `dont_touch` Cells and Nets

To report all the `dont_touch` cells and nets in the design and the types of `dont_touch` that apply to each reported object, use the `report_dont_touch` command. The `dont_touch` objects are reported for the complete design hierarchy.

Objects that are not `dont_touch` cells or nets are skipped. You can also specify a collection of cells or nets with the `report_dont_touch` command.

The following example shows a report of the `dont_touch` types for a specific net. In this example, the `report_dont_touch` command is used to investigate the types of `dont_touch` on the Multiplier/Product[0] net. The net is listed with all the types of `dont_touch` that are on the net. The table legend provides a detailed description for each type of `dont_touch` that is reported. There are many possible `dont_touch` types, but only the relevant types are listed.

```
prompt> report_dont_touch [get_nets Multiplier/Product[0]]
...
Description for Net dont_touch Types:
  inh_parent  - Net inherits dont_touch from parent
  mv_iso      - Prevents buffering between isolation cells and
                 power domain boundary

Object          Class        Types
-----
Multiplier/Product[0]    net        inh_parent, mv_iso
```

As the example shows, the Multiplier/Product[0] net inherited one `dont_touch` from a parent cell. This net is inside a hierarchical block on which an explicit `dont_touch` was set on the parent hierarchical cell. The net also has an implicit `dont_touch` on it as a result of multivoltage synthesis.

The following example shows a `dont_touch` report for all the cells in the design:

```
prompt> report_dont_touch -class cell -nosplit
...
Description for Cell dont_touch Types:
  inh_ref      - Cell inherits dont_touch from reference design or library cell
  ph_fixed_placement - Cell with fixed placement

Object          Class      Types
-----
headerfooter5_HDRID1BWPVHT_R5_C0    cell      ph_fixed_placement
headerfooter6_HDRID1BWPVHT_R6_C0    cell      ph_fixed_placement
u_des_soft_macro/u_8/u_mem         cell      inh_ref
-----
Total 3 dont_touch cells
1
```

To return an alphabetically sorted list of currently defined cell and net `dont_touch` types and their descriptions, use the `list_dont_touch_types` command. This command is useful when you want to look up the types available for building a collection of `dont_touch` objects. By default, the `list_dont_touch_types` command lists all `dont_touch` types for cell and net object classes.

Use the `-class class_name` option to limit the list of `dont_touch` types to either the cell or net object class. The `class_name` value can be either `cell` or `net`.

The following example shows an excerpt of the default `list_dont_touch_types` command output, which lists all `dont_touch` types for nets and cells:

```
prompt> list_dont_touch_types
*****
Report : Types of dont_touch
...
*****
Class  Type          Description
-----
net    cts_synthesized Net is synthesized with clock tree synthesis
net    dft_scanbuf    Net connected to input pin of scan compression buffer cell
net    dtm             Net in dont_touch network set by set_dont_touch_network command
net    dtm_charz       Net in dont_touch network through characterization
net    fp_abutted     Net is floorplan abutted net
net    fp_border       Net is dangling on floorplan border
net    idn             Net in ideal network set by set_ideal_network command
...
cell   cg_mo          Clock-gating cell has dont_touch derived from map_only setting
cell   charz          Cell has dont_touch derived from characterization
cell   dft_scan        Cell is a DFT scan cell
```

```

cell  dft_scandef      Cell is SCANDEF generation cell
cell  dft_scg          DFT scan cell has dont_touch because it is replaced by
                      clock-gating cell
cell  dt_conn          Cell connected to dont_touch net
cell  dtm              Cell in dont_touch network set by set_dont_touch_network
                      command
...
-----
1

```

Creating a Collection of `dont_touch` Cells and Nets

You can create a collection of `dont_touch` cells and nets in the current design, relative to the current instance, by using the `get_dont_touch_cells` and `get_dont_touch_nets` commands, respectively. If no cells or nets match the specified criteria, the command returns an empty string.

You can use the `get_dont_touch_cells` and `get_dont_touch_nets` commands at the command prompt, or you can nest them as an argument to another command, such as the `query_objects` command. You can also assign the command results to a variable.

When issued from the command prompt, the `get_dont_touch_cells` and `get_dont_touch_nets` commands behave as if the `query_objects` command has been called to display the objects in the collection. By default, the commands display a maximum of 100 objects. You can change this value by setting the `collection_result_display_limit` variable.

The following example queries the cells with `dont_touch` due to an `ideal_network` setting under the `InstDecode` block:

```
prompt> get_dont_touch_cells -type idn InstDecode/*
{InstDecode/reset_UPF_LS InstDecode/U38 InstDecode/U36}
```

The following example queries the multivoltage type `dont_touch` nets that begin with NET in a block named `block1`:

```
prompt> get_dont_touch_nets -type mv* block1/NET*
{block1/NET1QNX block1/NET2QNX}
```

Reporting `size_only` Cells

You can generate a report of cells with the `size_only` setting and why the cells have a `size_only` setting by using the `report_size_only` command. You can generate a report for all the cells in the design, a collection of cells, or a specific cell instance. The command reports `size_only` cells for the complete design hierarchy. Cells that are not `size_only` cells are skipped.

The `report_size_only` command can help debug your design. For example, run the `report_size_only` command before running the `compile_ultra` command to report the `size_only` settings from user constraints. Run the `report_size_only` command again after running `compile_ultra` to report additional `size_only` settings as a result of optimization.

The following example shows a report of all `size_only` cells in the current design. It shows a U1 cell with a `size_only` setting because it is an isolation cell. Cells can contain more than one `size_only` setting, as shown in cell A/U2. The A/U2 cell has a `size_only` setting because it is a cross-supply driver and it has an isolation-control signal pin.

```
prompt> report_size_only
*****
Report : size_only
*****

Description of size_only Types:
timing           - Cell with timing constraint
upf_cross_supply_dr - Cell is cross supply driver
upf_iso          - Isolation cell
upf_iso_ctrl     - Cell with isolation control signal pin
user             - Cell has specific user-set size_only attribute

Cell      Types
-----
U1        upf_iso
A/U2      upf_cross_supply_dr, upf_iso_ctrl
B/U3      timing
U5        user

...
-----
Summary:

Total 2026 cells with size-only

*****
Distribution By Size-Only Type
*****
Type            Size-Only
-----
timing          1299
upf_cross_supply_dr 4
upf_iso         716
upf_iso_ctrl    4
user            986
```

To return an alphabetically sorted list of cell `size_only` types and their descriptions, use the `list_size_only_types` command:

```
prompt> list_size_only_types
```

RTL Cross-Probing in the GUI

Design Compiler offers cross-probing capabilities in the GUI so you can check your RTL design in the RTL browser. You can cross-probe cells, pins, or timing paths in a design from one view, such as a schematic, cell list, layout, or timing view, and examine the corresponding RTL file. This allows you to identify lines in the code that can be modified to improve timing, congestion, or datapath extraction.

Note:

To use these cross-probing capabilities, you must have an elaborated or synthesized design from Design Compiler version I-2013.12 or later.

Design Compiler provides the following cross-probing capabilities in the GUI to the RTL file in the RTL browser:

- You can select a critical path and cross-probe to the corresponding RTL files to help debug the worst-case timing paths.
- You can cross-probe the cells in a highly congested area and inspect the RTL code for cells that are causing congestion.
- You can cross-probe from a datapath extraction or design resources report to the corresponding RTL file to help you understand the datapath extraction, which can help you improve the coding style for better extraction.

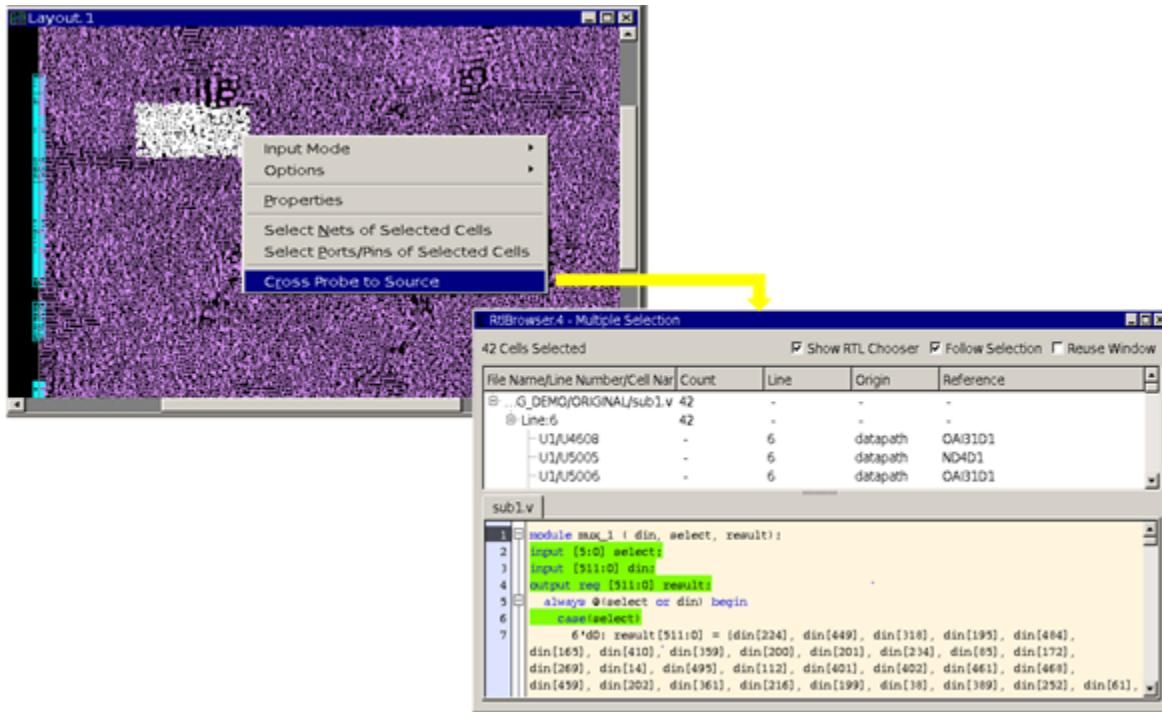
You can also open an RTL file that you used to elaborate or compile the design and cross-probe to cells associated with a line in the file. Similarly, you can cross-probe from an RTL file to the design resources report.

The tool does not allow you to

- Cross-probe from various design views to the UPF file.
- Select nets as cross-probing objects.

[Figure 24-4](#) shows cross-probing between the Design Vision layout view and the RTL browser.

Figure 24-4 Cross-Probing Between the Layout View and the RTL Browser



See Also

- The “Cross-Probing the RTL for Cells or Paths” topic in Design Vision Help

RTL Cross-Probing on the Command Line

To improve timing, congestion, or datapath extraction, you can check the RTL design and identify lines in the code to be modified by using the RTL cross-probing capabilities on the command line.

You must have a design that was analyzed and elaborated or synthesized using Design Compiler version I-2013.12 or later to use the following cross-probing commands:

- `get_cross_probing_info`

The command returns cross-probing data for specified cells or ports as a string, specifying where the cell or port was created and which file and line number the object came from. Some objects in the design might have been merged, resulting in multiple source locations that are returned.

The results of each object are separated by a pair of braces between each source position and between each object. For example, if you specify two objects, obj1 and obj2,

where obj1 has one source position and obj2 has two source positions, the tool returns the results in the following format:

```
{ {obj1_file1:obj1_line1} } { {obj2_file1:obj2_line1}
{obj2_file2:obj2_line2} }
```

To prevent duplicate file:linenumber combinations in the results, use the `-unique_source` option with the `get_cross_probing_info` command. In this case, the tool returns the results in the following format:

```
obj1_file1:obj1_line1  obj2_file1:obj2_line1  obj2_file2:obj2_line2
```

The following example shows results from the `get_cross_probing_info` command with the `-unique_source` option:

```
prompt> get_cross_probing_info [get_cells -hierarchical] \
          -unique_source
/usr/joe/my_design/m.v:21 /usr/joe/my_design/m.v:22 /usr/joe/
my_design/m.v:28 /usr/joe/my_design/m.v:12
/usr/joe/my_design/m.v:13 /usr/joe/my_design/m.v:14 /usr/joe/
my_design/m.v:4
```

- `cross_probing_filter`

The command allows you to filter a collection by specifying a source file and line number in the following format:

```
"file_pattern[:line_pattern]"
```

The following example shows results from the `cross_probing_filter` command, returning cells that originated in a source file ending in m.v on line 21:

```
prompt> cross_probing_filter [get_cells -hierarchical] \
          -source "*m.v:21"
{mid1/bot1/c1 mid1/bot2/c1}
```

Objects in the design that were merged, resulting in multiple source locations, are returned in the collection if any of the source positions match any of the specified source patterns.

- `report_cross_probing`

The command returns cross-probing data for specified cells or ports in a report. If the object was generated by the tool, the file and line number are not applicable and are represented by a hyphen (-). In this case, the report specifies whether the origin of the object comes from UPF, the DFT engine, or another source. Objects that were merged are displayed in the report multiple times, once for each source location.

The tool reports the following information for each specified object:

- Type: The type of design object, either cell or port.

- Reference: For cells, this is the library reference cell. For ports, it is one of the following: IN, OUT, INOUT.
- Filename: The base name of the file. If this information is not available, the tool displays a hyphen (-).
- Line: The line number associated with the file. If this information is not available, the tool displays a hyphen (-).
- Origin: The origin associated with the object. For example, this value can be rtl, datapath, upf, clock_gating, self_gating, or dft. If this information is not available, the tool displays a hyphen (-).
- Object: The object name. If an object has multiple source positions, the object name is indented for lines other than the first source line.

You can also display the original RTL hierarchical name to help identify where the cell originated by using the `-original_hierarchy` option. When you specify this option with the `report_cross_probing` command, the Object column in the report is displayed as Object/Original Hierarchy. For each object, the first row in this column shows the object name, and the second row in the column shows the object's original hierarchy.

Repeating RTL Cross-Probing

Each time the tool reads an RTL file for cross-probing, it stores the RTL line numbers and file paths in the database. If the RTL file was moved to a different location after the cross-probing, repeating the same cross-probing sequence no longer reflects the previous line numbers. As a result, the tool cannot locate the file and issues an error message.

To repeat a cross-probing sequence or reference previous cross-probing data, you must first find the status of the files that were cross-probed. You can obtain the status by running the following commands:

- `report_cross_probing_files`

The command reports the status of the cross-probed files, including OK, Missing, Modified, and No Permissions.

- `update_cross_probing_files`

The command updates the locations of the files that were cross-probed and lists the previous and current paths of the files. The command does not update the cross-probed files if they have been modified.

RTL Cross-Probing Report Example

The following example shows a command-line RTL cross-probing report with the original hierarchy information of the objects included:

```
prompt> report_cross_probing [get_cells U1*] -original_hierarchy
*****
Report      : cross_probing_files
Design     : data_switch_512
Version: ...
Date       : ...
*****
Type        Reference      Filename   Line   Origin    Object/Original Hierarchy
-----
cell        HS65_STF_AOI   test1.v    21536  rtl       U129
                           I_CORE/I_SUB1
cell        HS65_STF_AND2  test2.v    21548  rtl       U104
                           I_CORE/I_SUB2
...
...
```


25

Verifying Functional Equivalence

After optimization, you can use an equivalence checking tool to verify that your gate-level netlist is functionally equivalent to your RTL. This verification step ensures that the synthesis process or manual design changes did not introduce functional errors. You can use the Synopsys Formality tool or a third-party formal verification tool to perform functional equivalence checking, as described in the following topics:

- [Setting Up SVF Generation](#)
- [The Formality Tool](#)
- [Adjusting Optimization for Successful Verification](#)
- [Using Third-Party Formal Verification Tools](#)

Setting Up SVF Generation

The Design Compiler tool records setup information to facilitate alignment of compare points during matching in formal verification tools.

To record a setup information file for

- The Formality tool, use the `set_svf` command
- Other formal verification tools, use the `set_vsdc` command

To stop running the `set_svf` command, use the `set_svf -off` command or exit from the `dc_shell` prompt.

When you use the `set_svf` command, the Design Compiler tool records the following information in the `.svf` file:

- Name changes that occur during synthesis when you use commands such as `group`, `ungroup`, `uniquify`, and `change_names`
- Operations performed by the `compile` command
- UPF information
- Other relevant operations

To overwrite the existing SVF file, use the following command:

```
prompt> set_svf my_design.svf
```

To append the information to the existing SVF file, use the following command:

```
prompt> set_svf -append xyz.svf
```

The guide_hier_map (GHM) Flow

The `guide_hier_map` (GHM) flow adds `guide_hier_map` guidance to the SVF file for capturing all logical hierarchy transformations, including multi-level elaborations, parameterized elaborations, and so on.

To use the `guide_hier_map` (GHM) flow, create your verification guidance as follows:

1. (Optional) Specify the file name to use for your Formality automated setup file:

```
dc_shell> set_svf ./guidance/top.svf
```

By default, the tool creates a file named `default.svf` in the current directory.

2. Set the following variable before reading or analyzing any RTL files:

```
dc_shell> set_app_var hdlin_enable_hier_map true
```

This tracks all logical hierarchy transformations performed by the tool.

3. Execute all analyze, read, and elaborate commands.
4. Use the following command after reading or elaborating all RTL and linking the top-level design:

```
dc_shell> set_verification_top
```

This writes `guide_hier_map` guidance directives into the verification guidance, which provides an accurate reference for subsequent hierarchical transformations and name changes.

For example,

```
# specify SVF output file
set_svf ./guidance/top.svf

# read design files
set_app_var hdlin_enable_hier_map true
analyze -format sverilog {sub.sv top.sv}
elaborate top
current_design top
link

# issue this command after reading last RTL file
# (and before other commands that can modify hierarchy)
set_verification_top

# apply SDC, UPF, hierarchy manipulation, perform synthesis...
```

Note that the tool issues error messages if you forget to set the variable or forget to run the `set_verification_top` command after setting the variable.

You can create verification guidance without setting the `hdlin_enable_hier_map` variable or issuing the `set_verification_top` command. However, the resulting guidance might not handle complex logical hierarchy transformations. This flow is provided for backward compatibility only.

The Formality Tool

The Formality tool is an equivalence checking tool that can verify that your gate-level netlist is functionally equivalent to your RTL. This verification step ensures that the synthesis process or manual design changes did not introduce functional errors.

Formality uses formal techniques to prove or disprove the functional equivalence of two designs. It performs RTL-to-RTL, RTL-to-gate, and gate-to-gate verifications. Functional equivalence checking does not take into account timing; it is a static verification process.

Verification Guidance

By default, Design Compiler automatically creates a Formality automated setup file, which is also known as *verification guidance*, in your working directory. This file has an .svf extension and is named default.svf. The automated setup file provides a method for automatically conveying setup information to Formality. It alleviates the need to enter setup information manually, a task that can be time-consuming and error-prone.

The automated setup file is a binary file. When Formality reads this binary data, it automatically converts it to ASCII and writes it to a text file. This gives you an opportunity to review the setup information before you use it in the verification process.

The automated setup file can contain the following information:

- The settings of certain Tcl variables that affect how the names of design objects in the netlist are created, such as the `bus_naming_style` and `template_*` variables
- Optimizations that occur during RTL elaboration
- Name changes resulting from operations, such as `ungroup`, `group`, `uniquify`, `rename_design`, or the automatic uniquify process in `compile` with combinations of `group` and `ungroup` (these operations can change the names of design objects such as registers, black boxes, and top-level ports)
- Phase inversion and constant propagation optimizations performed during sequential mapping
- Datapath transformations, including information about multiplier architectures chosen during synthesis
- Retiming optimizations
- Information about FSM extraction

Additionally, the automated setup file records implicit ungroup operations. Implicit ungroup operations can occur in the following situations:

- During automatic ungrouping with the `compile_ultra` command, the `compile_ungroup -all` command, or the `compile -auto_ungroup` command
- If a design has an `ungroup` attribute set on it
- When DesignWare auto-ungroups DesignWare parts
- When certain user hierarchies are auto-ungrouped for datapath optimization

You can use the Design Compiler `set_svf` command to control the name of the automated setup file or to save it to a location other than your current working directory. To disable the generation of the .svf file, use the `-off` option. If you use the `set_svf` command, you must do so at the beginning of the synthesis process, before you read your design files.

See Also

- The *Formality User Guide*

Adjusting Optimization for Successful Verification

When Formality cannot complete verification due to issues, such as design complexity, it is considered a failing verification, or a *hard verification*. Datapath intensive designs or designs containing complex design components, such as parity generators, XOR trees, or very large cones of logic, can cause Formality to issue hard verifications or inconclusive verifications.

Design Compiler adjusts some of the optimizations to help reduce inconclusive and hard verifications, as discussed in the following sections. The first method, using the `set_verification_priority` command, is the recommended method for resolving inconclusive and hard verifications because it targets specific blocks or operators while minimizing QoR impact.

- [Using the `set_verification_priority` Command](#)
- [Using Single-Pass Verification](#)

Using the `set_verification_priority` Command

The `set_verification_priority` command sets the `verification_priority` attribute on the specified objects. During the `compile_ultra` command run, the optimizations of the design with the `verification_priority` attribute are adjusted depending on the verification priority level so that the potential for hard verification is reduced. The `set_verification_priority` command is available only in DC Ultra.

When you have inconclusive verifications in Formality, you can use the `analyze_points` command to examine the hard points and determine the possible cause of the hard verifications. The `analyze_points` command runs analysis on the most recent failed or hard verification. The `analyze_points` command generates a recommendation as to which blocks or operators the `set_verification_priority` command can be used with in Design Compiler to reduce hard verifications.

To adjust the optimizations to reduce hard verifications,

1. Run Design Compiler.

2. Run Formality.
3. Analyze the Formality verification reports.

The `analyze_points` command runs analysis on the most recent failed or hard verification. The command generates a recommendation regarding which blocks the `set_verification_priority` command can be used with in Design Compiler to reduce hard verifications.

The following example shows the output of the `analyze_points` command in Formality:

Try adding the following command(s) to your Design Compiler script right before the first `compile_ultra` command:

```
current_design test  
set_verification_priority [ get_cells { add_21 mult_21 mult_30 sub_30 } ]
```

4. Rerun Design Compiler, as follows:

- a. Specify the `set_verification_priority` command with the problem blocks that were identified by the `analyze_points` command in Formality. Add the commands that were suggested by Formality to your Design Compiler scripts.
- b. Run the `compile_ultra` command.

For more information about the `analyze_points` command, see the man page in Formality and the *Formality User Guide*. For more information about the `set_verification_priority` command, see the man page in Design Compiler.

Using Single-Pass Verification

Design Compiler single-pass verification functionality adjusts optimization so that formal verification compatibility is prioritized over QoR. This is helpful if you do not have tight QoR constraints in your design, you do not have register retiming, and you want to pass formal verification with as little effort as possible. You can enable the single-pass verification flow by setting the `simplified_verification_mode` variable to `true`. The variable is set to `false` by default.

By default, designs with the `optimize_registers` attribute and DW_div_pipe DesignWare components are not retimed because they can adversely affect verification success. However, optimizing designs with the `optimize_registers` attribute and DW_div_pipe DesignWare components without retiming them can have a large impact on QoR and runtime. To enable the retiming of these designs, set the `simplified_verification_mode_allow_retimming` variable to `true` when you enable the `simplified_verification_mode` variable.

Using Third-Party Formal Verification Tools

To record setup information for formal verification tools other than Formality, use the `set_vsdc` command. The command records setup information in V-SDC format for efficient compare point matching in third-party formal verification tools. The V-SDC format is a subset of the automated setup file used by Formality. The V-SDC file is written in plain text, whereas the automated setup file is encrypted and compressed.

The `set_vsdc` command records the following operations:

- Name changes resulting from operations such as `ungroup`, `group`, or `uniquify`, or the automatic uniquify process in `compile` with combinations of `group` and `ungroup`. These operations can change the names of design objects such as registers, black boxes, and top-level ports
- Operations performed by the `compile` command that result in register optimizations.

A

Design Example

Optimizing a design can involve using different compile strategies for different levels and components in the design. This appendix shows a design example that uses several compile strategies. Earlier chapters provide detailed descriptions of how to implement each compile strategy. Note that the design example used in this appendix does not represent a real-life application.

This appendix includes the following sections:

- [Design Description](#)
- [Setup File](#)
- [Default Constraints File](#)
- [Read Script](#)
- [Compile Scripts](#)

You can access the files described in these sections at `$SYNOPSIS/doc/syn/guidelines`, where `$SYNOPSIS` is the path to the installation directory.

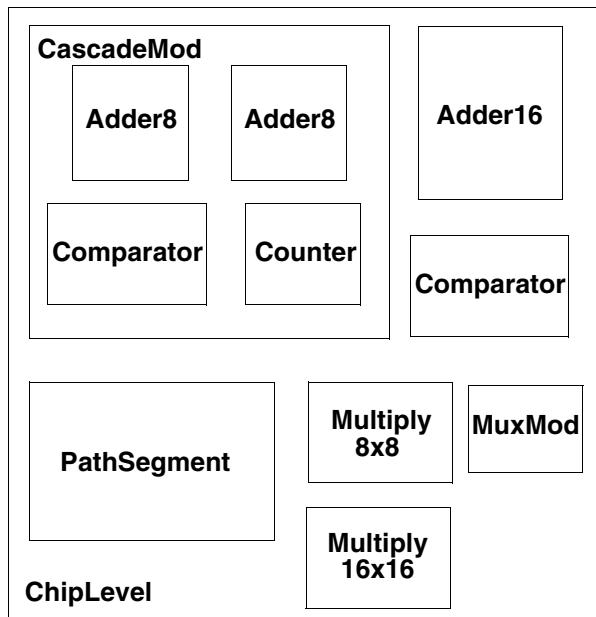
Design Description

The design example shows how you can constrain designs by using a subset of the commonly used dc_shell commands and how you can use scripts to implement various compile strategies.

The design uses synchronous RTL and combinational logic with clocked D flip-flops.

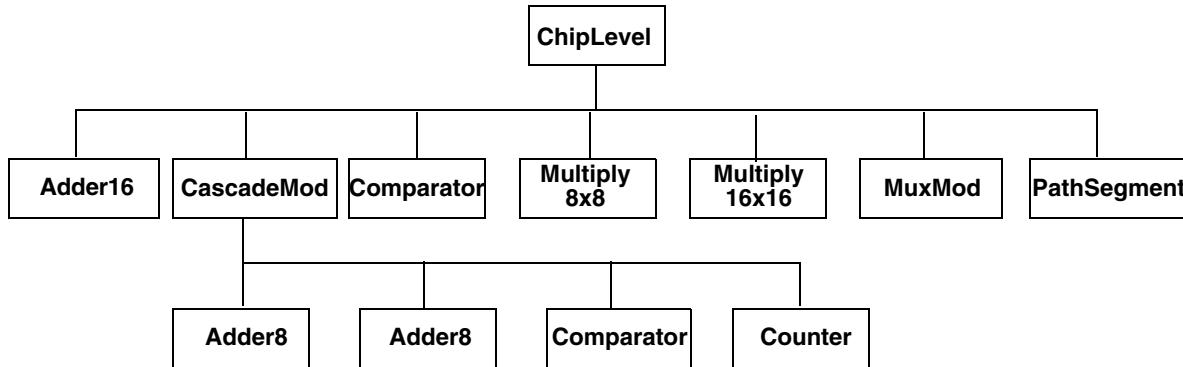
[Figure A-1](#) shows the block diagram for the design example. The design contains seven modules at the top level: Adder16, CascadeMod, Comparator, Multiply8x8, Multiply16x16, MuxMod, and PathSegment.

Figure A-1 Block Diagram for the Design Example



[Figure A-2](#) shows the hierarchy for the design example.

Figure A-2 Hierarchy for the Design Example



The top-level modules and the compilation strategies for optimizing them are

Adder16

Uses registered outputs to make constraining easier. Because the endpoints are the data pins of the registers, you do not need to set output delays on the output ports.

CascadeMod

Uses a hierarchical compile strategy. The compile script for this design sets the constraints at the top level (of CascadeMod) before compilation.

The CascadeMod design instantiates the Adder8 design twice. The script uses the compile-once-don't-touch method for the Comparator module.

Comparator

Is a combinational block. The compile script for this design uses the virtual clock concept to show the use of virtual clocks in a design.

The ChipLevel design instantiates Comparator twice. The compile script (for CascadeMod) uses the compile-once-don't-touch method to resolve the multiple instances.

The compile script specifies wire load model and mode instead of using automatic wire load selection.

Multiply8x8

Shows the basic timing and area constraints used for optimizing a design.

Multiply16x16

Ungroups DesignWare parts before compilation. Ungrouping your hierarchical module might help achieve better synthesis results. The compile script for this module defines a two-cycle path at the primary ports of the module.

MuxMod

Is a combinational block. The script for this design uses the virtual clock concept.

PathSegment

Uses path segmentation within a module. The script uses the `set_multicycle_path` command for a two-cycle path within the module and the `group` command to create a new level of hierarchy.

[Example A-1](#) through [Example A-11](#) provide the Verilog source code for the ChipLevel design.

Example A-1 ChipLevel.v

```
/* Date: May 11, 1995 */
/* Example Circuit for Baseline Methodology for Synthesis */
/* Design does not show any real-life application but rather
   it is used to illustrate the commands used in the Baseline
   Methodology */

module ChipLevel (data16_a, data16_b, data16_c, data16_d, clk, cin, din_a,
                  din_b, sel, rst, start, mux_out, cout1, cout2, s1, s2, op,
                  comp_out1, comp_out2, m32_out, regout);

  input [15:0] data16_a, data16_b, data16_c, data16_d;
  input [7:0] din_a, din_b;
  input [1:0] sel;
  input clk, cin, rst, start;
  input s1, s2, op;
  output [15:0] mux_out, regout;
  output [31:0] m32_out;
  output cout1, cout2, comp_out1, comp_out2;

  wire [15:0] ad16_sout, ad8_sout, m16_out, cnt;

Adder16 u1 (.ain(data16_a), .bin(data16_b), .cin(cin), .cout(cout1),
             .sout(ad16_sout), .clk(clk));

CascadeMod u2 (.data1(data16_a), .data2(data16_b), .cin(cin), .s(ad8_sout),
               .cout(cout2), .clk(clk), .comp_out(comp_out1), .cnt(cnt),
               .rst(rst), .start(start) );

Comparator u3 (.ain(ad16_sout), .bin(ad8_sout), .cp_out(comp_out2));

Multiply8x8 u4 (.op1(din_a), .op2(din_b), .res(m16_out), .clk(clk));

Multiply16x16 u5 (.op1(data16_a), .op2(data16_b), .res(m32_out), .clk(clk));

MuxMod u6 (.Y_IN(mux_out), .MUX_CNT(sel), .D(ad16_sout), .R(ad8_sout),
```

```

.F(m16_out), .UPC(cnt));

PathSegment u7 (.R1(data16_a), .R2(data16_b), .R3(data16_c), .R4(data16_d),
               .S2(s2), .S1(s1), .OP(op), .REGOUT(regout), .clk(clk));
endmodule

```

Example A-2 Adder16.v

```

module Adder16 (ain, bin, cin, sout, cout, clk);
/* 16-Bit Adder Module */
output [15:0] sout;
output cout;
input [15:0] ain, bin;
input cin, clk;

wire [15:0] sout_tmp, ain, bin;
wire cout_tmp;
reg [15:0] sout, ain_tmp, bin_tmp;
reg cout, cin_tmp;

always @ (posedge clk) begin
    cout = cout_tmp;
    sout = sout_tmp;
    ain_tmp = ain;
    bin_tmp = bin;
    cin_tmp = cin;
end
assign {cout_tmp,sout_tmp} = ain_tmp + bin_tmp + cin_tmp;
endmodule

```

Example A-3 CascadeMod.v

```

module CascadeMod (data1, data2, s, clk, cin, cout, comp_out, cnt, rst, start);
input [15:0] data1, data2;
output [15:0] s, cnt;
input clk, cin, rst, start;
output cout, comp_out;
wire co;

Adder8 u10 (.ain(data1[7:0]), .bin(data2[7:0]), .cin(cin), .clk(clk),
.sout(s[7:0]), .cout(co));
Adder8 u11 (.ain(data1[15:8]), .bin(data2[15:8]), .cin(co), .clk(clk),
.sout(s[15:8]), .cout(cout));
Comparator u12 (.ain(s), .bin(cnt), .cp_out(comp_out));

Counter u13 (.count(cnt), .start(start), .clk(clk), .rst(rst));
endmodule

```

Example A-4 Adder8.v

```

module Adder8 (ain, bin, cin, sout, cout, clk);
/* 8-Bit Adder Module */
output [7:0] sout;
output cout;
input [7:0] ain, bin;
input cin, clk;

wire [7:0] sout_tmp, ain, bin;
wire cout_tmp;
reg [7:0] sout, ain_tmp, bin_tmp;
reg cout, cin_tmp;

always @ (posedge clk) begin
    cout = cout_tmp;
    sout = sout_tmp;
    ain_tmp = ain;
    bin_tmp = bin;
    cin_tmp = cin;
end
assign {cout_tmp, sout_tmp} = ain_tmp + bin_tmp + cin_tmp;
endmodule

```

Example A-5 Counter.v

```

module Counter (count, start, clk, rst);
/* Counter module */
input clk;
input rst;
input start;
output [15:0] count;

wire clk;
reg [15:0] count_N;
reg [15:0] count;

always @ (posedge clk or posedge rst)
begin : counter_S
    if (rst) begin
        count = 0; // reset logic for the block
    end
    else begin
        count = count_N; // set specified registers of the block
    end
end

always @ (count or start)
begin : counter_C
    count_N = count; // initialize outputs of the block
    if (start) count_N = 1; // user specified logic for the block
    else count_N = count + 1;
end
endmodule

```

Example A-6 Comparator.v

```
module Comparator (cp_out, ain, bin);
/* Comparator for 2 integer values */
output cp_out;
input [15:0] ain, bin;
assign cp_out = ain < bin;
endmodule
```

Example A-7 Multiply8x8.v

```
module Multiply8x8 (op1, op2, res, clk);
/* 8-Bit multiplier */
input [7:0] op1, op2;
output [15:0] res;
input clk;

wire [15:0] res_tmp;
reg [15:0] res;

always @(posedge clk) begin
    res = res_tmp;
end
assign res_tmp = op1 * op2;
endmodule
```

Example A-8 Multiply16x16.v

```
module Multiply16x16 (op1, op2, res, clk);
/* 16-Bit multiplier */
input [15:0] op1, op2;
output [31:0] res;
input clk;

wire [31:0] res_tmp;
reg [31:0] res;

always @(posedge clk) begin
    res = res_tmp;
end
assign res_tmp = op1 * op2;
endmodule
```

Example A-9 def_macro.v

```
`define DATA 2'b00
`define REG 2'b01
`define STACKIN 2'b10
`define UPCOUT 2'b11
```

Example A-10 MuxMod.v

```

module MuxMod (Y_IN, MUX_CNT, D, R, F, UPC);
`include "def_macro.v"
  output [15:0] Y_IN;
  input [1:0] MUX_CNT;
  input [15:0] D, F, R, UPC;

  reg [15:0] Y_IN;

  always @ ( MUX_CNT or D or R or F or UPC ) begin
    case ( MUX_CNT )
      `DATA :
        Y_IN = D ;
      `REG :
        Y_IN = R ;
      `STACKIN :
        Y_IN = F ;
      `UPCOUT :
        Y_IN = UPC;
    endcase
  end
endmodule

```

Example A-11 PathSegment.v

```

module PathSegment (R1, R2, R3, R4, S2, S1, OP, REGOUT, clk);
/* Example for path segmentation */
input [15:0] R1, R2, R3, R4;
input S2, S1, clk;
input OP;
output [15:0] REGOUT;

reg [15:0] ADATA, BDATA;
reg [15:0] REGOUT;
reg MODE;

wire [15:0] product ;

always @(posedge clk)
begin : selector_block
  case(S1)
    1'b0: ADATA <= R1;
    1'b1: ADATA <= R2;
    default: ADATA <= 16'bx;
  endcase
  case(S2)
    1'b0: BDATA <= R3;
    1'b1: BDATA <= R4;
    default: BDATA <= 16'bx;
  endcase
end
/* Only Lower Byte gets multiplied */

```

```
// instantiate DW02_mult
DW02_mult #(8,8) U100 (.A(ADATA[7:0]), .B(BDATA[7:0]), .TC(1'b0),
.PRODUCT(product));

always @ (posedge clk)
begin : alu_block
    case (OP)
        1'b0 : begin
            REGOUT <= ADATA + BDATA;
        end
        1'b1 : begin
            REGOUT <= product;
        end
        default : REGOUT <= 16'bx;
    endcase
end

endmodule
```

Setup File

When running the design example, copy the project-specific setup file in [Example A-12](#) to your project working directory. This setup file is written in the Tcl subset and can be used in the Tcl command language. For more information about the Tcl subset, see *Using Tcl With Synopsys Tools*.

For details on the synthesis setup files, see [The Setup Files](#).

Example A-12 .synopsys_dc.setup File

```
# Define the target logic library, symbol library,
# and link libraries
set target_library lsi_10k.db
set symbol_library lsi_10k.sdb
set link_library [concat $target_library "*"]
set search_path [concat $search_path ./src]
set designer "Your Name"
set company "Synopsys, Inc."
# Define path directories for file locations
set source_path "./src/"
set script_path "./scr/"
set log_path "./log/"
set ddc_path "./ddc/"
set db_path "./db/"
set netlist_path "./netlist/"
```

Default Constraints File

The file shown in [Example A-13](#) defines the default constraints for the design. In the scripts that follow, Design Compiler reads this file first for each module. If the script for a module contains additional constraints or constraint values different from those defined in the default constraints file, Design Compiler uses the module-specific constraints.

Example A-13 defaults.con

```
# Define system clock period
set clk_period 20

# Create real clock if clock port is found
if {[sizeof_collection [get_ports clk]] > 0} {
    set clk_name clk
    create_clock -period $clk_period clk
}

# Create virtual clock if clock port is not found
if {[sizeof_collection [get_ports clk]] == 0} {
    set clk_name vclk
    create_clock -period $clk_period -name vclk
}

# Apply default drive strengths and typical loads
# for I/O ports
set_load 1.5 [all_outputs]
set_driving_cell -lib_cell IV [all_inputs]

# If real clock, set infinite drive strength
if {[sizeof_collection [get_ports clk]] > 0} {
    set_drive 0 clk
}

# Apply default timing constraints for modules
set_input_delay 1.2 [all_inputs] -clock $clk_name
set_output_delay 1.5 [all_outputs] -clock $clk_name
set_clock_uncertainty -setup 0.45 $clk_name

# Set operating conditions
set_operating_conditions WCCOM

# Turn on auto wire load selection
# (library must support this feature)
set auto_wire_load_selection true
```

Read Script

[Example A-15](#) provides the Tcl script used to read in the ChipLevel design.

The `read.tcl` script reads design information from the specified Verilog files into memory.

Example A-14 read.tcl

```
read_file -format verilog ChipLevel.v
read_file -format verilog Adder16.v
read_file -format verilog CascadeMod.v
read_file -format verilog Adder8.v
read_file -format verilog Counter.v
read_file -format verilog Comparator.v
read_file -format verilog Multiply8x8.v
read_file -format verilog Multiply16x16.v
read_file -format verilog MuxMod.v
read_file -format verilog PathSegment.v
```

Compile Scripts

[Example A-15](#) through [Example A-26](#) provide the Tcl scripts used to compile the ChipLevel design.

The compile script for each module is named for that module to ease recognition. The initial Tcl script files have the `.tcl` suffix. Scripts generated by the `write_script` command have the `.wtcl` suffix.

Example A-15 run.tcl

```
# Initial compile with estimated constraints
source "${script_path}initial_compile.tcl"

current_design ChipLevel
if {[shell_is_in_xg_mode]==0} {
    write -hier -o "${db_path}ChipLevel_init.db"
} else {
    write -format ddc -hier -o "${ddc_path}ChipLevel_init.ddc"
}

# Characterize and write_script for all modules
source "${script_path}characterize.tcl"

# Recompile all modules using write_script constraints
remove_design -all
source "${script_path}recompile.tcl"

current_design ChipLevel
if {[shell_is_in_xg_mode]==0} {
    write -hier -out "${db_path}ChipLevel_final.db"
```

```

} else {
write -format ddc -hier -out "${ddc_path}ChipLevel_final.ddc"
}

```

Example A-16 initial_compile.tcl

```

# Initial compile with estimated constraints
source "${script_path}read.tcl"

current_design ChipLevel
source "${script_path}defaults.con"

source "${script_path}adder16.tcl"
source "${script_path}cascademod.tcl"
source "${script_path}comp16.tcl"
source "${script_path}mult8.tcl"
source "${script_path}mult16.tcl"
source "${script_path}muxmod.tcl"
source "${script_path}pathseg.tcl"

```

Example A-17 adder16.tcl

```

# Script file for constraining Adder16
set rpt_file "adder16.rpt"
set design "adder16"

current_design Adder16
source "${script_path}defaults.con"

# Define design environment
set_load 2.2 sout
set_load 1.5 cout
set_driving_cell -lib_cell FD1 [all_inputs]
set_drive 0 $clk_name

# Define design constraints
set_input_delay 1.35 -clock $clk_name {ain bin}
set_input_delay 3.5 -clock $clk_name cin
set_max_area 0

compile

if {[shell_is_in_xg_mode]==0} {
write -hier -o "${db_path}${design}.db"
} else {
write -format ddc -hier -o "${ddc_path}${design}.ddc"
}

source "${script_path}report.tcl"

```

Example A-18 cascademod.tcl

```

# Script file for constraining CascadeMod
# Constraints are set at this level and then a
# hierarchical compile approach is used

set rpt_file "cascademod.rpt"
set design "cascademod"

current_design CascadeMod
source "${script_path}defaults.con"

# Define design environment
set_load 2.5 [all_outputs]
set_driving_cell -lib_cell FD1 [all_inputs]
set_drive 0 $clk_name

# Define design constraints
set_input_delay 1.35 -clock $clk_name {data1 data2}
set_input_delay 3.5 -clock $clk_name cin
set_input_delay 4.5 -clock $clk_name {rst start}
set_output_delay 5.5 -clock $clk_name comp_out
set_max_area 0

# Use compile-once, dont_touch approach for Comparator
set_dont_touch u12

compile

if {[shell_is_in_xg_mode]==0} {
    write -hier -o "${db_path}${design}.db"
} else {
    write -format ddc -hier -o "${ddc_path}${design}.ddc"
}

source "${script_path}report.tcl"

```

Example A-19 comp16.tcl

```

# Script file for constraining Comparator
set rpt_file "comp16.rpt"
set design "comp16"

current_design Comparator
source "${script_path}defaults.con"

# Define design environment
set_load 2.5 cp_out
set_driving_cell -lib_cell FD1 [all_inputs]

# Override auto wire load selection
set_wire_load_model -name "05x05"
set_wire_load_mode enclosed

```

```
# Define design constraints
set_input_delay 1.35 -clock $clk_name {ain bin}
set_output_delay 5.1 -clock $clk_name {cp_out}
set_max_area 0

compile

if {[shell_is_in_xg_mode]==0} {
    write -hier -o "${db_path}${design}.db"
} else {
    write -format ddc -hier -o "${ddc_path}${design}.ddc"
}

source "${script_path}report.tcl"
```

Example A-20 mult8.tcl

```
# Script file for constraining Multiply8x8
set rpt_file "mult8.rpt"
set design "mult8"

current_design Multiply8x8
source "${script_path}defaults.con"

# Define design environment
set_load 2.2 res
set_driving_cell -lib_cell FD1P [all_inputs]
set_drive 0 $clk_name

# Define design constraints
set_input_delay 1.35 -clock $clk_name {op1 op2}
set_max_area 0

compile

if {[shell_is_in_xg_mode]==0} {
    write -hier -o "${db_path}${design}.db"
} else {
    write -format ddc -hier -o "${ddc_path}${design}.ddc"
}

source "${script_path}report.tcl"
```

Example A-21 mult16.tcl

```
# Script file for constraining Multiply16x16
set rpt_file "mult16.rpt"
set design "mult16"

current_design Multiply16x16
source "${script_path}defaults.con"

# Define design environment
set_load 2.2 res
set_driving_cell -lib_cell FD1 [all_inputs]
set_drive 0 $clk_name

# Define design constraints
set_input_delay 1.35 -clock $clk_name {op1 op2}
set_max_area 0

# Define multicycle path for multiplier
set_multicycle_path 2 -from [all_inputs] \
    -to [all_registers -data_pins -edge_triggered]

# Ungroup DesignWare parts
set designware_cells [get_cells \
    -filter "@is_oper==true"]
if {[sizeof_collection $designware_cells] > 0} {
    set_ungroup $designware_cells true
}

compile

if {[shell_is_in_xg_mode]==0} {
    write -hier -o "${db_path}${design}.db"
} else {
    write -format ddc -hier -o "${ddc_path}${design}.ddc"
}

source "${script_path}report.tcl"
report_timing_requirements -ignore \
    >> "${log_path}${rpt_file}"
```

Example A-22 muxmod.tcl

```

# Script file for constraining MuxMod
set rpt_file "muxmod.rpt"
set design "muxmod"

current_design MuxMod
source "${script_path}defaults.con"

# Define design environment
set_load 2.2 Y_IN
set_driving_cell -lib_cell FD1 [all_inputs]

# Define design constraints
set_input_delay 1.35 -clock $clk_name {D R F UPC}
set_input_delay 2.35 -clock $clk_name MUX_CNT
set_output_delay 5.1 -clock $clk_name {Y_IN}
set_max_area 0

compile

if {[shell_is_in_xg_mode]==0} {
    write -hier -o "${db_path}${design}.db"
} else {
    write -format ddc -hier -o "${ddc_path}${design}.ddc"
}

source "${script_path}report.tcl"

```

Example A-23 pathseg.tcl

```

# Script file for constraining path_segment
set rpt_file "pathseg.rpt"
set design "pathseg"

current_design PathSegment
source "${script_path}defaults.con"

# Define design environment
set_load 2.5 [all_outputs]
set_driving_cell -lib_cell FD1 [all_inputs]
set_drive 0 $clk_name

# Define design rules
set_max_fanout 6 {S1 S2}

# Define design constraints
set_input_delay 2.2 -clock $clk_name {R1 R2}
set_input_delay 2.2 -clock $clk_name {R3 R4}
set_input_delay 5 -clock $clk_name {S2 S1 OP}
set_max_area 0

# Perform path segmentation for multiplier
group -design mult -cell mult U100

```

```

set_input_delay 10 -clock $clk_name mult/product*
set_output_delay 5 -clock $clk_name mult/product*
set_multicycle_path 2 -to mult/product*

compile

if {[shell_is_in_xg_mode]==0} {
write -hier -o "${db_path}${design}.db"
} else {
write -format ddc -hier -o "${ddc_path}${design}.ddc"

source "${script_path}report.tcl"
report_timing_requirements -ignore \
    >> "${log_path}${rpt_file}"
}

```

Example A-24 *characterize.tcl*

```

# Characterize and write_script for all modules
current_design ChipLevel
characterize u1
current_design Adder16
write_script > "${script_path}adder16.wtcl"

current_design ChipLevel
characterize u2
current_design CascadeMod
write_script -format tcl"${script_path}cascademod.wtcl"

current_design ChipLevel
characterize u3
current_design Comparator
write_script -format tcl > "${script_path}comp16.wtcl"

current_design ChipLevel
characterize u4
current_design Multiply8x8
write_script -format tcl > "${script_path}mult8.wtcl"

current_design ChipLevel
characterize u5
current_design Multiply16x16
write_script -format tcl > "${script_path}mult16.wtcl"

current_design ChipLevel
characterize u6
current_design MuxMod
write_script -format tcl > "${script_path}muxmod.wtcl"

current_design ChipLevel
characterize u7
current_design PathSegment

```

```

echo "current_design PathSegment" > \
"${script_path}pathseg.wtcl"

echo "group -design mult -cell mult U100" >> \
"${script_path}pathseg.wtcl"
write_script -format tcl >> "${script_path}pathseg.wtcl"

```

Example A-25 recompile.tcl

```

source "${script_path}read.tcl"

current_design ChipLevel
source "${script_path}defaults.con"

source "${script_path}adder16.wtcl"
compile
if {[shell_is_in_xg_mode]==0} {
  write -hier -o "${db_path}adder16_wtcl.db"
} else {
  write -format ddc -hier -o "${ddc_path}adder16_wtcl.ddc"
  set rpt_file adder16_wtcl.rpt
  source "${script_path}report.tcl"

  source "${script_path}cascademod.wtcl"
  dont_touch u12
  compile
  if {[shell_is_in_xg_mode]==0} {
    write -hier -o "${db_path}cascademod_wtcl.db"
  } else {
    write -format ddc -hier -o "${ddc_path}cascademod_wtcl.ddc"
    set rpt_file cascade_wtcl.rpt
    source "${script_path}report.tcl"

  source "${script_path}comp16.wtcl"
  compile
  if {[shell_is_in_xg_mode]==0} {
    write -hier -o "${db_path}comp16_wtcl.db"
  } else {
    write -format ddc -hier -o "${ddc_path}comp16_wtcl.ddc"
    set rpt_file comp16_wtcl.rpt
    source "${script_path}report.tcl"

  source "${script_path}mult8.wtcl"
  compile
  if {[shell_is_in_xg_mode]==0} {
    write -hier -o "${db_path}mult8_wtcl.db"
  } else {
    write -format ddc -hier -o "${ddc_path}mult8_wtcl.ddc"
    set rpt_file mult8_wtcl.rpt
    source "${script_path}report.tcl"

  source "${script_path}mult16.wtcl"

```

```

compile -ungroup_all
if {[shell_is_in_xg_mode]==0} {
write -hier -o "${db_path}mult16_wtcl.db"
} else {
write -format ddc -hier -o "${ddc_path}mult16_wtcl.ddc"
set rpt_file mult16_wtcl.rpt
source "${script_path}report.tcl"
report_timing_requirements -ignore \
    >> "${log_path}${rpt_file}""

source "${script_path}muxmod.wtcl"
compile
if {[shell_is_in_xg_mode]==0} {
write -hier -o "${db_path}muxmod_wtcl.db"
} else {
write -format ddc -hier -o "${ddc_path}muxmod_wtcl.ddc"
set rpt_file muxmod_wtcl.rpt
source "${script_path}report.tcl"

source "${script_path}pathseg.wtcl"
compile
if {[shell_is_in_xg_mode]==0} {
write -hier -o "${db_path}pathseg_wtcl.db"
} else {
write -format ddc -hier -o "${ddc_path}pathseg_wtcl.ddc"
set rpt_file pathseg_wtcl.rpt
source "${script_path}report.tcl"
report_timing_requirements -ignore \
    >> "${log_path}${rpt_file}"
}
}
}

```

Example A-26 report.tcl

```

# This script file creates reports for all modules
set maxpaths 15

check_design > "${log_path}${rpt_file}"
report_area >> "${log_path}${rpt_file}"
report_design >> "${log_path}${rpt_file}"
report_cell >> "${log_path}${rpt_file}"
report_reference >> "${log_path}${rpt_file}"
report_port -verbose >> "${log_path}${rpt_file}"
report_net >> "${log_path}${rpt_file}"
report_compile_options >> "${log_path}${rpt_file}"
report_constraint -all_violators -verbose \
    >> "${log_path}${rpt_file}"
report_timing -path end >> "${log_path}${rpt_file}"
report_timing -max_path $maxpaths \
    >> "${log_path}${rpt_file}"
report_qor >> "${log_path}${rpt_file}"

```


B

Basic Commands

This appendix lists some of the basic dc_shell commands for synthesis and provides a brief description for each command. The commands are grouped in the following sections:

- [Commands for Defining Design Rules](#)
- [Commands for Defining Design Environments](#)
- [Commands for Setting Design Constraints](#)
- [Commands for Analyzing and Resolving Design Problems](#)

Within each section the commands are listed in alphabetical order.

Commands for Defining Design Rules

The commands that define design rules are

`set_max_capacitance`

Sets a maximum capacitance for the nets attached to the specified ports or to all the nets in a design.

`set_max_fanout`

Sets the expected fanout load value for output ports.

`set_max_transition`

Sets a maximum transition time for the nets attached to the specified ports or to all the nets in a design.

`set_min_capacitance`

Sets a minimum capacitance for the nets attached to the specified ports or to all the nets in a design.

Commands for Defining Design Environments

The commands that define the design environment are

`set_drive`

Sets the drive value of input or inout ports. The `set_drive` command is superseded by the `set_driving_cell` command.

`set_driving_cell`

Sets attributes on input or inout ports, specifying that a library cell or library pin drives the ports. This command associates a library pin with an input port so that delay calculators can accurately model the drive capability of an external driver.

`set_fanout_load`

Defines the external fanout load values on output ports.

`set_load`

Defines the external load values on input and output ports and nets.

`set_operating_conditions`

Defines the operating conditions for the current design.

```
set_wire_load_model
```

Sets the wire load model for the current design or for the specified ports. With this command, you can specify the wire load model to use for the external net connected to the output port.

Commands for Setting Design Constraints

The basic commands that set design constraints are

```
create_clock
```

Creates a clock object and defines its waveform in the current design.

```
set_clock_latency, set_clock_uncertainty, set_propagated_clock,  
set_clock_transition
```

Sets clock attributes on clock objects or flip-flop clock pins.

```
set_input_delay
```

Sets input delay on pins or input ports relative to a clock signal.

```
set_max_area
```

Specifies the maximum area for the current design.

```
set_output_delay
```

Sets output delay on pins or output ports relative to a clock signal.

The advanced commands that set design constraints are

```
group_path
```

Groups a set of paths or endpoints for cost function calculation. This command is used to create path groups, to add paths to existing groups, or to change the weight of existing groups.

```
set_false_path
```

Marks paths between specified points as false. This command eliminates the selected paths from timing analysis.

```
set_max_delay
```

Specifies a maximum delay target for selected paths in the current design.

```
set_min_delay
```

Specifies a minimum delay target for selected paths in the current design.

```
set_multicycle_path
```

Allows you to specify the time of a timing path to exceed the time of one clock signal.

Commands for Analyzing and Resolving Design Problems

The commands for analyzing and resolving design problems are

```
all_connected
```

Lists all fanouts on a net.

```
all_registers
```

Lists sequential elements or pins in a design.

```
check_design
```

Checks the internal representation of the current design for consistency and issues error and warning messages as appropriate.

```
check_timing
```

Checks the timing attributes placed on the current design.

```
get_attribute
```

Reports the value of the specified attribute.

```
link
```

Locates the reference for each cell in the design.

```
report_area
```

Provides area information and statistics on the current design.

```
report_attribute
```

Lists the attributes and their values for the selected object. An object can be a cell, net, pin, port, instance, or design.

```
report_cell
```

Lists the cells in the current design and their cell attributes.

```
report_clock
```

Displays clock-related information about the current design.

```
report_constraint
```

Lists the constraints on the current design and their cost, weight, and weighted cost.

`report_delay_calculation`

Reports the details of a delay arc calculation.

`report_design`

Displays the operating conditions, wire load model and mode, timing ranges, internal input and output, and disabled timing arcs defined for the current design.

`report_hierarchy`

Lists the subdesigns of the current design.

`report_net`

Displays net information for the design of the current instance, if set; otherwise, displays net information for the current design.

`report_path_group`

Lists all timing path groups in the current design.

`report_port`

Lists information about ports in the current design.

`report_qor`

Displays information about the quality of results and other statistics for the current design.

`report_resources`

Displays information about the resource implementation.

`report_timing`

Lists timing information for the current design.

`report_timing_requirements`

Lists timing path requirements and related information.

`report_transitive_fanin`

Lists the fanin logic for selected pins, nets, or ports of the current instance.

`report_transitive_fanout`

Lists the fanout logic for selected pins, nets, or ports of the current instance.

C

Predefined Attributes

This appendix lists some of the most commonly used Design Compiler predefined attributes for each object type.

Table C-1 Clock Attributes

Attribute name	Value
<code>clock_fall_transition</code>	<code>float</code>
<code>clock_min_fall_transition</code>	<code>float</code>
<code>clock_min_rise_transition</code>	<code>float</code>
<code>clock_rise_transition</code>	<code>float</code>
<code>dont_touch_network</code>	<code>{true, false}</code>
<code>dont_touch_network_no_propagate</code>	<code>{true, false}</code>
<code>fall_delay</code>	<code>float</code>
<code>fall_min_delay</code>	<code>float</code>
<code>fix_hold</code>	<code>boolean</code>
<code>full_name</code>	<code>string</code>

Table C-1 Clock Attributes (Continued)

Attribute name	Value
max_time_borrow	float
minus_uncertainty	float
name	string
object_class	string
period	float
plus_uncertainty	float
propagated_clock	boolean
rise_delay	float
rise_min_delay	float

Table C-2 Design Attributes

Attribute name	Value
actual_max_net_capacitance	float
actual_min_net_capacitance	float
boundary_optimization	{true, false}
default_flip_flop_type	internally generated string
default_flip_flop_type_exact	library_cell_name
default_latch_type	library_cell_name
design_type	{equation, fsm, pla, netlist}
dont_touch	{true, false}
dont_touch_network	{true, false}
driven_by_logic_one	{true, false}

Table C-2 Design Attributes (Continued)

Attribute name	Value
driven_by_logic_zero	{true, false}
driving_cell_dont_scale	string
driving_cell_fall	string
driving_cell_from_pin_fall	string
driving_cell_from_pin_rise	string
driving_cell_library_fall	string
driving_cell_library_rise	string
driving_cell_multiplier	float
driving_cell_pin_fall	string
driving_cell_pin_rise	string
driving_cell_rise	string
fall_drive	float
fanout_load	float
flatten	{true, false}
flatten_effort	{true, false}
flatten_minimize	{true, false}
flatten_phase	{true, false}
flip_flop_type	internally generated string
flip_flop_type_exact	library_cell_name
is_black_box	{true, false}
is_combinatorial	{true, false}
is_hierarchical	{true, false}

Table C-2 Design Attributes (Continued)

Attribute name	Value
is_mapped	{true, false}
is_sequential	{true, false}
is_test_circuitry	{true, false}
is_unmapped	{true, false}
latch_type	internally generated string
latch_type_exact	library_cell_name
load	float
local_link_library	design_or_lib_file_name
max_capacitance	float
max_fanout	float
max_time_borrow	float
max_transition	float
min_capacitance	float
minus_uncertainty	float
output_not_used	{true, false}
pad_location (XNF only)	string
part (XNF only)	string
plus_uncertainty	float
port_direction	{in, inout, out, unknown}
port_is_pad	{true, false}
ref_name	reference_name
rise_drive	float

Table C-2 Design Attributes (Continued)

Attribute name	Value
structure	{true, false}
ungroup	{true, false}
wired_logic_disable	{true, false}

Table C-3 Library Attributes

Attribute name	Value
default_values	float
k_process_values	float
k_temp_values	float
k_volt_values	float
nom_process	float
nom_temperature	float
nom_voltage	float

Table C-4 Library Cell Attributes

Attribute name	Value
area	float
dont_touch	{true, false}
dont_use	{true, false}
preferred	{true, false}

Table C-5 Net Attributes

Attribute name	Value
ba_net_resistance	float
dont_touch	{true, false}
load	float
subtract_pin_load	{true, false}
wired_and	{true, false}
wired_or	{true, false}

Table C-6 Pin Attributes

Attribute name	Value
disable_timing	{true, false}
max_time_borrow	float
pin_direction	{in, inout, out, unknown}

Table C-7 Reference Attributes

Attribute name	Value
dont_touch	{true, false}
is_black_box	{true, false}
is_combinatorial	{true, false}
is_hierarchical	{true, false}
is_mapped	{true, false}
is_sequential	{true, false}
is_unmapped	{true, false}
ungroup	{true, false}

D

Latch-Based Design Code Examples

For code examples of designs that use various types of latches, see the following topics:

- [SR Latch](#)
- [D Latch](#)
- [D Latch With Asynchronous Reset](#)
- [D Latch With Asynchronous Set and Reset](#)
- [D Latch With Enable \(Avoiding Clock Gating\)](#)
- [D Latch With Enable and Asynchronous Reset](#)
- [D Latch With Enable and Asynchronous Set](#)
- [D Latch With Enable and Asynchronous Set and Reset](#)

SR Latch

The following topics show a code example of an SR latch in VHDL and Verilog and the resulting inference report and synthesized design:

- [VHDL and Verilog Code Examples for SR Latch](#)
 - [Inference Report for an SR Latch](#)
 - [Synthesized Design for an SR Latch](#)
-

VHDL and Verilog Code Examples for SR Latch

To implement an SR latch in either VHDL or Verilog, you must set the `hdlin_report_inferred_modules` variable to `true`, and you must set the following attribute:

```
attribute async_set_reset of RESET, SET : signal is "true";
```

[Example D-1](#) shows the VHDL code that infers an SR latch.

Example D-1 VHDL Code for an SR Latch

```
library IEEE, SYNOPSYS;
use IEEE.std_logic_1164.all;
use SYNOPSYS.attributes.all;

entity SR_LATCH is
    port ( RESET, SET : in std_logic;
           Y : out std_logic );
end SR_LATCH;

architecture BEHAVIORAL of SR_LATCH is
attribute async_set_reset of RESET, SET : signal is "true";
begin
    infer : process ( RESET, SET )
    begin
        if ( RESET = '0' ) then
            y <= '0';
        elsif ( SET = '0' ) then
            y <= '1';
        end if;
    end process infer;
end BEHAVIORAL;
```

[Example D-2](#) shows Verilog code that infers an SR latch.

Example D-2 Verilog Code Example for an SR Latch

```
module SR_LATCH( reset,set, y);
  input reset, set ;
  output y ;
  // synopsys async_set_reset "reset,set"
  reg y ;

  always @(set or reset)
    begin : infer
      if (reset == 0)
        y = 1'b0 ;
      else if (set == 0)
        y = 1'b1 ;
    end
endmodule
```

Inference Report for an SR Latch

[Example D-3](#) shows the inference report generated for an SR latch from the VHDL code shown in [Example D-1](#) or the Verilog code in [Example D-2](#).

Example D-3 Inference Report for an SR Latch

```
Inferred memory devices in process 'infer'
  in routine SR_LATCH line 13 in
  file '/home/sudipto/work/latch_appl/rtl/vhdl/sr_latch.vhdl'.
```

```
=====
|   Register Name |   Type    | Width | Bus | MB | AR | AS | SR | SS | ST |
=====
|     Y_reg      |   Latch   |   1   | -  | -  | Y  | Y  | -  | -  | -  |
=====
Y_reg
-----
  Async-reset: RESET'
  Async-set: SET'
  Async-set and Async-reset ==> Q: 0
```

Synthesized Design for an SR Latch

[Figure D-1](#) shows the synthesized design for the SR-latch-based design resulting from compilation of the code shown in [Example D-1](#) or [Example D-2](#). In this synthesis, LSR0 is an SR latch and both S and R are active-low Inputs. Here is the target library description of the latch for the cell LSR0:

```
latch ("IQ","IQN") {  
    clear      : "R'";  
    preset     : "S'";  
    clear_preset_var1 : L;  
    clear_preset_var2 : L;  
}
```

Figure D-1 Synthesized Design for an SR Latch



D Latch

The following topics show the VHDL code that implements a design that uses a simple D latch and the resulting inference report and synthesized design:

- [VHDL Code for a D Latch](#)
- [Inference Report for a D Latch](#)
- [Synthesized Design for a D Latch](#)

VHDL Code for a D Latch

To implement a D latch in VHDL, you must set the `hdlin_report_inferred_modules` variable to `true`. [Example D-4](#) shows the VHDL code that implements a design using a simple D latch.

Example D-4 VHDL Code for a D Latch

```

library IEEE, SYNOPSYS;
use IEEE.std_logic_1164.all;
use SYNOPSYS.attributes.all;

entity d_latch is
    port ( enable, data : in std_logic;
           y : out std_logic );
end d_latch;

architecture behavioral of d_latch is
begin
    infer : process ( enable, data )
    begin
        if ( enable = '1' )
        then
            y <= data;
        end if;
    end process infer;
end behavioral;

```

Inference Report for a D Latch

[Example D-5](#) shows the inference report for a D latch resulting from compilation of the code in [Example D-4](#).

Example D-5 Inference Report for a D Latch

```

Inferred memory devices in process 'infer'
  in routine d_latch line 13 in file
    '/home/sudipto/work/latch_appl/rtl/vhdl
     /d_latch.vhdl'.
=====
|   Register Name   |   Type    | Width | Bus | MB | AR | AS | SR | SS | ST |
=====
|       y_reg      |   Latch   |    1   | -  | -  | N  | N  | -  | -  | -  |
=====
y_reg
-----
reset/set: none

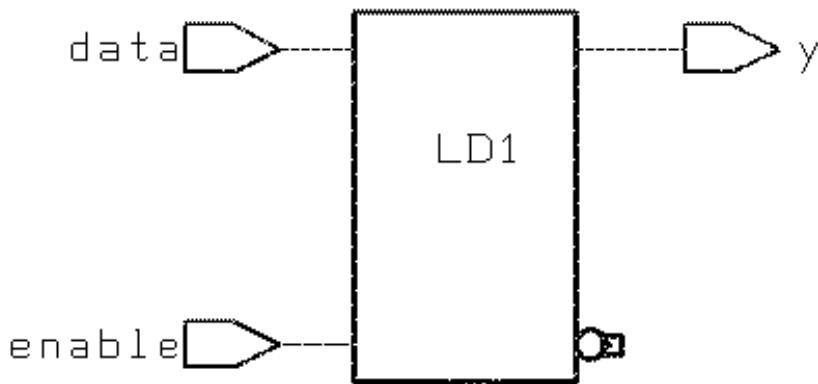
```

Synthesized Design for a D Latch

Figure D-2 shows the synthesized design for the D latch resulting from compilation of the code in [Example D-4](#). In this design, LD1 is the simple D latch. Here is the target library description of the latch for the cell LD1:

```
latch ("IQ", "IQN") {  
    enable : "G";  
    data_in : "D";  
}
```

Figure D-2 Synthesized Design for a D Latch



D Latch With Asynchronous Reset

The following topics show a VHDL and Verilog code example of a D latch with asynchronous reset and the resulting inference report and synthesized design:

- [VHDL and Verilog Code for a D Latch With Asynchronous Reset](#)
- [Inference Report for a D Latch With Asynchronous Reset](#)
- [Synthesized Design for a D Latch With Asynchronous Reset](#)

VHDL and Verilog Code for a D Latch With Asynchronous Reset

To implement a D latch with asynchronous reset in either VHDL or Verilog, you must set the `hdlin_report_inferred_modules` variable to `true`. You must also set the following attribute, as illustrated by the code examples:

```
attribute async_set_reset of RESET, SET : signal is "true";
```

[Example D-6](#) shows VHDL code for a D latch with asynchronous reset.

Example D-6 VHDL Code for a D Latch With Asynchronous Reset

```
library IEEE, SYNOPSYS;
use IEEE.std_logic_1164.all;
use SYNOPSYS.attributes.all;

entity d_latch_async_reset is
    port ( enable, reset, data : in std_logic;
           q : out std_logic );
end d_latch_async_reset;

architecture behavioral of d_latch_async_reset is
attribute async_set_reset of reset : signal is "true";
begin
    infer : process ( enable, reset, data )
    begin
        if ( reset = '1' )
        then
            q <= '0';
        elsif ( enable = '1' )
        then
            q <= data;
        end if;
    end process infer;
end behavioral;
```

[Example D-7](#) shows Verilog code for the D latch with asynchronous reset.

Example D-7 Verilog Code for a D Latch With Asynchronous Reset

```
module d_latch_async_reset (enable, reset, data, q) ;
    input enable, data, reset ;
    output q ;
    // synopsys async_set_reset "reset"
    reg q ;

    always @ (reset or enable or data)
        begin : infer
            if (reset == 1)
                q = 1'b0 ;
            else if (enable == 1)
                q = data ;
        end
    endmodule
```

Inference Report for a D Latch With Asynchronous Reset

[Example D-8](#) shows the inference report for a D latch with asynchronous reset resulting from compilation of the code in [Example D-6](#) or [Example D-7](#).

Example D-8 Inference Report for a D Latch With Asynchronous Reset

```
Inferred memory devices in process 'infer'
  in routine d_latch_async_reset
    line 13 in file
      '/home/sudipto/work/latch_appl/rtl/vhdl/d_latch_async_reset.vhdl'.
=====
|   Register Name    |   Type     | Width | Bus | MB | AR | AS | SR | SS | ST |
=====
|       q_reg        |   Latch    |   1   | -  | -  | Y  | N  | -  | -  | -  |
=====
q_reg
-----
  Async-reset: reset
```

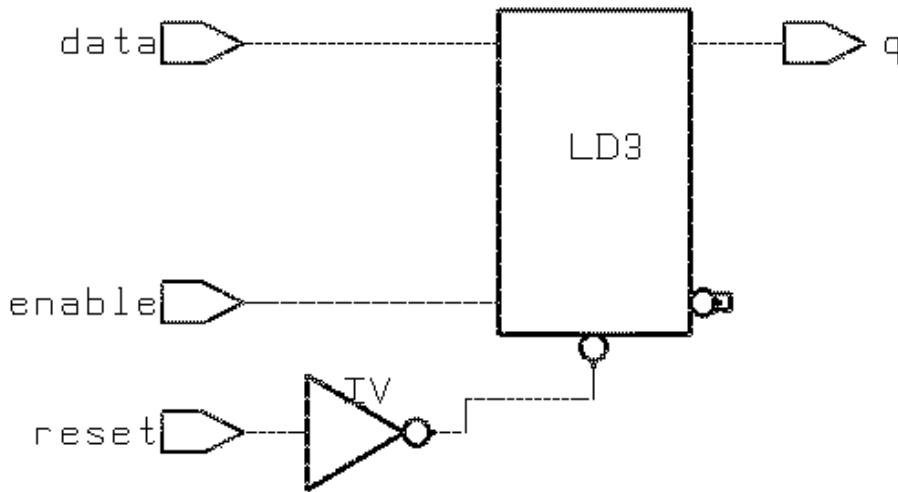
Synthesized Design for a D Latch With Asynchronous Reset

[Figure D-3](#) shows the synthesized design for the D latch resulting from compilation of the code shown in [Example D-6](#) and [Example D-7](#). In this design, LD3 is the simple D latch. Here is the target library description of the latch for the cell LD3:

LD3 is a D latch with asynchronous active-low reset (CD). The description of the latch for the cell LD3 in the target library is as follows:

```
latch ("IQ","IQN") {
  enable : "G";
  data_in : "D";
  clear : "CD'";
}
```

Figure D-3 Synthesized Design for a D Latch With Asynchronous Reset



D Latch With Asynchronous Set and Reset

The following topics show a code example of a D latch with asynchronous set and reset in VHDL and Verilog and the resulting inference report and synthesized design:

- [VHDL and Verilog Code for a D Latch With Asynchronous Set and Reset](#)
- [Inference Report for a D Latch With Asynchronous Set and Reset](#)
- [Synthesized Design for a D Latch With Asynchronous Set and Reset](#)

VHDL and Verilog Code for a D Latch With Asynchronous Set and Reset

To implement a D latch with asynchronous set and reset in either VHDL or Verilog, you must set the `hdlin_report_inferred_modules` variable to `true`. You must also set the following attributes, as illustrated by the code examples:

```
attribute async_set_reset of set, reset : signal is "true";
attribute one_hot of set, reset : signal is "true";
```

[Example D-9](#) shows the VHDL code for a D latch with asynchronous set and reset attributes.

Example D-9 VHDL Code for a D Latch With Asynchronous Set and Reset

```
library IEEE, SYNOPSYS;
use IEEE.std_logic_1164.all;
use SYNOPSYS.attributes.all;

entity d_latch_async_set_reset is
    port ( enable, set, reset, data : in std_logic;
           q : out std_logic );
end d_latch_async_set_reset;

architecture behavioral of d_latch_async_set_reset is
attribute async_set_reset of set, reset : signal is "true";
attribute one_hot of set, reset : signal is "true";
begin
    infer : process ( enable, set, reset, data )
begin
    if ( reset = '1' )
    then
        q <= '0';
    elsif ( set = '1' )
    then
        q <= '1';
    elsif ( enable = '1' )
    then
        q <= data;
    end if;
end process infer;
end behavioral;
```

[Example D-10](#) shows the Verilog code for a D latch with asynchronous set and reset attributes.

Example D-10 Verilog Code for a D Latch With Asynchronous Set and Reset

```
module d_latch_async_set_reset
(enable, set, reset, q, data) ;

input enable, set, reset, data ;
output q ;

// synopsys async_set_reset "set, reset"
// synopsys one_hot "set, reset"

reg q ;

always @ (enable or set or reset or data)
begin : infer
    if (reset == 1)
        q = 1'b0 ;
    else if (set == 1)
        q = 1'b1 ;
    else if (enable == 1)
        q = data ;
end
endmodule
```

Inference Report for a D Latch With Asynchronous Set and Reset

[Example D-11](#) shows the inference report for a D latch with asynchronous set and reset resulting from compilation of the code shown in [Example D-9 on page D-10](#) or [Example D-10](#).

Example D-11 Inference Report for a D Latch With Asynchronous Set and Reset

```
Inferred memory devices in process 'infer'
  in routine d_latch_async_set_reset
  line 14 in file
  '/home/sudipto/work/latch_appl/rtl/vhdl/d_latch_async_set_reset.vhdl'.
=====
| Register Name | Type | Width | Bus | MB | AR | AS | SR | SS | ST |
=====
|     q_reg     | Latch |   1   | - | - | Y | Y | - | - | - |
=====

q_reg
-----
  Async-reset: reset
  Async-set: set
  Async-set and Async-reset ==> Q: X
```

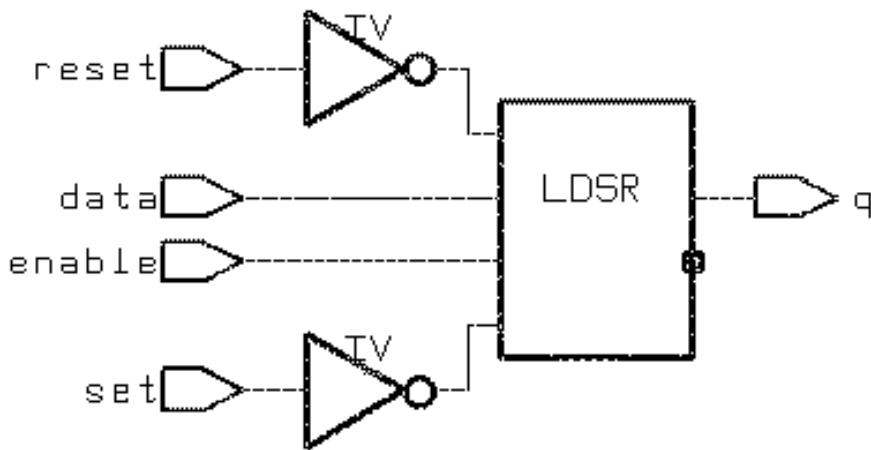
Synthesized Design for a D Latch With Asynchronous Set and Reset

Figure D-4 shows the synthesized design for the D latch resulting from compilation of the code in [Example D-9](#) and [Example D-10](#). In this design, LDSR is a D latch with active-low asynchronous set (SET) and reset (CLR).

The description of the latch for the cell LDSR in the target library is as follows:

```
latch ("IQ", "IQN") {  
    enable : "G";  
    data_in : "D";  
    clear : "CLR'";  
    preset : "SET'";  
    clear_preset_var1 : L;  
    clear_preset_var2 : L;  
}
```

Figure D-4 Synthesized Design for a D Latch With Asynchronous Set and Reset



D Latch With Enable (Avoiding Clock Gating)

The following topics show a VHDL and Verilog code example of a D latch with the enable attribute to avoid clock gating and the resulting inference report and synthesized design:

- [VHDL and Verilog Code for a D Latch With Enable](#)
- [Inference Report for a D Latch With Enable](#)

- [Synthesized Design for a D Latch With Enable](#)
- [Inferring Gated Clocks](#)

VHDL and Verilog Code for a D Latch With Enable

To implement a D latch with enable in either VHDL or Verilog, you must set the `hdlin_report_inferred_modules` variable to `true`. You must also set the `hdlin_keep_feedback` attribute to `true`.

[Example D-12](#) shows the VHDL code for a D latch with enable.

Example D-12 VHDL Code for a D Latch With Enable

```
library IEEE;
use IEEE.STD_LOGIC_1164.all;

entity d_latch_enab is
    port ( enable, clock, data : in std_logic;
           q : buffer std_logic );
end d_latch_enab;

architecture behavioral of d_latch_enab is
begin
    infer : process ( enable, clock, data )
        begin
            if ( clock = '1' )
            then
                if ( enable = '1' )
                then
                    q <= data;
                else
                    q <= q;
                end if;
            end if;
        end process infer;
    end behavioral;
```

[Example D-13](#) shows the Verilog code for a D latch with enable.

Example D-13 Verilog Code for a D Latch With Enable

```
module d_latch_enab ( enable, clock, data, q) ;
    input enable, clock, data ;
    output q ;
    reg q ;

    always @ (enable or clock or data)
        begin :infer
            if (clock == 1)
                begin
                    if (enable == 1)
                        q = data ;
                    else
                        q = q ;
                end
        end
    endmodule
```

Inference Report for a D Latch With Enable

[Example D-14](#) shows the inference report for a D latch with enable resulting from compilation of the code in [Example D-12 on page D-13](#) or [Example D-13](#).

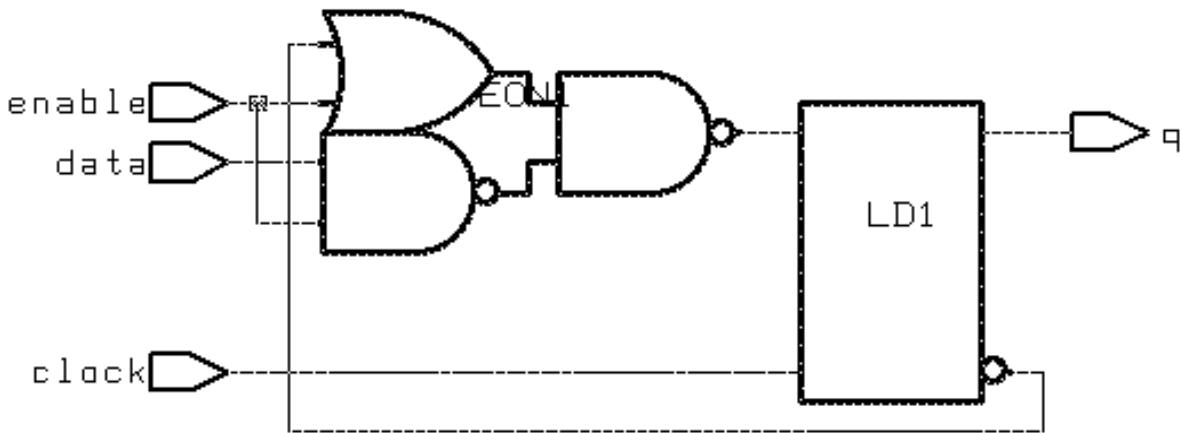
Example D-14 Inference Report for a D Latch With Enable

```
Inferred memory devices in process 'infer'
    in routine d_latch_enab line 13 in
        file '/home/sudipto/work/latch_appl/rtl/vhdl/d_latch_enab.vhdl'.
=====
|   Register Name   |   Type    | Width | Bus | MB | AR | AS | SR | SS | ST |
=====
|       q_reg      |   Latch   |    1   | -  | -  | N  | N  | -  | -  | -  |
=====
q_reg
-----
reset/set: none
```

Synthesized Design for a D Latch With Enable

Figure D-5 shows the synthesized design for the D latch with enable resulting from compilation of the code in [Example D-12](#) or [Example D-13](#).

Figure D-5 Synthesized Design for a D Latch With Enable



Inferring Gated Clocks

This topic describes two cases—Case 1 and Case 2—in which HDL Compiler infers gated clocks.

Case 1

If the variable `hdlin_keep_feedback` is not set to `true`, HDL Compiler assumes the default value of `false` and removes all feedback loops. For example, feedback loops inferred from a statement such as the following

`Q = Q`

are removed.

The loop that is inferred from the following statement, shown in VHDL code, is removed.

```
if ( enable = '1' )
then
    q<= data;
else
    q <= q;
end if;
```

The code indicates that the gated clock in the synthesized design in [Figure D-6](#), which does not have a feedback loop, is removed.

Case 2

Gated clocks can also be inferred from the coding style used to implement a design. For example, if the VHDL code is written in either of the coding styles in [Example D-15](#) or [Example D-16](#), regardless of whether `hdlin_keep_feedback` is set to `true`, Design Compiler will create a gated clock for the design.

[Example D-15](#) implies a priority coding style—that is, the clock value is assessed first and `enable` is considered only if the clock is a certain value.

Example D-15 Coding Style A

```
library IEEE;
use IEEE.STD_LOGIC_1164.all;

entity d_latch_enab is
    port ( enable, clock, data : in std_logic;
           q : out std_logic );
end d_latch_enab;

architecture behavioral of d_latch_enab is
begin
    infer : process ( enable, clock, data )
    begin
        if ( clock = '1' )
        then
            if ( enable = '1' )
            then
                q <= data;
            end if;
        end if;
    end process infer;
end behavioral;
```

For [Example D-16](#), no priority is implied.

Example D-16 Coding Style B

```
library IEEE;
use IEEE.STD_LOGIC_1164.all;

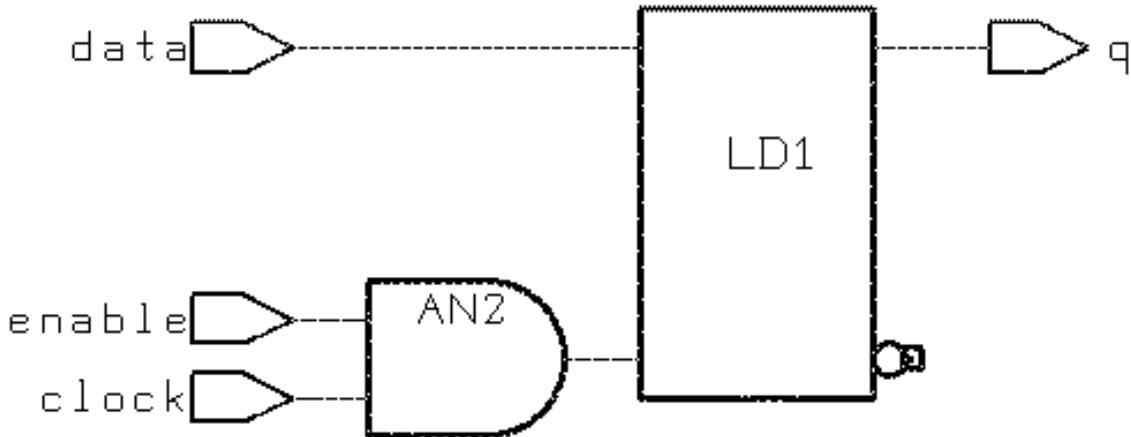
entity d_latch_enab is
    port ( enable, clock, data : in std_logic;
           q : out std_logic );
end d_latch_enab;

architecture behavioral of d_latch_enab is
begin
    infer : process ( enable, clock, data )
    begin
        if ( clock = '1' and enable = '1' )
            q <= data;
        end if;
    end process infer;
end behavioral;
```

Synthesized Design With Enable and Gated Clock

[Figure D-6](#) shows the synthesized design for the D latch with enable and clock gating resulting from compilation of the code in [Example D-15](#) on page [D-16](#) and [Example D-16](#).

Figure D-6 D Latch With Enable and Gated Clock



D Latch With Enable and Asynchronous Reset

The following topics show the VHDL and Verilog code that implements a design that uses a D latch with the enable and asynchronous reset attributes:

- [VHDL and Verilog Code for a D Latch With Enable and Asynchronous Reset](#)
 - [Synthesized Design for a D Latch With Enable and Asynchronous Reset](#)
-

VHDL and Verilog Code for a D Latch With Enable and Asynchronous Reset

To implement a D latch with enable and asynchronous reset in either VHDL or Verilog, you must set the `hdlin_report_inferred_modules` and `hdlin_keep_feedback` variables to true. You must also set the following attribute:

```
attribute async_set_reset of reset : signal is "true";
```

[Example D-17](#) shows the VHDL code for a D latch with enable and asynchronous reset.

Example D-17 VHDL Code for a D Latch With Enable and Asynchronous Reset

```
library IEEE, SYNOPSYS;
use IEEE.STD_LOGIC_1164.all;
use SYNOPSYS.attributes.all;

entity d_latch_enab_async_reset is
    port ( enable, clock, reset, data : in std_logic;
           q : buffer std_logic );
end d_latch_enab_async_reset;

architecture behavioral of d_latch_enab_async_reset is
begin
    attribute async_set_reset of reset : signal is "true";
    infer : process ( enable, clock, reset, data )
        variable temp : std_logic;
    begin
        temp := q;
        if ( reset = '1' ) then
            q <= '0';
        elsif ( clock = '1' ) then
            case enable is
                when '1' => q <= data;
                when others => q <= temp;
            end case;
        end if;
    end process infer;
end behavioral;
```

[Example D-18](#) shows the Verilog code for a D latch with enable and asynchronous reset.

Example D-18 Verilog Code for a D Latch With Enable and Asynchronous Reset

```
module d_latch_enab_async_reset
  (enable, clock, reset, q, data) ;

  input enable, clock, reset, data ;
  output q ;

  // synopsys async_set_reset "reset"

  reg q ;

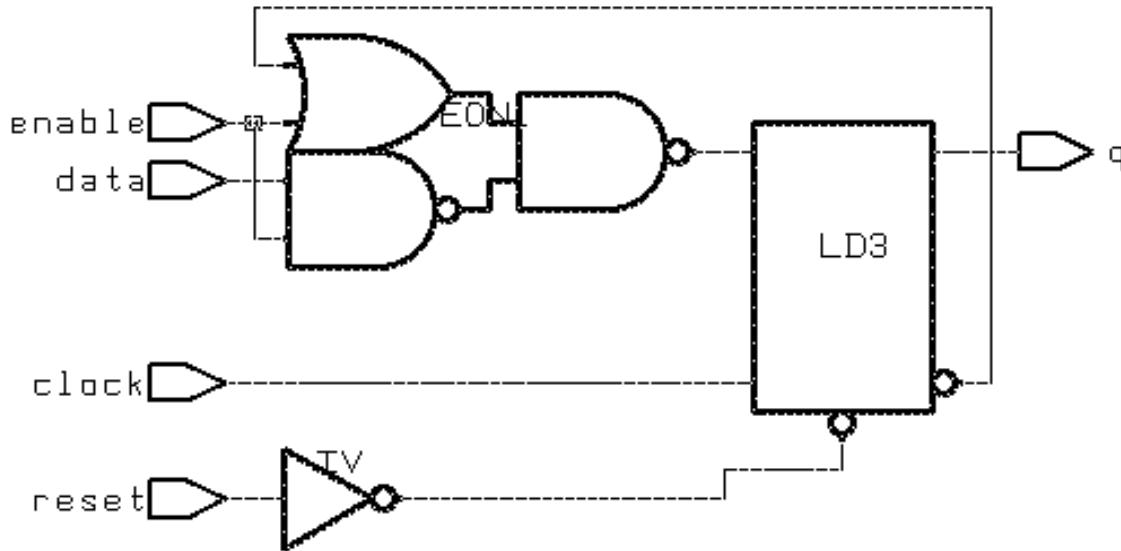
  always @ (enable or clock or reset or data)
    begin : infer
      if (reset == 1)
        q = 1'b0 ;

      else if (clock == 1)
        begin
          if (enable == 1)
            q = data ;
          else
            q = q ;
        end
    end
  endmodule
```

Synthesized Design for a D Latch With Enable and Asynchronous Reset

Figure D-7 shows the synthesized design for the D latch with enable and asynchronous reset resulting from compilation of the code in [Example D-17](#) or [Example D-18](#).

Figure D-7 Synthesized Design for a D Latch With Enable and Asynchronous Reset



D Latch With Enable and Asynchronous Set

The following topics show the VHDL and Verilog code that implements a design that uses a D latch with the enable and asynchronous set attributes:

- [VHDL and Verilog Code for a D Latch With Enable and Asynchronous Set](#)
- [Synthesized Design for D Latch With Enable and Asynchronous Set](#)

VHDL and Verilog Code for a D Latch With Enable and Asynchronous Set

To implement a D latch with enable and asynchronous set in either VHDL or Verilog, you must set the `hdlin_report_inferred_modules` and `hdlin_keep_feedback` variables to true. You must also set the following attribute:

```
attribute async_set_reset of set : signal is "true";
```

Example D-19 shows the VHDL code for a D latch with enable and asynchronous reset.

Example D-19 VHDL Code for D Latch With Enable and Asynchronous Set

```
library IEEE, SYNOPSYS;
use IEEE.STD_LOGIC_1164.all;
use SYNOPSYS.attributes.all;

entity d_latch_enab_async_set is
    port ( enable, clock, set, data : in std_logic;
           q : buffer std_logic );
end d_latch_enab_async_set;

architecture behavioral of d_latch_enab_async_set is
attribute async_set_reset of set : signal is "true";
begin
    infer : process ( enable, clock, set, data )
    begin
        if ( set = '1' )
        then
            q <= '1';
        elsif ( clock = '1' )
        then
            if ( enable = '1' )
            then
                q<= data;
            else
                q <= q;
            end if;
        end if;
    end process infer;
end behavioral;
```

[Example D-20](#) shows the Verilog code for a D latch with enable and asynchronous set.

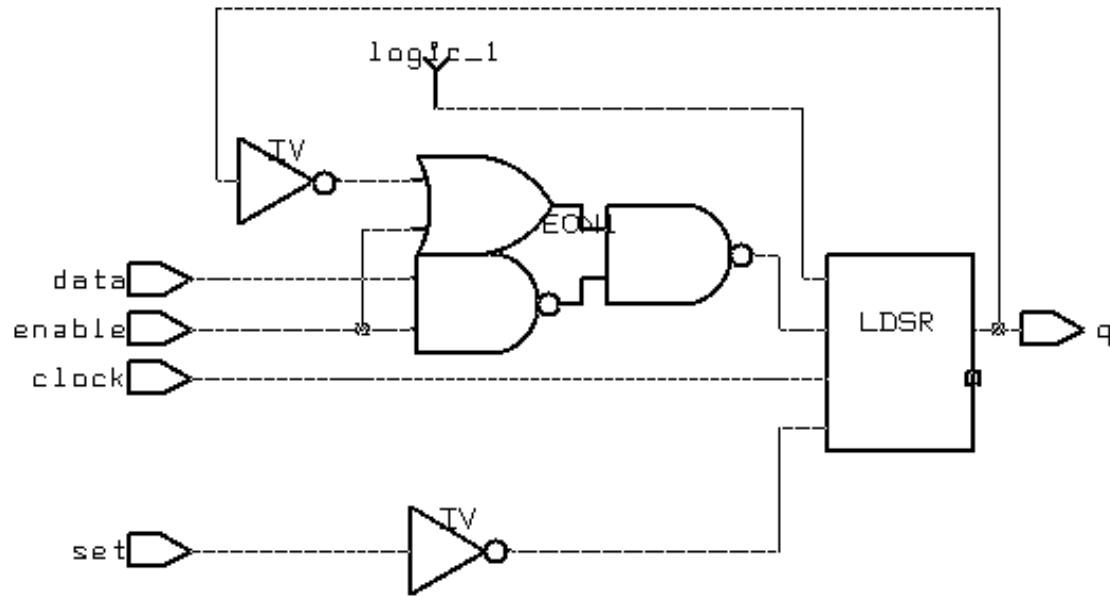
Example D-20 Verilog Code for D Latch With Enable and Asynchronous Set

```
module d_latch_enab_async_set (enable, clock, set, q, data) ;  
  
    input enable, clock, set, data ;  
    output q ;  
  
    // synopsys async_set_reset "set"  
  
    reg q ;  
  
    always @ (enable or clock or set or data)  
        begin : infer  
            if (set == 1)  
                q = 1'b1 ;  
  
            else if (clock == 1)  
                begin  
                    if (enable == 1)  
                        q = data ;  
                    else  
                        q = q ;  
                end  
            end  
        endmodule
```

Synthesized Design for D Latch With Enable and Asynchronous Set

Figure D-8 shows the synthesized design for the D latch with enable and asynchronous set resulting from compilation of the code shown in [Example D-19](#) or [Example D-20](#).

Figure D-8 Synthesized Design for D Latch With Enable and Asynchronous Set



D Latch With Enable and Asynchronous Set and Reset

The following topics show the VHDL and Verilog code that implements a design that uses a D latch with the enable and asynchronous set and reset attributes:

- [VHDL and Verilog Code for D Latch With Enable and Asynchronous Set and Reset](#)
- [Synthesized Design for D Latch With Enable and Asynchronous Set and Reset](#)

VHDL and Verilog Code for D Latch With Enable and Asynchronous Set and Reset

To implement a D latch with enable and asynchronous set and reset in either VHDL or Verilog, you must set the `hdlin_report_inferred_modules` and `hdlin_keep_feedback` variables to `true`. You must also set the following attributes:

```
attribute async_set_reset of set : signal is "true";
attribute one_hot of set, reset: signal is "true";
```

[Example D-21](#) shows the VHDL code for a D latch with enable and asynchronous set and reset.

Example D-21 VHDL Code for D Latch With Enable and Asynchronous Set and Reset

```
library IEEE, SYNOPSYS;
use IEEE.STD_LOGIC_1164.all;
use SYNOPSYS.attributes.all;

entity d_latch_enab_async_set_reset is
    port ( enable, clock, set, reset, data : in std_logic;
           q : buffer std_logic );
end d_latch_enab_async_set_reset;

architecture behavioral of d_latch_enab_async_set_reset is
attribute async_set_reset of set, reset: signal is "true";
attribute one_hot of set, reset : signal is "true";
begin
    infer : process (enable,clock, set, reset, data)
    begin
        if ( set = '1' ) then
            q <= '1';
        elsif ( reset = '1' ) then
            q <= '0';
        elsif ( clock = '1' ) then
            if ( enable = '1') then
                q <= data;
            else
                q <= q;
            end if;
        end if;
    end process infer;
end behavioral;
```

[Example D-22](#) shows the Verilog code for a D latch with enable and asynchronous set and reset.

Example D-22 Verilog Code for D Latch With Enable and Asynchronous Set and Reset

```
module d_latch_enab_async_set_reset (enable, clock, set,
reset, q, data);
input enable, clock, set, reset, data ;
output q ;
// synopsys async_set_reset "set, reset"
// synopsys one_hot "set, reset"
reg q ;

always @ (enable or clock or set or reset or data)
begin : infer
    if (reset == 1)
        q = 1'b0 ;
    else if (set == 1)
        q = 1'b1 ;
    else if (clock == 1)
        begin
            if (enable == 1)
                q = data ;
            else
                q = q ;
        end
    end
endmodule
```

Synthesized Design for D Latch With Enable and Asynchronous Set and Reset

Figure D-9 shows the synthesized design for the D latch with enable and asynchronous set and reset resulting from compilation of the code shown in [Example D-21](#) or [Example D-22](#).

Figure D-9 Synthesized Design for D Latch With Enable and Asynchronous Set and Reset

