

UltraFast Design Methodology Guide for Xilinx FPGAs and SoCs

UG949 (v2021.2) November 19, 2021

Xilinx is creating an environment where employees, customers, and partners feel welcome and included. To that end, we're removing non-inclusive language from our products and related collateral. We've launched an internal initiative to remove language that could exclude people or reinforce historical biases, including terms embedded in our software and IPs. You may still find examples of non-inclusive language in our older products as we work to make these changes and align with evolving industry standards. Follow this [link](#) for more information.



Revision History

The following table shows the revision history for this document.

Section	Revision Summary
11/19/2021 Version 2021.2	
Reduce the Number of Partition Pins	Added PPLOC examples and graphics.
Make Pblocks as Rectangular as Possible to Avoid Unroutability at the Edges	Updated examples.
Incremental Synthesis	Added information on different modes.
ML Strategies	Added non-project mode description and ML strategy suggestions.
Connecting a Net to a Free External Pin Using Post-Route ECO	Updated section.
08/18/2021 Version 2021.1	
Power Distribution System	Added XPE landing page and changed XADC to Sysmon.
Power Rail Consolidation Impacting Power	Added tip about power rail constraints.
Clocking Recommendations for Platforms and Dynamic Function eXchange	Added new section.
Chapter 4: Design Constraints	Added note about traditional and platform-based design flows.
Constraining Input and Output Ports	Added note about I/O logic.
Defining Power and Thermal Constraints	Added new section.
Floorplanning Constraints for Dynamic Function eXchange	Added new section.
Chapter 6: Design Closure	Updated design closure description.
Timing Closure	Added timing result note.
Checking for Valid Constraints	Added baselining design to note.
Checking for Positive Timing Slacks	Updated timing score description.
Checking That Your Design is Properly Constrained	Added timing constraint note.
Fixing Issues Flagged by report_methodology	Added methodology violation note and link to methodology blog.
Methodology DRCs with Impact on Timing Closure	Added link to <i>Vivado Design Suite User Guide: Design Analysis and Closure Techniques (UG906)</i> .
Assessing the Maximum Frequency of the Design	Updated WNS description.
Clock Skew and Uncertainty	Added clock uncertainty description and related links.
Using Intelligent Design Runs	Added new section.
Power Closure	Added power optimization capabilities description.
Power Timing Slack	Added new section.

Table of Contents

Revision History	2
Chapter 1: Introduction	5
About the UltraFast Design Methodology.....	5
Understanding UltraFast Design Methodology Concepts.....	8
Using the Vivado Design Suite.....	12
Accessing Additional Documentation and Training.....	13
Chapter 2: Board and Device Planning	14
PCB Layout Recommendations.....	14
Device Power Aspects and System Dependencies.....	19
Clock Resource Planning and Assignment.....	23
I/O Planning Design Flows.....	23
Designing with SSI Devices.....	29
Designing with HBM Devices.....	36
Configuration.....	40
Chapter 3: Design Creation with RTL	42
Defining a Good RTL Design Hierarchy.....	42
Working with Intellectual Property (IP).....	46
RTL Coding Guidelines.....	49
Clocking Guidelines.....	87
Clock Domain Crossing.....	139
Chapter 4: Design Constraints	144
Organizing the Design Constraints	144
Defining Timing Constraints in Four Steps.....	150
Defining Clock Constraints.....	151
Constraining Input and Output Ports.....	159
Defining Clock Groups and CDC Constraints.....	169
Specifying Timing Exceptions.....	177
Adding Multicycle Path Constraints.....	181
Other Advanced Timing Constraints.....	184

Defining Power and Thermal Constraints.....	185
Defining Physical Constraints.....	186
Chapter 5: Design Implementation.....	198
Running Synthesis.....	198
Moving Past Synthesis.....	206
Implementing the Design.....	211
Chapter 6: Design Closure.....	219
Timing Closure.....	220
Power Closure.....	307
Configuration and Debug.....	313
Appendix A: Additional Resources and Legal Notices.....	323
Xilinx Resources.....	323
Solution Centers.....	323
Documentation Navigator and Design Hubs.....	323
References.....	324
Training Resources.....	326
Please Read: Important Legal Notices.....	327

Introduction

About the UltraFast Design Methodology

The Xilinx[®] UltraFast[™] design methodology is a set of best practices intended to help streamline the design process for today's devices. The size and complexity of these designs require specific steps and design tasks to ensure success at each stage of the design. Following these steps and adhering to the best practices will help you achieve your desired design goals as quickly and efficiently as possible.

- This guide, which describes the various design tasks, analysis and reporting features, and best practices for design creation and closure.
- *UltraFast Design Methodology Quick Reference Guide (UG1231)*, which highlights key design methodology steps in an easy-to-use, double-sided card format.
- *UltraFast Design Methodology Timing Closure Quick Reference Guide (UG1292)*, which covers recommendations for closing timing, including running initial design checks, baselining the design, and resolving timing violations.
- *UltraFast Design Methodology Checklist (XTP301)*, which is available in the Xilinx Documentation Navigator and as a standalone spreadsheet. You can use this checklist to identify common mistakes and decision points throughout the design process.
- UltraFast Design Methodology System-Level Design Flow diagram representing the entire Vivado[®] Design Suite design flow, which is available in the Xilinx Documentation Navigator. You can click a design step in the diagram to open related documentation, collateral, and FAQs to help get you started.



RECOMMENDED: In addition to these resources, Xilinx recommends the *UltraFast Embedded Design Methodology Guide (UG1046)* when working with embedded designs and the *Vitis HLS Methodology in the Vitis HLS User Guide (UG1399)* when developing complex systems using Vivado IP integrator with C-based IP.

Xilinx provides the following resources to help you take advantage of the UltraFast design methodology:



TIP: Xilinx also provides methodology-related design rule checks (DRCs) for each design stage, which are available using the `report_methodology` Tcl command in the Vivado Design Suite.

Using This Guide

This guide provides a set of best practices that maximize productivity for both system integration and design implementation. It includes high-level information, design guidelines, and design decision trade-offs for the following topics:

- **Chapter 2: Board and Device Planning:** Covers decisions and design tasks that Xilinx recommends accomplishing prior to design creation. These include I/O and clock planning, PCB layout considerations, device capacity and throughput assessment, alternate device definition, power estimation, and debugging.
- **Chapter 3: Design Creation with RTL:** Covers the best practices for RTL definition and IP configuration and management.
- **Chapter 4: Design Constraints:** Provides recommendations for creating proper timing, power, and physical constraints as well as specifying additional constraints, attributes, and other elements used during synthesis and implementation.
- **Chapter 5: Design Implementation:** Covers the options available and best practices for synthesizing and implementing the design.
- **Chapter 6: Design Closure:** Covers the various design analysis and implementation techniques used to close timing on the design or to reduce power consumption. It also includes considerations for adding debug logic to the design for hardware verification purposes.

This guide includes references to other documents such as the Vivado Design Suite User Guides, Vivado Design Suite Tutorials, and Quick-Take Video Tutorials. This guide is not a replacement for those documents. Xilinx still recommends referring to those documents for detailed information, including descriptions of tool use and design methodology.

This information is designed for use with the Vivado Design Suite, but you can use most of the conceptual information with the ISE® Design Suite as well.

Related Information

[Additional Resources and Legal Notices](#)

Using the UltraFast Design Methodology Checklist

To take full advantage of the UltraFast design methodology, use this guide with the *UltraFast Design Methodology Checklist (XTP301)*. The checklist is available from the Xilinx Documentation Navigator or as a standalone spreadsheet.

The questions in the UltraFast Design Methodology Checklist highlight typical areas in which design decisions are likely to have downstream impact and draw attention to issues that are often overlooked or ignored. Each tab in the checklist:

- Targets a specific role within a typical design team.

- Includes common questions and recommended actions to take during each design flow step, including project planning, board and device planning, IP and submodule design, and top-level design closure.
- Includes a Documentation and Training section that lists resources related to the design flow step.
- Provides links to content in this guide or other Xilinx documentation, which offer guidance on addressing the design concerns raised by the questions.



VIDEO: For a demonstration of the checklist, see the [Vivado Design Suite QuickTake Video: Introducing the UltraFast Design Methodology Checklist](#).

Using the UltraFast Design Methodology DRCs

The Vivado Design Suite contains a set of methodology-related DRCs you can run using the `report_methodology` Tcl command. This command has rules for each of the following design stages:

- Before synthesis in the elaborated RTL design to validate RTL constructs
- After synthesis to validate the netlist and constraints
- After implementation to validate constraints and timing related concerns.



RECOMMENDED: For maximum effect, run the methodology DRCs at each design stage and address Critical Warnings and Warnings prior to moving to the next stage.

For more information on the design methodology DRCs, see the [report_methodology](#) Tcl command in the *Vivado Design Suite Tcl Command Reference Guide* (UG835).

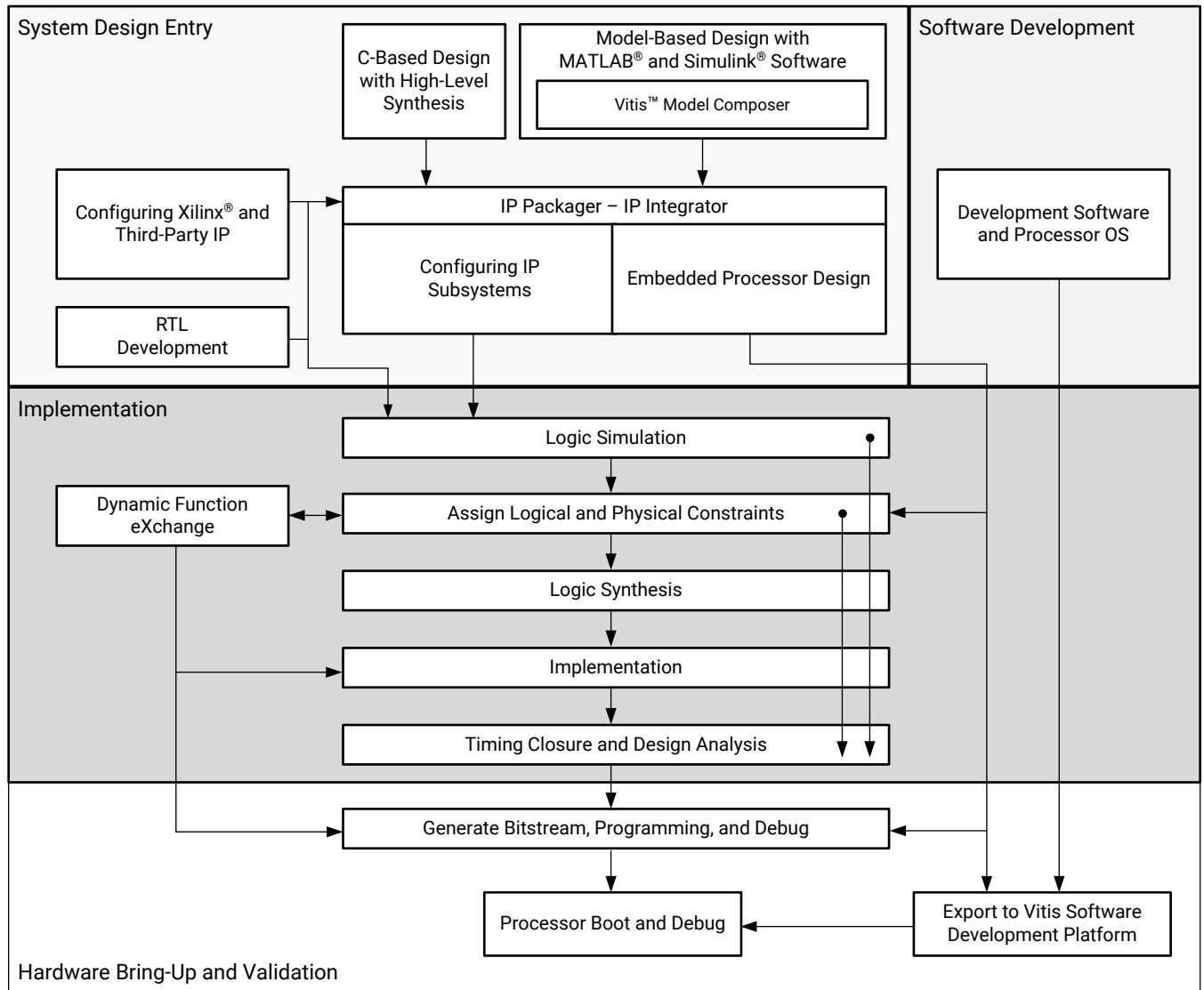
Related Information

[Running Report Methodology](#)

Using the UltraFast Design Methodology System-Level Design Flow Diagram

The following figure shows the various design steps and features included in the Vivado Design Suite. From the Xilinx Documentation Navigator Design Hub View, you can access an interactive version of this graphic in which you can click each step for links to related resources.

Figure 1: System-Level Design Flow



X15150-063021

Understanding UltraFast Design Methodology Concepts

It is important to take the correct approach from the start of your design and to pay attention to design goals from the early stages, including RTL, clock, pin, and PCB planning. Properly defining and validating the design at each design stage helps alleviate timing closure, routing closure, and power usage issues during subsequent stages of implementation.

Creating and Implementing a Hardware Design

After planning your device I/O, planning how to lay out your PCB, and deciding on your use model, you can begin creating your design. Design creation includes:

- Planning the hierarchy of your design
- Identifying the IP cores to use and customize in your design
- Instantiating RTL modules that are needed for special interconnect or functionality that is not available in the IP catalog
- Creating timing, power, and physical constraints
- Specifying additional constraints, attributes, and other elements used during synthesis and implementation

When creating your design, the main points to consider include:

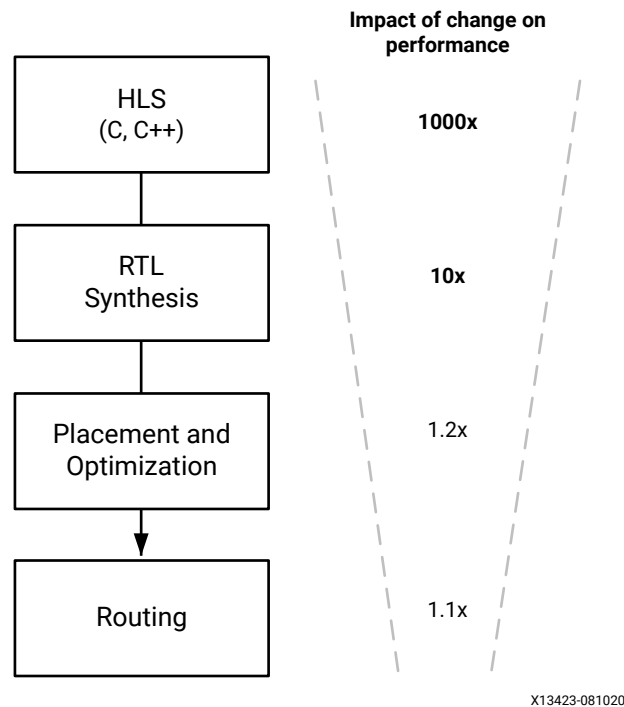
- Achieving the desired functionality
- Operating at the desired frequency
- Operating with the desired degree of reliability
- Fitting within the silicon resource and power budget

Decisions made at this stage affect the end product. A wrong decision at this point can result in problems at a later stage, causing issues throughout the entire design cycle. Spending time early in the process to carefully plan your design helps to ensure that you meet your design goals and minimize debug time in the lab.

Maximizing Impact Early in the Development Cycle

As shown in the following figure, early stages in the design flow (C, C++, and RTL synthesis) have a much higher impact on design performance, density, and power than the later implementation stages. Therefore, if the design does not meet timing goals, Xilinx recommends that you revisit the synthesis stage, including HDL and constraints, rather than iterating for a solution in the implementation stages only.

Figure 2: Impact of Design Changes Throughout the Flow



X13423-081020

Validating at Each Design Stage

The UltraFast design methodology emphasizes the importance of monitoring design budgets, such as area, power, latency, and timing, and correcting the design from early stages as follows:

- Create optimal RTL constructs with Xilinx templates, and validate your RTL with methodology DRCs prior to synthesis, after elaboration.

Because the Vivado tools use timing-driven algorithms throughout, the design must be properly constrained from the beginning of the design flow.

- Perform timing analysis after synthesis.

To specify correct timing, you must analyze the relationship between each master clock and related generated clocks in the design. In the Vivado tools, each clock interaction is timed unless explicitly declared as an asynchronous or false path.

- Meet timing using the right constraints before proceeding to the next design stage.

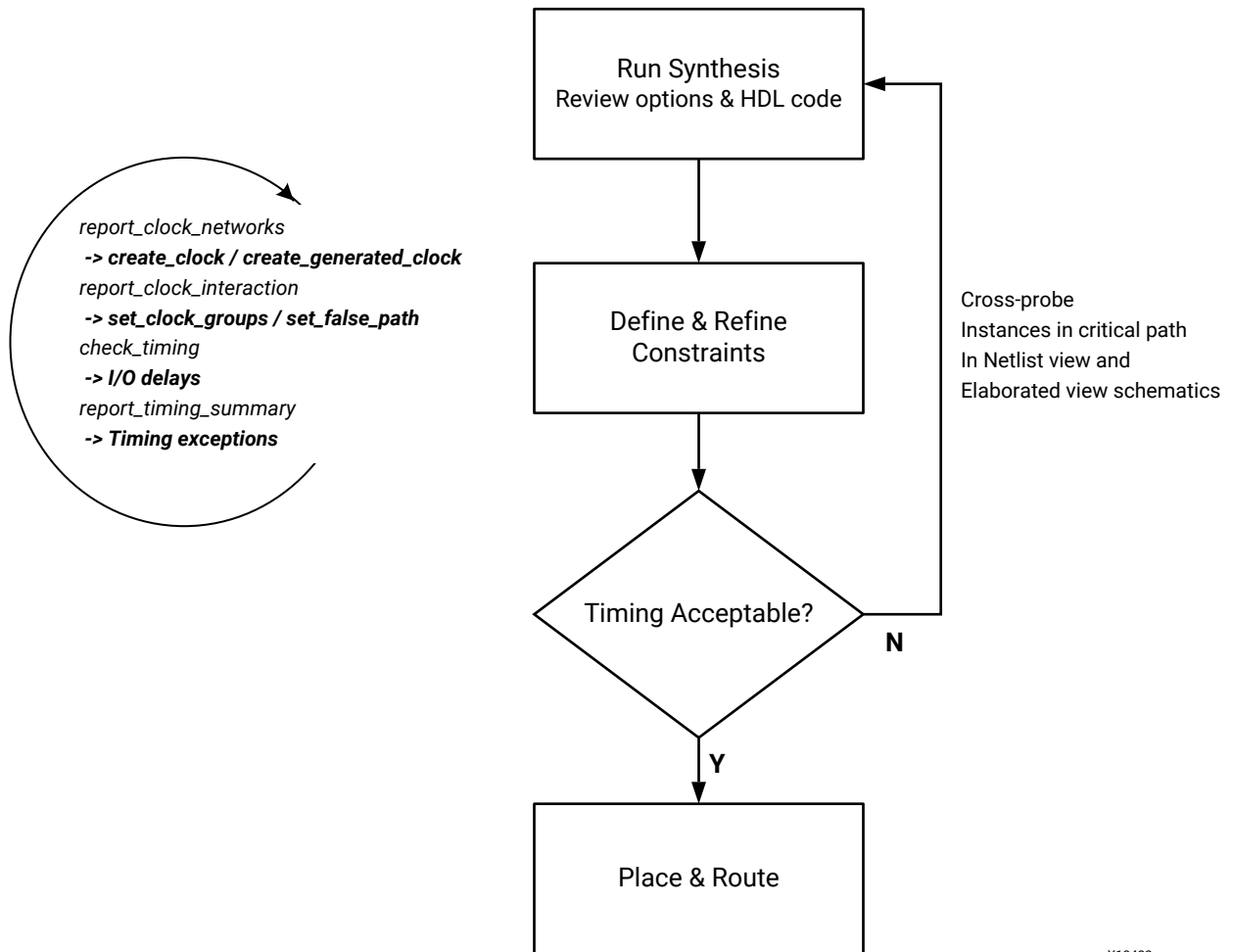
You can accelerate overall timing and implementation convergence by following this recommendation and by using the interactive analysis environment of the Vivado Design Suite.



TIP: You can achieve further acceleration by combining these recommendations with the HDL design guidelines in this guide.

The following figure shows this recommended design methodology.

Figure 3: RTL Design Methodology for Rapid Convergence



X13422

Synthesis is considered complete when the design goals are met with a positive margin or a relatively small negative timing margin. For example, if post-synthesis timing is not met, placement and routing results are not likely to meet timing. However, you can still go ahead with the rest of the flow even if timing is not met. Implementation tools might be able to close timing if they can allocate the best resources to the failing paths. In addition, proceeding with the flow provides a more accurate understanding of the negative slack magnitude, which helps you determine how much you need to improve the post-synthesis worst negative slack (WNS). You can use this information when you return to the synthesis stage with improvements to HDL and constraints.

Taking Advantage of Rapid Validation

This guide also introduces the concept of rapid validation of specific aspects of the system architecture and micro-architecture as follows:

- In the context of system design, the I/O bandwidth is validated in-system, before implementing the entire design. Validating I/O bandwidth can highlight the need to revise system architecture and interface choices before finalizing on I/Os.
- As part of design implementation, baselining is used to write the simplest set of constraints, which can identify internal device timing challenges. Baselining is a process used to identify the need to revise RTL micro-architecture choices before moving to the implementation phase.

Related Information

[Interface Bandwidth Validation](#)

[Baselining the Design](#)

Using the Vivado Design Suite

The Vivado Design Suite has a flexible use model to accommodate various development flows and different types of designs. For detailed information on how to use the features within the Vivado Design Suite, see the *Vivado Design Suite User Guide: Design Flows Overview* ([UG892](#)) and other Vivado Design Suite documentation.

Managing Vivado Design Suite Sources with a Revision Control System

Most design teams manage their design sources and results with a commercially available revision control system. The Vivado Design Suite allows various use models for managing design and IP data. For more information on using the Vivado tools with a revision control system, see this [link](#) in the *Vivado Design Suite User Guide: Design Flows Overview* ([UG892](#)).

Upgrading to New Vivado Design Suite Releases

New releases of the Vivado Design Suite often contain updates to Xilinx IP. Carefully consider whether you want to upgrade your IP, because upgrading can result in design changes. In addition, you must follow specific rules when using IP configured with previous releases going forward. For more information, see this [link](#) in the *Vivado Design Suite User Guide: Designing with IP* ([UG896](#)).

Accessing Additional Documentation and Training


This guide supplements the information in the Vivado Design Suite documentation, including user guides, reference guides, tutorials, and QuickTake videos. The Xilinx Documentation Navigator provides access to the Vivado Design Suite documentation and support resources, which you can filter and search to find information. To open the Xilinx Documentation Navigator (DocNav):

- From the Vivado IDE, select **Help** → **Documentation and Tutorials**.
- On Windows, select **Start** → **All Programs** → **Xilinx Design Tools** → **DocNav**.
- At the Linux command prompt, enter: `docnav`

Xilinx Design Hubs provide links to documentation organized by design tasks and other topics, which you can use to learn key concepts and address frequently asked questions. To access the Design Hubs:

- In the Xilinx Documentation Navigator, click the **Design Hubs View** tab.
- On the Xilinx website, see the [Design Hubs](#) page.



TIP: For quick access to information on different parts of the Vivado IDE, click the **Quick Help** button  in the window or dialog box. For detailed information on Tcl commands, enter the command followed by `-help` in the Tcl Console.

Board and Device Planning

Properly planning the device orientation on the board and assigning signals to specific pins can lead to dramatic improvements in overall system performance, power consumption, thermal performance, and design cycle times. Visualizing how the device interacts physically and logically with the printed circuit board (PCB) enables you to streamline the data flow through the device.

Failing to properly plan the I/O configuration can lead to decreased system performance and longer design closure times. Xilinx highly recommends that you consider I/O planning in conjunction with board planning.

For more information, see the following resources:

- [Vivado Design Suite User Guide: I/O and Clock Planning \(UG899\)](#)
- [Vivado Design Suite QuickTake Video: I/O Planning Overview](#)

PCB Layout Recommendations

The layout of the device on the board relative to other components with which it interacts can significantly impact the I/O planning.

Aligning with Physical Components on the PCB

The orientation of the device on the PCB should first be established. Consider the location of fixed PCB components, as well as internal device resources. For example, aligning the GT interfaces on the device package to be as close to the components with which they interface on the PCB will lead to shorter PCB trace lengths and fewer PCB vias.

A sketch of the PCB including the critical interfaces can often help determine the best orientation for the device on the PCB, as well as placement of the PCB components. After completion, the rest of the device I/O interface can be planned.

High-speed interfaces such as memory can benefit from having very short and direct connections with the PCB components with which they interface. These PCB traces often have to be matched length and not use PCB vias, if possible. In these cases, the package pins closest to the edge of the device are preferred in order to keep the connections short and to avoid routing out of the large matrix of BGA pins.

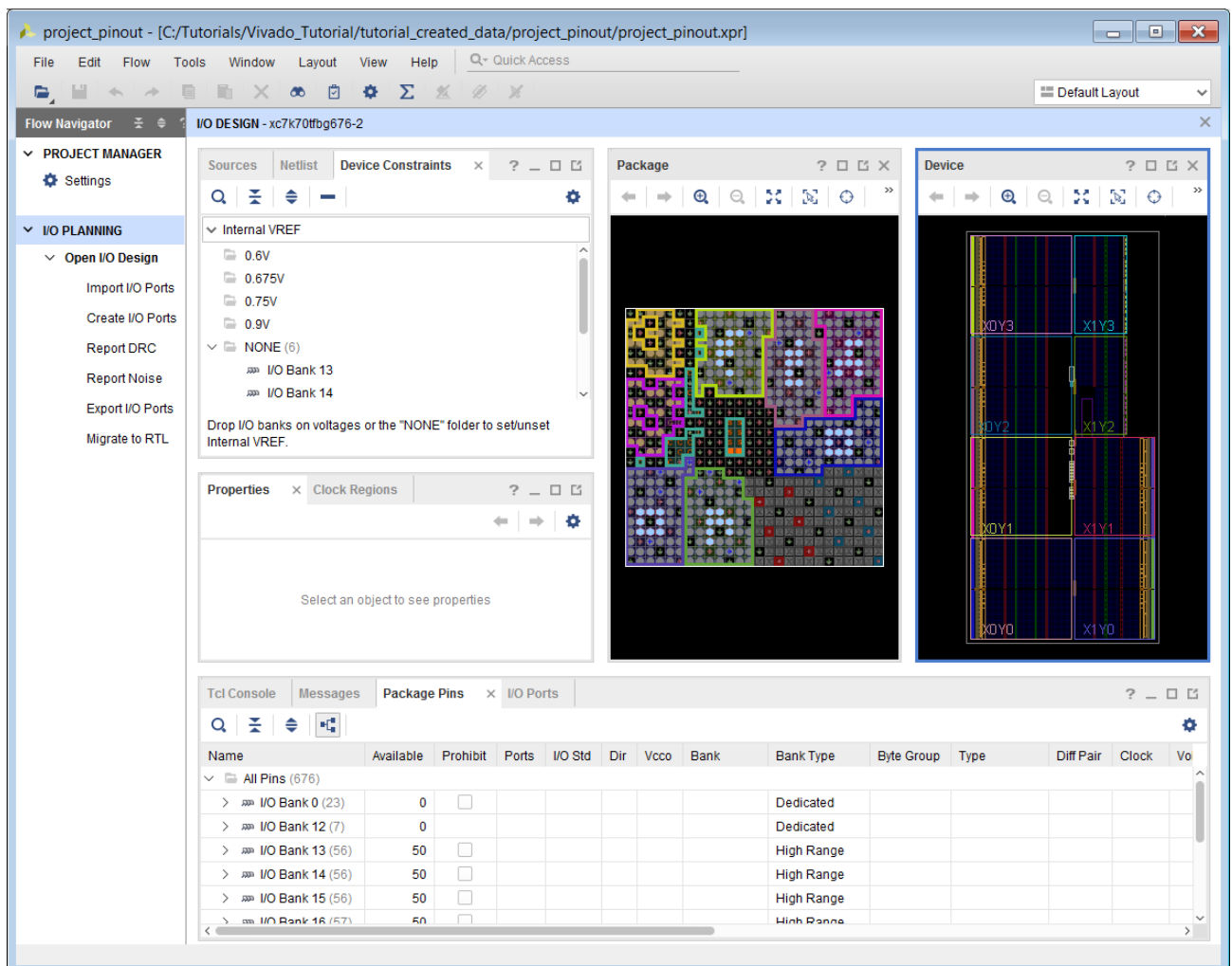
The I/O Planning view layout in the Vivado® IDE is useful in this stage for visualizing I/O connectivity relative to the physical device dimensions, showing both top-side and bottom-side views.



THERMAL TIP: For thermally-challenged designs, be aware of device placement in relation to other high-power components to minimize thermal coupling and maximize airflow. Avoid placement where the device is positioned in the exhaust of another high-power component or where board heating might negatively impact the operating temperature. Xilinx recommends thermal simulation to understand how the placement and environmental conditions can affect the junction temperature of the device.

The following figure shows the I/O Planning view layout.

Figure 4: I/O Planning View Layout



Power Distribution System

Board designers are faced with a unique task when designing a power distribution system (PDS) for a Xilinx® device. Most other large, dense integrated circuits (such as large microprocessors) come with very specific bypass capacitor requirements. Because these devices are designed only to implement specific tasks in their hardened silicon architecture, their power supply demands are fixed and fluctuate typically within a certain range.

Xilinx devices do not share this property. Devices can implement an almost infinite number of applications at user-determined frequencies, and in multiple clock domains.

For this reason, it is critical that you understand the power requirements of the design, which you can assess by completing a power estimation using the [Xilinx Power Estimator \(XPE\)](#) available from the Xilinx website. Also refer to the PCB Design Guide for your device to fully understand the PDS placement and generic decoupling requirements prior to a power estimation.

Key factors to consider during PDS design include:

- Selecting the right voltage regulators to meet the noise and current requirements based on power estimation.

Note: To enable and simplify your power design, Xilinx partners with key power vendors to design, build, document, and test reference designs that meet all power requirements. For more information, see the Power Delivery Solutions tab on the [Power](#) page of the Xilinx website.

- Consolidating power. For supported consolidation options in UltraScale™ devices, see this [link](#) in the *UltraScale Architecture PCB Design User Guide (UG583)*.



POWER TIP: Xilinx recommends adding a shunt resistor to allow the power on each rail to be monitored. Alternatively, you can use a PMBus-enabled regulator or current monitoring integrated circuit (IC).

- Setting up the Sysmon power supply (Vrefp and Vrefn pins).
- Running power distribution network (PDN) simulation. For UltraScale devices, use the recommended number of decoupling capacitors listed in the *UltraScale Architecture PCB Design User Guide (UG583)*, which are based on the assumptions listed in the guide. If the assumptions differ for your design, simulate your design to determine whether more or less decoupling is required. Running PDN simulations can help to confirm the exact amount of decoupling capacitors required to guarantee power supplies that are within the recommended operating range.

Note: See the *7 Series FPGAs PCB Design Guide (UG483)*, *UltraScale Architecture PCB Design User Guide (UG583)*, or *Zynq-7000 SoC PCB Design Guide (UG933)* to find the details for your device.

For more information on PDN simulation, see *Simulating FPGA Power Integrity Using S-Parameter Models (WP411)*.



POWER TIP: Xilinx recommends simulating your power supply design using the SIMPLIS simulator in SIMetrix/SIMPLIS to ensure your design is within the Xilinx recommended operating conditions. The majority of power vendors provide a limited version of SIMPLIS and supply the models to allow you to run this simulation. SIMPLIS is a third-party software used for transient and AC analysis of voltage regulators. For more information about simulating your power delivery, contact SIMPLIS or your preferred power delivery vendor.



POWER TIP: The Vivado tools `report_power` command can analyze power on a per regulator or voltage regulator module (VRM) basis to ensure the required current on each rail does not exceed the intended power delivery system.

Related Information

[Power Closure](#)

Thermal Solution Considerations

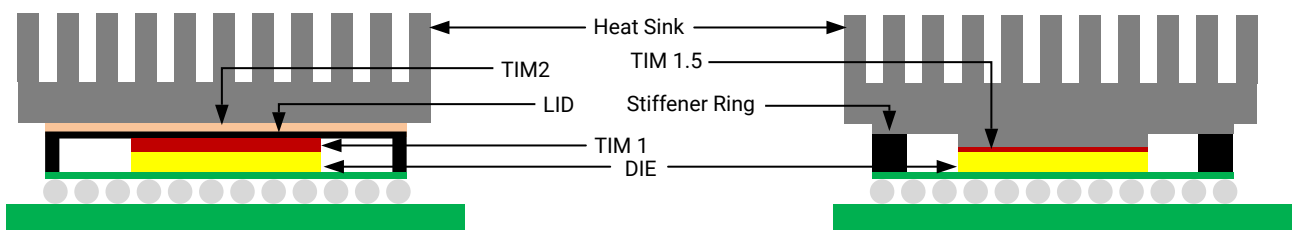
When considering the power estimation of a design, understanding the efficiency of the thermal solution is crucial. The lower the junction temperature, the lower the static power of a design.

Xilinx recommends using lidless packaging if it is available for your device. Lidless packaging offers a more efficient thermal solution and allows direct contact with the heat source, removing a thermal interface material (TIM) layer. Xilinx lidded and lidless parts have the same handling and manufacturing requirements. The following figure compares the heat sink application for a lidded and lidless device.



THERMAL TIP: Xilinx recommends between 20 and 50 pound-force per square inch (PSI) for the heat sink, which ensures the smallest bond line thickness (BLT), and recommends using 4-hole mounting to ensure even pressure for both lidded and lidless devices. For more information on lidless techniques, see *Mechanical and Thermal Design Guidelines for Lidless Flip-Chip Packages* ([XAPP1301](#)).

Figure 5: Heat Sink Example



X23524-112719

Xilinx also recommends thermal simulation to ensure that there is adequate margin and accurate power estimation. In the Xilinx Power Estimator (XPE), you have control over the following thermal settings:

- **Junction Temperature Tj:** You can override this setting to a desired junction temperature to match your thermal simulation. If you are not running a thermal simulation, set the junction temperature to the worst case.

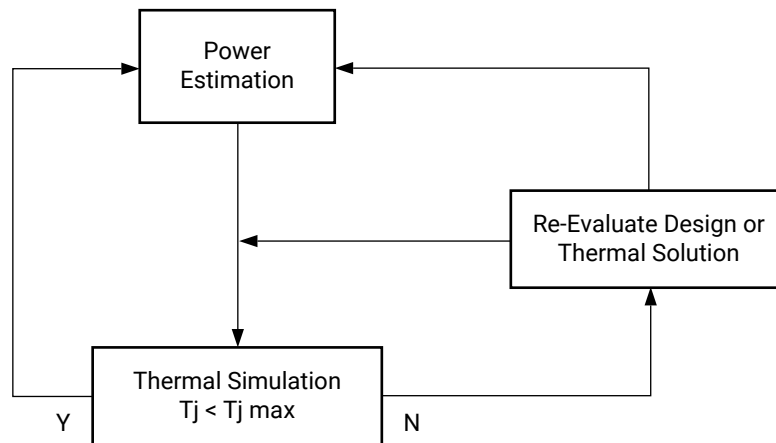
- Effective Θ_{JA} :** Describes the thermal efficiency of a thermal solution, the units are measured in degrees Celsius per watt ($^{\circ}\text{C}/\text{W}$). For example, an Θ_{JA} of $2.1^{\circ}\text{C}/\text{W}$ means that for every watt dissipated in the device, the junction temperature increases by 2.1°C . For a 10W design, the increase is 21°C above the ambient temperature.

Note: You can obtain the Θ_{JA} through thermal simulation using the following formula:

$$\Theta_{Ja} = (T_j - T_a) / \text{PowerDissipated}$$

The following figure shows the recommended flow for thermal validation.

Figure 6: Recommended Thermal Validation Flow



X23525-111319

After the junction temperature is within specification and sufficient margin is considered, the thermal solution is considered effective.



THERMAL TIP: Add the results of the power estimation and thermal simulation to the Vivado design constraints. You can use the following XDC constraints, which you can export from XPE using the Export option, as described in the Xilinx Power Estimator User Guide (UG440):

```

# Standard Constraints:
set_operating_conditions -process Maximum
set_operating_conditions -design_power_budget <value>
#If thermal simulation completed
set_operating_conditions -ambient_temp <value>
set_operating_conditions -thetaja <value>
#Else if no thermal simulation completed
set_operating_conditions -junction_temp <value>
  
```

PCB Design Considerations

The PCB should be designed considering the fastest signal interfacing with the device. These high-speed signals are extremely sensitive to trace geometry, vias, loss, and crosstalk. These aspects become even more prominent for multi-layer PCBs. For high-speed interfaces perform a signal integrity simulation. A board redesign with improved PCB material or altered trace geometries might be necessary to obtain the desired performance.

Xilinx recommends following these steps when designing your PCB:

1. Review the following device documentation:
 - PCB Design Guide for your device.
 - Board design guidelines in the transceiver user guide for your device.
2. Review memory IP and PCIe® design guidelines in the IP product guides.
3. Use the Vivado tools to validate your I/O planning:
 - Run simultaneous switching noise (SSN) analysis.
 - Run built-in DRCs.
 - Export I/O buffer information specification (IBIS) models.
4. Run signal integrity analysis as follows:
 - For gigabit transceivers (GTs), run Spice or IBIS-AMI simulations using channel parameters.
 - For lower performance interfaces, run IBIS simulation to check for issues with overshoot or undershoot.
5. Use the XPE with Process set to Maximum to generate an early estimate of the power consumption for the design.
6. Complete and adhere to the schematic checklist for your device.

Note: See the *7 Series Schematic Review Recommendations (XMP277)*, *Kintex UltraScale and Virtex UltraScale FPGAs Schematic Review Checklist (XTP344)*, or *UltraScale+ FPGAs and Zynq Ultrascale+ Devices Schematic Review Checklist (XTP427)*.
7. Use the XPE to generate a Xilinx design constraints (XDC) file, and import this file into the corresponding Vivado project. The XPE environment settings are translated to XDC constraints. The estimated total on-chip power becomes the design power budget for Vivado power analysis. For more information, see the *Vivado Design Suite User Guide: Power Analysis and Optimization (UG907)*.

Related Information

[Other Xilinx Documentation](#)

Device Power Aspects and System Dependencies

When planning the PCB, you must take power into consideration:

- The device and the user design create system power supply and heat dissipation requirements.

- Power supplies must be able to meet maximum power requirements and the device must remain within the recommended voltage and temperature operating conditions during operation. Power estimation and thermal modeling are required to ensure that the device stays within these limits.
- Plan for the consolidation of power rails and their impact on power domain switching.
- Although consolidation is possible, Xilinx recommends using full power management to give maximum flexibility where possible.

For these reasons, you must understand the power and cooling requirements of the device. These must be designed on the board.



POWER TIP: For a list of Xilinx partners and Xilinx-approved power delivery reference designs, see the [Power](#) page on the Xilinx website.

Power Supply Paths on Devices

Multiple power supplies are required to power a device and must be provided in a specific sequence. Consider the use of power monitoring or sequencing circuitry to provide the correct power-on sequence to the device as well as any additional active components on the board. More complex environments might benefit from the use of a microcontroller or system and power management bus such as SMBUS or PMBUS to control the power and reset process. Specific details regarding on/off sequencing can be found in the device data sheet. For more information on supply consolidation and topologies, see the *7 Series FPGAs PCB Design Guide (UG483)*, *UltraScale Architecture PCB Design User Guide (UG583)*, or *Zynq-7000 SoC PCB Design Guide (UG933)* depending on your device.

The separate sources provide the required power for the different device resources. This allows different resources to work at different voltage levels for increased performance or signal strength, while preserving a high immunity to noise and parasitic effects.

Power Types

A device goes through several power phases from power up to power down with varying power requirements.

Power-On

Power-on power is the transient spike current that occurs when power is first applied to the device. This current varies for each voltage supply and depends on the device construction, the ability of the power supply source to ramp up to the nominal voltage, and the device operating conditions, such as temperature and sequencing between the different supplies.

Spike currents are not a concern in modern device architectures when the proper power-on sequencing guidelines are followed.

Startup Power

Startup power is the power required during the initial bring-up and configuration of the device. This power generally occurs over a very short period of time and thus is not a concern for thermal dissipation. However, current requirements must still be met. In most cases, the active current of an operating design will be higher and thus no changes are necessary. However, for lower-power designs where active current can be low, a higher current requirement during this time might be necessary. XPE can be used to understand this requirement. When Process is set to Maximum, the current requirement for each voltage rail will be specified to either the operating current or the startup current, whichever is higher. XPE will display the current value in blue if the startup current is the higher value.

Static Power

Design static power (also called *standby power*) is the power supplied when the device is configured with your design and no activity is applied externally or generated internally. Static power represents the minimum continuous power that the supplies must provide while the design operates.

Static power is a function of junction temperature. Therefore, ensuring the ambient and thermal solution parameters are correctly modeled is critical to allow the power estimation tools accuracy report the static power.

Related Information

[Recommended Power Constraints](#)

Dynamic Power

Dynamic power is the power required when the device is running your application and undergoing switching activity as clocks and datapaths toggle between High and Low logic values. Dynamic power is calculated based on the average switching activity of device circuits over a period of time. Total power includes static power plus dynamic power.

Environmental Factors Impacting Power

In addition to the design itself, environmental factors affect power. These factors influence the voltage and the junction temperature of the device, which impacts the power dissipation. For more information, see this [link](#) in the *Vivado Design Suite User Guide: Power Analysis and Optimization* (UG907).



THERMAL TIP: The -2LI and -2LE UltraScale+™ devices allow a temperature excursion of up to 110°C for a defined period of time, which enables a reduction in the thermal solution cost. For more information, see [Extending the Thermal Solution by Utilizing Excursion Temperatures \(WP517\)](#).

Power Rail Consolidation Impacting Power

To take advantage of the power management switching of power domains, your design must keep some discrete power rails. This allows individual rails to be powered off with the power domain switching logic at the cost of using additional voltage regulators or regulator outputs. For more information, see this [link](#) in the *UltraScale Architecture PCB Design User Guide (UG583)*.



TIP: The Vivado tools also support power rail constraints. For information, see this [link](#) in the *Vivado Design Suite User Guide: Power Analysis and Optimization (UG907)*.

Power Models Accuracy

The accuracy of the characterization data embedded in the tools evolves over time to reflect the device availability or manufacturing process maturity. For details, see this [link](#) in the *Vivado Design Suite User Guide: Power Analysis and Optimization (UG907)*.



POWER TIP: Power estimation is only as accurate as the data entered. Xilinx recommends conducting a thorough estimation and using the results of this estimation as well as the thermal evaluation as a design constraint.

Device Power and the Overall System Design Process

From project conception to completion, various aspects of the design process affect power. For details, see this [link](#) in the *Vivado Design Suite User Guide: Power Analysis and Optimization (UG907)*.



POWER TIP: During the design process, you can compare the total power of the design to the power budget using the `set_operating_conditions -design_power_budget <Power in Watts>` XDC constraint. If the power budget is exceeded, early intervention is the easiest way to correct design power.

Worst Case Power Analysis Using Xilinx Power Estimator (XPE)

Xilinx recommends designing the board for worst-case power. For details, see this [link](#) in the *Vivado Design Suite User Guide: Power Analysis and Optimization (UG907)*.

Clock Resource Planning and Assignment

Xilinx recommends that you select clocking resources as one of the first steps of your design, well before pinout selection. Your clocking selections can dictate a particular pinout and can also direct logic placement for that logic, especially for stacked silicon interconnect (SSI) technology devices. Proper clocking selections can yield superior results. Consider the following:

- Constraint creation, particularly in large devices with high utilization in conjunction with clock planning.
- Manual placement of clocking resources if needed for design closure.
- Device-specific functionality that might require up-front planning to avoid issues and take advantage of device features. For information on 7 series features, see this [link](#) and this [link](#) in the *7 Series FPGAs Clocking Resources User Guide (UG472)*. For information on UltraScale device features, see this [link](#) in the *UltraScale Architecture Clocking Resources User Guide (UG572)*.

Related Information

[Clocking Guidelines](#)

[Auto-Pipelining Considerations](#)

[SLR Crossing for Wide Buses](#)

I/O Planning Design Flows

The Vivado IDE allows you to interactively explore, visualize, assign, and validate the I/O ports and clock logic in your design. The environment ensures correct-by-construction I/O assignment. It also provides visualization of the external package pins in correlation with the internal die pads.

You can visualize the data flow through the device and properly plan I/Os from both an external and internal perspective. After the I/Os are assigned and configured through the Vivado IDE, constraints are then automatically created for the implementation tools.

For more information on Vivado Design Suite I/O and clock planning capabilities, see the following resources:

- *Vivado Design Suite User Guide: I/O and Clock Planning (UG899)*
- [Vivado Design Suite QuickTake Video: I/O Planning Overview](#)

Types of Vivado Design Suite Projects for I/O Planning

You can perform I/O planning with either of the following types of projects:

- **I/O planning project** : An I/O planning project is an easy entry point that allows you to specify select I/O constraints and generate a top-level RTL file from the defined pins.
- **RTL project** : An RTL project allows synthesis and implementation, which enables more comprehensive design rule checks (DRCs). An RTL project also allows generation of IP cores, which is important for memory interface pinout planning and any cores using GTs.



TIP: You can also start by using an I/O planning project and migrate to an RTL project later.

You can run more comprehensive DRCs on a post-synthesis netlist. The same is true after implementation and bitstream generation. Therefore, Xilinx recommends using a skeleton design that includes clocking components and some basic logic to exercise the DRCs. This builds confidence that the pin definition for the board will not have issues later.

The recommended sign-off process is to run the RTL project through to bitstream generation to exercise all the DRCs. However, not all design cycles allow enough time for this process. Often the I/O configuration must be defined before you have synthesizable RTL. Although the Vivado tools enable pre-RTL I/O planning, the level of DRCs performed are fairly basic. Alternatively, you can use a dummy top-level design with I/O standards and pin assignments to help perform DRCs related to banking rules.

Pre-RTL I/O Planning

If your design cycle forces you to define the I/O configuration before you have a synthesized netlist, take great care to ensure adherence to all relevant rules. The Vivado tools include a Pin Planning Project environment that allows you to import I/O definitions using a CSV or XDC format file. You can also create a dummy RTL with just the port directions defined. Availability of port direction makes SSN analysis more accurate as input and output signals have different contributions to SSN.

I/O ports can also be created and configured interactively. Basic I/O bank DRC rules are provided.

See the *7 Series FPGAs PCB Design Guide* ([UG483](#)), *UltraScale Architecture PCB Design User Guide* ([UG583](#)), or *Zynq-7000 SoC PCB Design Guide* ([UG933](#)) to ensure proper I/O configuration for your device. For more information, see this [link](#) in the *Vivado Design Suite User Guide: I/O and Clock Planning* ([UG899](#)).

Netlist-Based I/O Planning

The recommended time in the design cycle to assign I/Os and clock logic constraints is after the design has been synthesized. The clock logic paths are established in the netlist for constraint assignment purposes. The I/O and clock logic DRCs are also more comprehensive.

See the *7 Series FPGAs PCB Design Guide (UG483)*, *UltraScale Architecture PCB Design User Guide (UG583)*, or *Zynq-7000 SoC PCB Design Guide (UG933)* to ensure proper I/O configuration for your device. For more information, see this [link](#) in the *Vivado Design Suite User Guide: I/O and Clock Planning (UG899)*.

Identifying Pin Compatible Devices

It is often difficult to predict the final device size for any given design during initial planning. Logic can be added or removed during the course of the design cycle, which can result in the need to change the device size. The Vivado tools enable you to identify alternate devices to ensure that the I/O pin configuration defined is compatible across all selected devices, as long as the package is the same. For information, see this [link](#) in the *Vivado Design Suite User Guide: I/O and Clock Planning (UG899)*.



IMPORTANT! *The device must be in the same package.*



TIP: *To migrate your design with reduced risk, carefully plan the following at the beginning of the design process: device selection, pinout selection, and design criteria. Take the following into account when migrating to a larger or smaller device in the same package: pinout, clocking, and resource management.*

Pin Assignment

Good pinout selection leads to good design logic placement, shorter routes, reduced power consumption, and improved performance. Good pinout selection is especially important for large devices, because a pinout that is spread out can cause related signals to span longer distances. For more information, see this [link](#) in the *Vivado Design Suite User Guide: I/O and Clock Planning (UG899)*.

Using Xilinx Tools in Pinout Selection

Xilinx tools assist in interactive design planning and pin selection. These tools are only as effective as the information you provide them. Tools such as the Vivado I/O Planner can assist pinout efforts. These tools can graphically display the I/O placement, show relationships among clocks and I/O components, and provide DRCs to analyze pin selection.

If a design version is available, a quick top-level floorplan can be created to analyze the data flow through the device. For more information, see the *Vivado Design Suite User Guide: Design Analysis and Closure Techniques (UG906)*.

Required Information

For the tools to work effectively, you must provide as much information about the I/O characteristics and topologies as possible. You must specify the electrical characteristics, including the I/O standard, drive, slew, and direction of the I/O.

You must also take into account all other relevant information, including clock topology and timing constraints. Clocking choices in particular can have a significant influence on pinout selection, and vice versa.

For IP that have I/O requirements, such as transceivers, PCIe, and memory interfaces, you must configure the IP prior to completing I/O pin assignment. For more information on specifying the electrical characteristics for an I/O, see this [link](#) in the *Vivado Design Suite User Guide: I/O and Clock Planning* (UG899).

Related Information

[Clocking Guidelines](#)

Pinout Selection

Xilinx recommends careful pinout selection for some specific signals as discussed below.

General Pinout Selection Guidelines

Following are general guidelines:

- Group the same interface data, address, and control pins into the same bank. If you cannot group these components into the same bank, group them into adjacent banks.

Note: For SSI technology devices, adjacent banks must also be located within the same super logic region (SLR).

- Place the following interface control signals in the middle of the data buses they control: clocking, enables, resets, and strobes.
- Place very high fanout, design-wide control signals towards the center of the device.

Note: For SSI technology devices, place the signals in the SLR located in the middle of the SLR components they drive.

Configuration Pins

To design an efficient system, you must choose the device configuration mode that best matches the system requirements. Factors to consider include:

- Using dedicated vs. dual purpose configuration pins.

Each configuration mode dedicates certain device pins and can temporarily use other multi-function pins during configuration only. These multi-function pins are then released for general use when configuration is completed.

- Using configuration mode to place voltage restrictions on some device I/O banks.
- Choosing suitable terminations for different configuration pins.
- Using the recommended values of pull-up or pull-down resistors for configuration pins.



RECOMMENDED: *Even though configuration clocks are slow speed, perform signal integrity analysis on the board to ensure clean signals.*

There are several configuration options. Although the options are flexible, there is often an optimal solution for each system. Consider the following when choosing the best configuration option:

- Setup
- Speed
- Cost
- Complexity

For more information on device configuration options, see *Vivado Design Suite User Guide: Programming and Debugging* ([UG908](#)).

Related Information

[Configuration](#)

Memory Interfaces

Additional I/O pin planning steps are required when using Xilinx Memory IP. After the IP is customized, assign the top-level IP ports to physical package pins in either the elaborated or synthesized design in the Vivado IDE. All of the ports associated with each Memory IP are grouped together into an I/O Port Interface for easier identification and assignment. A Memory Bank/Byte Planner is provided to assist you with assigning Memory I/O pin groups to byte lanes on the physical device pins. For more information, see this [link](#) in the *Vivado Design Suite User Guide: I/O and Clock Planning* ([UG899](#)).

Take care when assigning memory interfaces and try to limit congestion as much as possible, especially with devices that have a center I/O column. Bunching memory interfaces together can create routing bottlenecks across the device. The *Zynq-7000 SoC and 7 series Devices Memory Interface Solutions* ([UG586](#)) and the *UltraScale Architecture-Based FPGAs Memory IP LogiCORE IP Product Guide* ([PG150](#)) contain design and pinout guidelines. Be sure that you follow the trace length match recommendations in these guides, verify that the correct termination is used, and validate the pinout in by running the DRCs after memory IP I/O assignment. For more information on memory interface signal termination and routing guidelines, see the *UltraScale Architecture PCB Design User Guide* ([UG583](#)).

Gigabit Transceivers (GTs)

Gigabit transceivers (GTs) have specific pinout requirements, and you must consider the following:

- Sharing of reference clocks
- Sharing of PLLs within a quad
- Placement of hard blocks, such as PCIe, and their proximity to transceivers
- In SSI technology devices, crossing of SLR boundaries

Xilinx recommends that you use the GT wizard to generate the core. Alternatively, you can use the Xilinx IP core for the protocol. For pinout recommendations, see the related product guide.

For clock resource balancing, the Vivado placer attempts to constrain loads clocked by GT output clocks (TXOUTCLK or RXOUTCLK) next to the GTs sourcing the clocks. For SSI technology devices, if the GTs are located in the clock regions adjacent to another SLR, the routing resources required for signals entering or exiting SLLs have to compete with the routing resources required by the GT output clock loads. Therefore, GTs located in clock regions next to SLR crossings might reduce the available routing connections to and from the SLL crossings available in those clock regions.

High Speed I/O

HP (high-performance) and HR (high-range) banks have difference in the speed with which they can transmit and receive signals. Depending upon the I/O speed you need, choose between HP or HR banks.

Internal VREF and DCI Cascade Constraints

Based on the settings for DCI Cascade and Internal VREF, you can free up pins to be used for regular I/Os. These settings also ensure that related DRC checks are run to validate the legality of the constraints. For more information, see either the *7 Series FPGAs SelectIO Resources User Guide* ([UG471](#)) or the *UltraScale Architecture SelectIO Resources User Guide* ([UG571](#)), depending on your device.

Interface Bandwidth Validation

Create small connectivity designs to validate each interface on the device. These small designs exercise only the specific hardware interface, which enables the following:

- Full DRC checks on pinout, clocking, and timing
- Hardware test design when the board is returned
- Rapid implementation through the Vivado tools, providing the fastest way to debug the interface

There are multiple options to assist in generating test data for these interfaces. For some of the interface IP cores, the Vivado tools can provide the test designs:

- IBERT for SerDes
- Example design within IP cores



TIP: *If a test design does not exist, consider using AXI traffic generators.*

You might need to create a separate design for a system-level test in a production environment. Usually, this is a single design that includes tested interfaces and optionally includes processors. You can construct this design using the small connectivity designs to take advantage of design reuse. Although this design is *not* required early in the flow, it can enable better DRC checks and early software development, and you can quickly create it using the Vivado IP integrator.

Designing with SSI Devices

SSI Pinout Considerations

When planning pinouts for components that are located in a particular SLR, place the pins into the same SLR. For example, when using the device DNA information as a part of an external interface, place the pins for that interface in the master SLR in which the DNA_PORT exists. Additional considerations include the following:

- Group all pins of a particular interface into the same SLR.
- For signals driving components in multiple SLRs, place those signals in the middle SLR.
- Balance CCIO or CMT components across SLRs.
- Reduce SLR crossings.

Super Logic Region (SLR)

A super logic region (SLR) is a single device die slice contained in an SSI technology device. Each SLR contains a subset of device resources, such as CLBs, block RAMs, DSP tiles, and GTs, with a similar structure to non-SSI devices.

Multiple SLR components are stacked vertically and connected through an interposer to create an SSI technology device. The bottom SLR is SLR0, and subsequent SLR components are named incrementally as they ascend vertically. For example, the XC7V2000T device includes four SLR components. The bottom SLR is SLR0, the SLR directly above SLR0 is SLR1, the SLR directly above SLR1 is SLR2, and the top SLR is SLR3.

Note: The Xilinx tools clearly identify SLR components in the graphical user interface (GUI) and in reports.

SLR Nomenclature

Understanding SLR nomenclature for your target device is important in:

- Pin selection
- Floorplanning
- Analyzing timing and other reports
- Identifying where logic exists and where that logic is sourced or destined

You can use the Vivado Tcl command `get_slrs` to get specific information about SLRs for a particular device. For example, use the following commands:

- `llength [get_slrs]` to obtain the number of SLRs in the device
- `get_slrs -of_objects [get_cells my_cell]` to get the SLR in which `my_cell` is placed

Master Super Logic Region

Every SSI technology device has a single master SLR. The master SLR contains the primary configuration logic that initiates configuration of the device and all other SLR components. The master SLR contains the circuitry that is used for configuration, DNA_PORT, and EFUSE_USER. When using these components, the place and route tools can assign associated pins and logic to the appropriate SLR. In general, no additional intervention is required.



TIP: To query which SLR is the master SLR in the Vivado Design Suite, you can enter the `get_slrs -filter IS_MASTER` Tcl command.

Silicon Interposer

The silicon interposer is a passive layer in the SSI technology device, which routes the following between SLR components:

- Configuration
- Global clocking
- General interconnect

Super Long Line (SLL) Routes

Super Long Line (SLL) routes connect signals from one SLR to another inside the device.



TIP: To determine the number of available SLLs between SLRs, use SLR properties. For example:

```
get_property NUM_TOP_SLLS [get_slrs SLR0]
get_property NUM_BOT_SLLS [get_slrs SLR1]
```

Propagation Limitations



TIP: For high-speed propagation across SLRs, be sure to register signals that cross SLR boundaries.

SLL signals are the only data connections between SLR components.

The following do not propagate across SLR components:

- Carry chains
- DSP cascades
- Block RAM address cascades
- Other dedicated connections, such as DCI cascades and block RAM cascades

The tools normally take this limit on propagation into account. To ensure that designs route properly and meet your design goals, you must also take this limit into account when you:

- Build a very long DSP cascade and manually place such logic near SLR boundaries
- Specify a pinout for the design

SLR Utilization Considerations

The Vivado implementation tools use a special algorithm to partition logic into multiple SLRs. For challenging designs, you can improve timing closure for designs that target SSI technology devices using the following guidelines.

To improve timing closure and compile times, you can use Pblocks to assign logic to each SLR and validate that individual SLRs do not have excessive utilization across all fabric resource types. For example, a design with block RAM utilization of 70% might cause issues with timing closure if the block RAM resources are not balanced across SLRs and one SLR is using over 85% block RAM.



TIP: You can define SLR Pblocks by specifying a complete SLR (e.g., `resize_pblock pblock_SLR0 -add SLR0`).

The following example utilization report for a vu160 shows that the overall block RAM utilization is 56% with 59% in SLR0, 40% in SLR1, and 58% in SLR2. The block RAM utilization is evenly distributed across SLRs with reasonable utilization in each SLR, which allows the Vivado implementation commands more flexibility to meet timing.

Figure 7: Block RAM Section in Utilization Report

```

3. BLOCKRAM
-----
+-----+-----+-----+-----+-----+
| Site Type | Used | Fixed | Available | Util% |
+-----+-----+-----+-----+-----+
| Block RAM Tile | 1843 | 0 | 3276 | 56.26 |
|   RAMB36/FIFO* | 1820 | 2 | 3276 | 55.56 |
|   FIFO36E2 only | 88 | | | |
|   RAMB36E2 only | 1732 | | | |
|   RAMB18 | 46 | 8 | 6552 | 0.70 |
|   RAMB18E2 only | 46 | | | |
+-----+-----+-----+-----+-----+
    
```


Figure 8: SLR Section in Utilization Report

14. SLR CLB Logic and Dedicated Block Utilization

Site Type	SLR0	SLR1	SLR2	SLR0 %	SLR1 %	SLR2 %
CLB	22361	40858	42493	61.70	91.28	94.94
CLBL	17061	31936	33311	60.76	90.99	94.90
CLBM	5300	8922	9182	64.95	92.36	95.05
CLB LUTs	104506	196677	236523	36.05	54.93	66.05
LUT as Logic	104482	194364	235584	36.04	54.28	65.79
using O5 output only	268	913	789	0.09	0.25	0.22
using O6 output only	101383	180318	219746	34.97	50.36	61.37
using O5 and O6	2831	13133	15049	0.98	3.67	4.20
LUT as Memory	24	2313	939	0.04	2.99	1.22
LUT as Distributed RAM	24	1316	136	0.04	1.70	0.18
using O5 output only	0	0	0	0.00	0.00	0.00
using O6 output only	8	48	64	0.01	0.06	0.08
using O5 and O6	16	1268	72	0.02	1.64	0.09
LUT as Shift Register	0	997	803	0.00	1.29	1.04
CLB Registers	191575	330568	473777	33.04	46.16	66.16
CARRY8	901	1715	3816	2.49	3.83	8.53
F7 Muxes	1725	10631	4762	1.19	5.94	2.66
F8 Muxes	216	2891	366	0.30	3.23	0.41
F9 Muxes	0	0	0	0.00	0.00	0.00
Block RAM Tile	599	512	732	59.42	40.63	58.10
RAMB36/FIFO	598	501	721	59.33	39.76	57.22
RAMB36E2 only	598	472	662	59.33	37.46	52.54
RAMB18	2	22	22	0.10	0.87	0.87
RAMB18E2 only	2	22	22	0.10	0.87	0.87
URAM	0	0	0	0.00	0.00	0.00
DSPs	1	2	12	0.21	0.33	2.00
PLL	0	0	0	0.00	0.00	0.00
MMCM	0	0	0	0.00	0.00	0.00
Unique Control Sets	1125	3482	8567	1.55	3.89	9.57

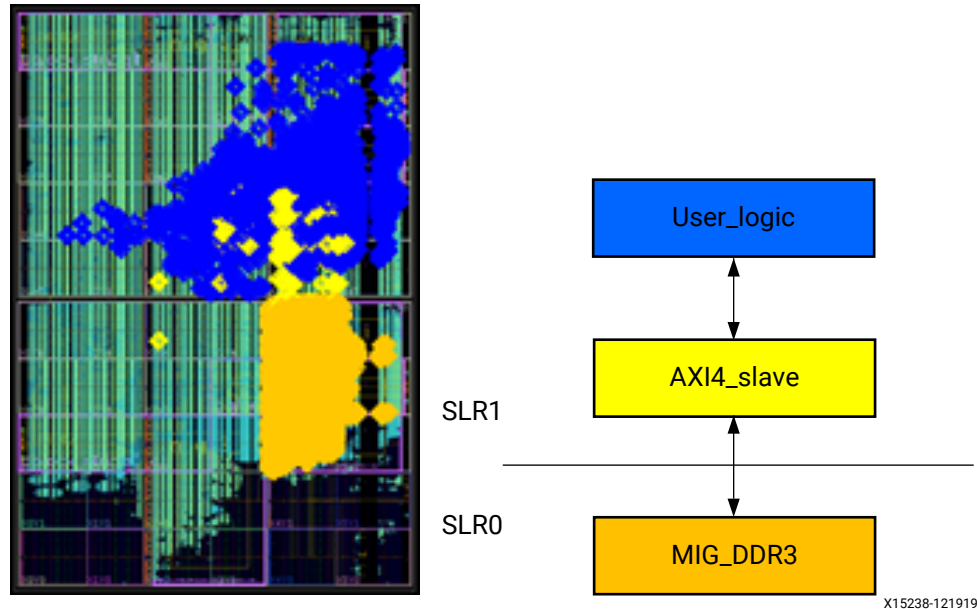
Xilinx recommends assigning block RAM and DSP groups to SLR Pblocks to minimize SLR crossings of shared signals. For example, an address bus that fans out to a group of block RAMs that are spread out over multiple SLRs can make timing closure more difficult to achieve, because the SLR crossing incurs additional delay for the timing critical signals.

Device resource location or user I/O selection anchors IP to SLRs, for example, GT, ILKN, PCIe, and CMAC dedicated block or memory interface controllers. Xilinx recommends the following:

- Pay special attention to dedicated block location and pinout selection to avoid data flow crossing SLR boundaries multiple times.
- Keep tightly interconnected modules and IP within the same SLR. If that is not possible, you can add pipeline registers to allow the placer more flexibility to find a good solution despite the SLR crossing between logic groups.
- Keep critical logic within the same SLR. By ensuring that main modules are properly pipelined at their interfaces, the placer is more likely to find SLR partitions with flip-flop to flip-flop SLR crossings.

In the following figure, a memory interface that is constrained to SLR0 needs to drive user logic in SLR1. An AXI4-Lite slave interface connects to the memory IP backend, and the well-defined boundary between the memory IP and the AXI4-Lite slave interface provides a good transition from SLR0 to SLR1.

Figure 9: Memory Interface in SLR0 Driving User Logic in SLR1



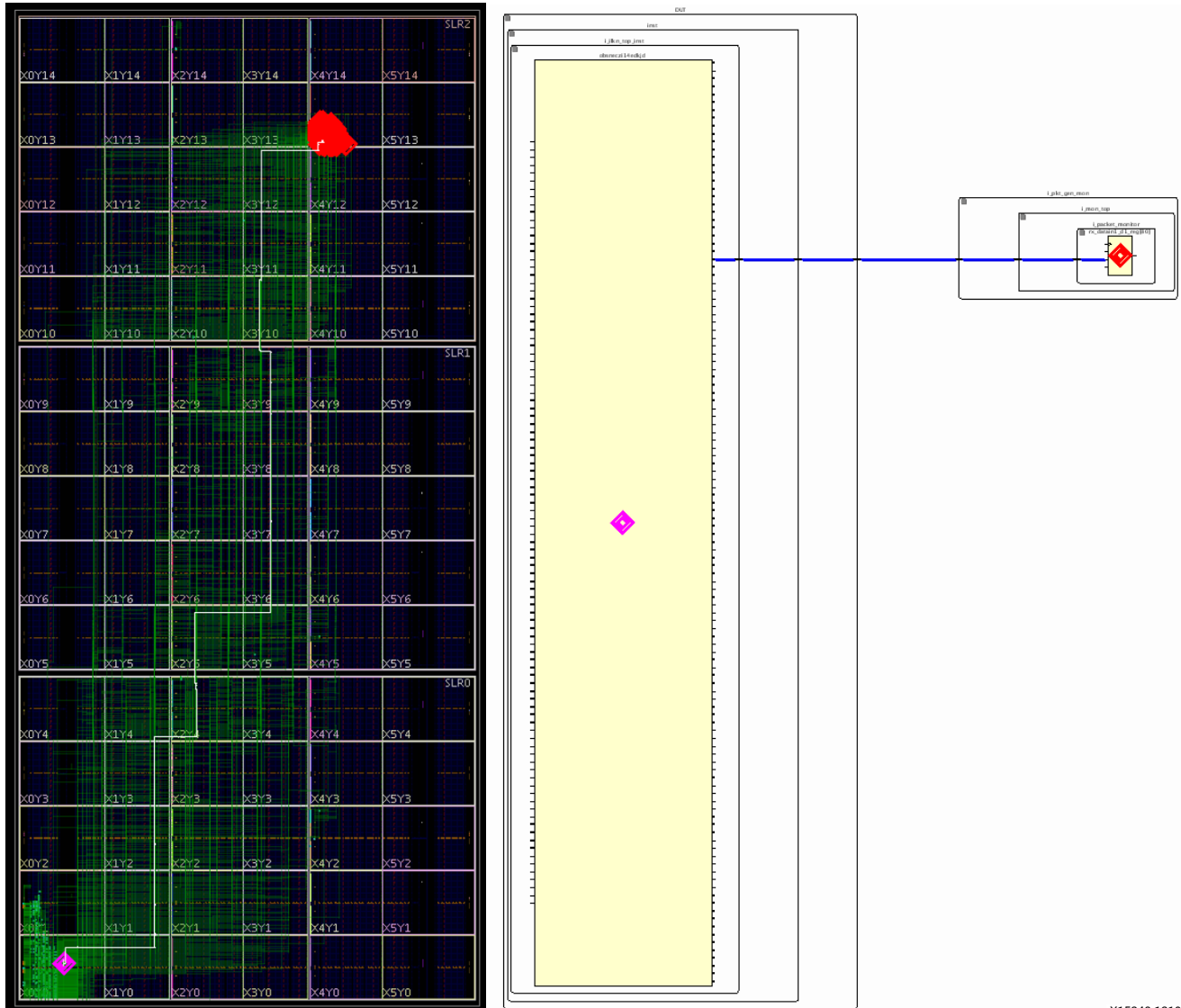
X15238-121919

SLR Crossing for Wide Buses

When data flow requirements require that wide buses cross SLRs, use pipelining strategies to improve timing closure and alleviate routing congestion of long resources. For wide buses operating above 250 MHz, Xilinx recommends using at least three pipeline stages to cross an SLR: one at the top, one at the bottom, and one in the middle of the SLR. Additional pipeline stages might be required for very high performance buses or when traversing horizontal as well as vertical distances.

The following figure illustrates a worst case crossing for a vu190-2 device. This example starts at an Interlaken dedicated block in the bottom left of SLR0 to a packet monitor block assigned to the top right of SLR2. Without pipeline registers for the data bus to and from the packet monitor, the design misses the 300 MHz timing requirement by a wide margin.

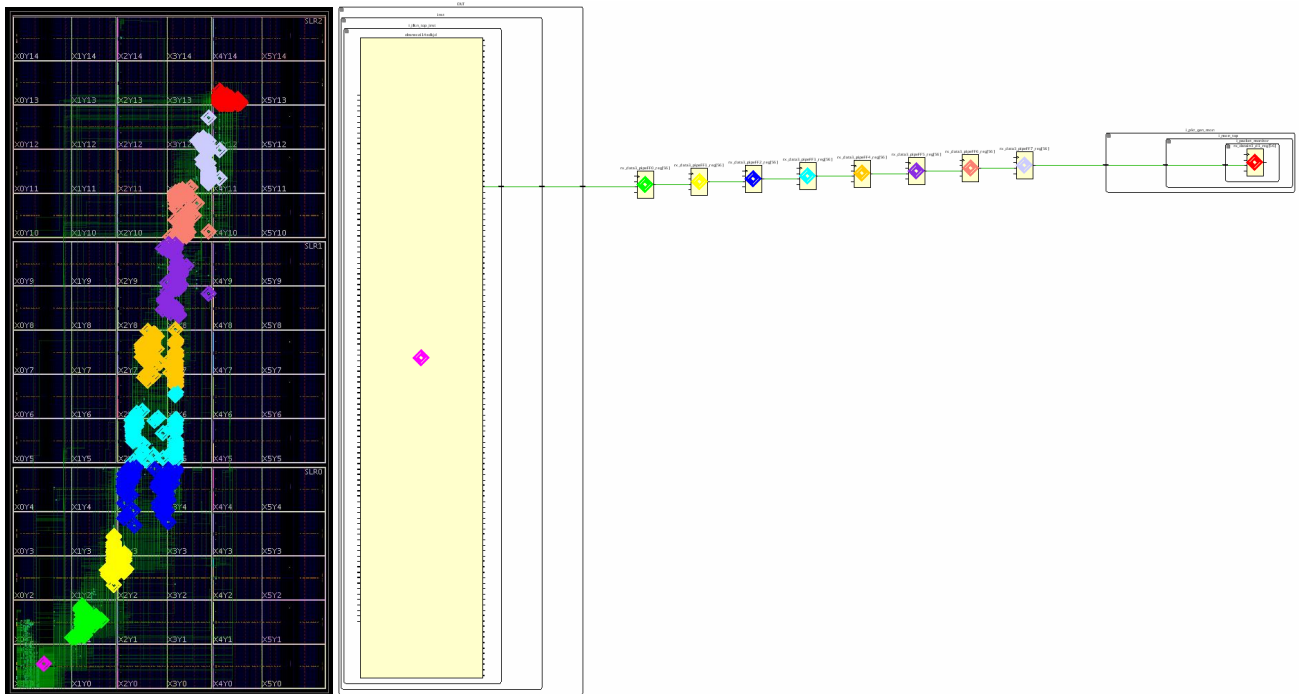
Figure 10: Data Path Crossing SLR without Pipeline Flip-Flop



X15240-121919

However, adding seven pipeline stages to aid in the traversal from SLR0 to SLR2 allows the design to meet timing. It also reduces the use of vertical and horizontal long routing resources, as shown in the following figure.

Figure 11: Data Path Crossing SLR with Pipeline Flip-Flop Added



X15239-110415



TIP: Use the AXI Register Slice IP or your custom auto-pipelining IP to close timing on wide buses across SLRs.

Related Information

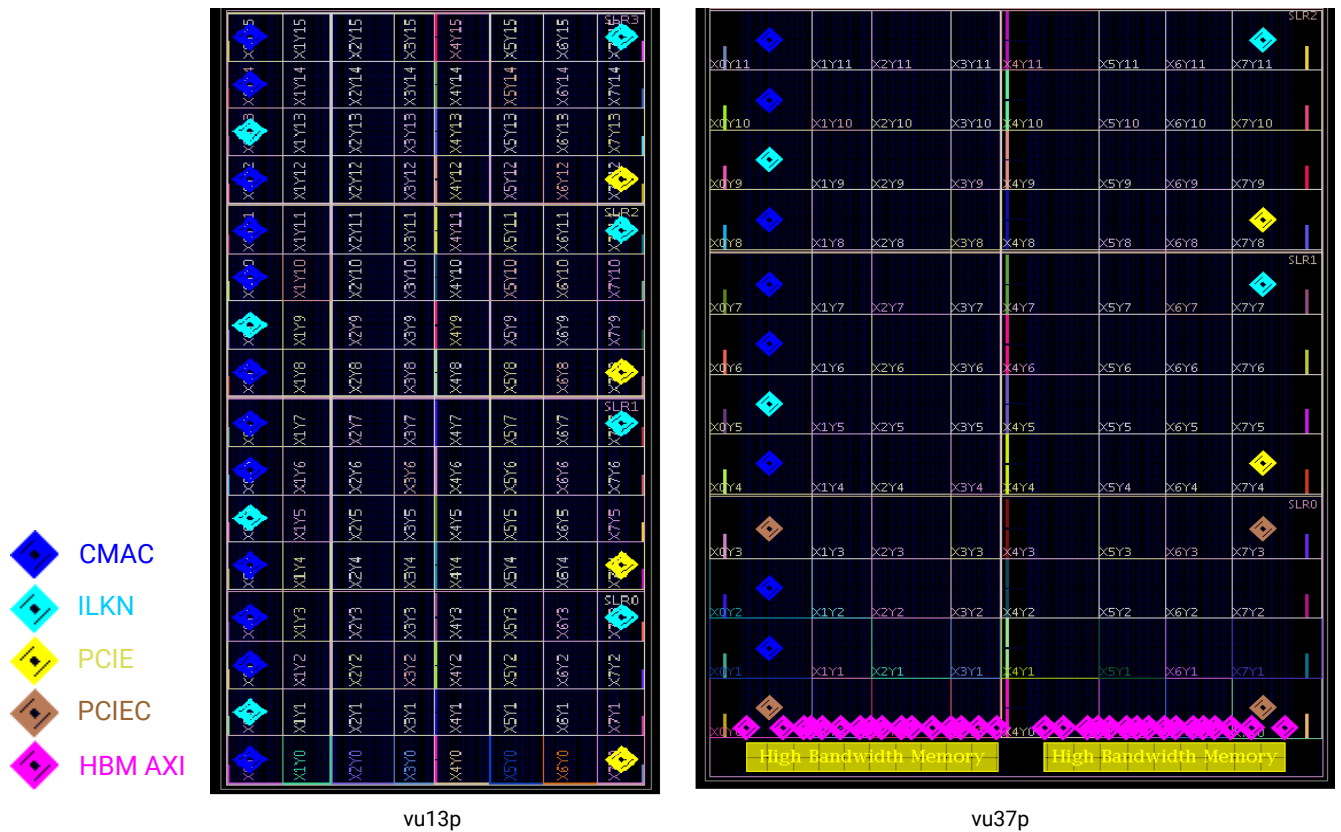
[Auto-Pipelining Considerations](#)

Designing with HBM Devices

Virtex® UltraScale+™ HBM devices incorporate 4 GB high-bandwidth memory (HBM) stacks adjacent to the device die. Using SSI technology, the device communicates to the HBM stacks through memory controllers that connect through the silicon interposer at the bottom of the device. Each Virtex UltraScale+ HBM device contains one or two 4 GB HBM stacks, resulting in up to 8 GB of HBM per device. The device includes 32 HBM AXI interfaces used to communicate with the HBM. The flexible addressing feature that is provided by a built-in switch allows for any of the 32 HBM AXI interface to access any memory address on either one or both of the HBM stacks. This flexible connection between the device and the HBM stacks is helpful for floorplanning and timing closure.

The following figure shows the Virtex UltraScale+ HBM vu37p device adjacent to a Virtex UltraScale+ vu13p device. In the VU37P device, the bottom two SLRs of the VU13P device are replaced by the HBM stacks (SLR0 in the vu13p device) and an SLR that contains the 32 HBM AXI interfaces (SLR1 in the vu13p device). The top two SLRs of the vu13p and vu37p device are identical.

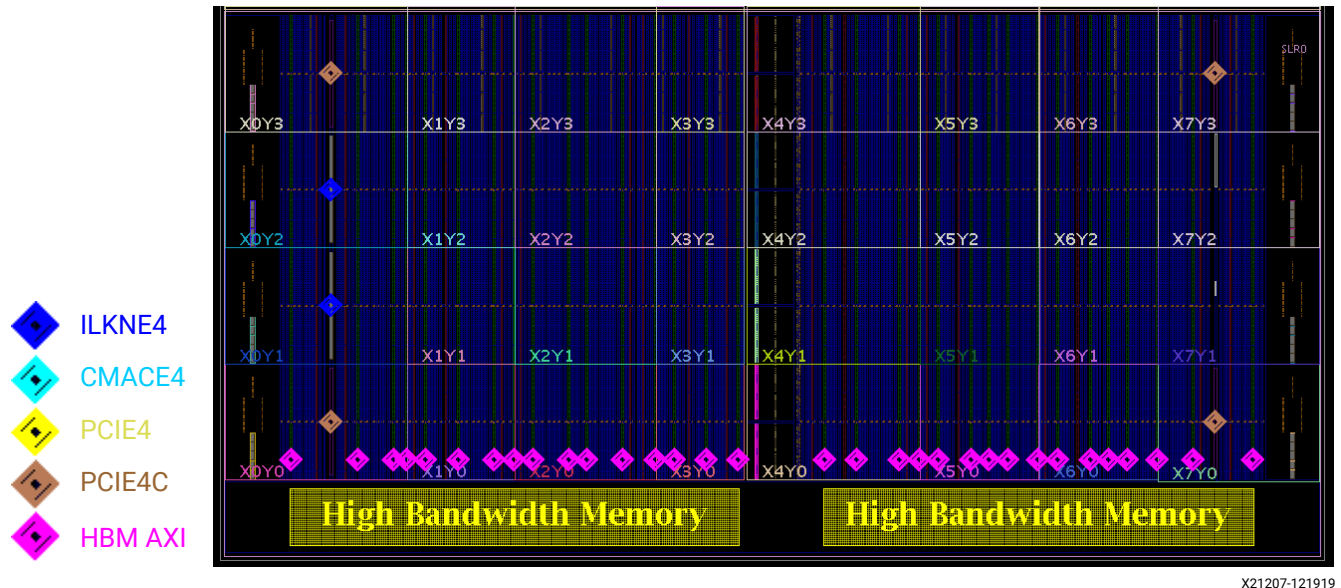
Figure 12: Device View of the vu13p and vu37p



X21195-121919

In the vu37p device, the SLR0 contains 4 PCIE4C sites, 2 ILKNE4 sites, and the 32 HBM AXI interfaces. The 4 PCIE4C sites in the Virtex UltraScale+ HBM SLR0 are unique because they allow for the Cache Coherent Interconnect for Accelerators (CCIX) protocol using PCIe Gen3 x 16 when V_{CCINT} is at 0.72V.

Figure 13: SLR0 of a Virtex UltraScale+ HBM vu37p Device



X21207-121919

Placement Considerations When Using HBM Devices

Pipelining Considerations for Crossing SLRs

The pipeline considerations for crossing SLRs in Virtex UltraScale+ HBM devices are the same as for other UltraScale and Virtex UltraScale+ SSI technology devices.

Paths from fabric logic in SLR2 to the HBM AXI Interfaces in SLR0 often require five or more pipeline stages to meet timing. Thoughtful design planning of Virtex UltraScale+ HBM devices can reduce the need for additional pipeline stages and reduce routing congestion. The following figure shows an example of SLR crossings to the HBM AXI Interfaces from SLR2.



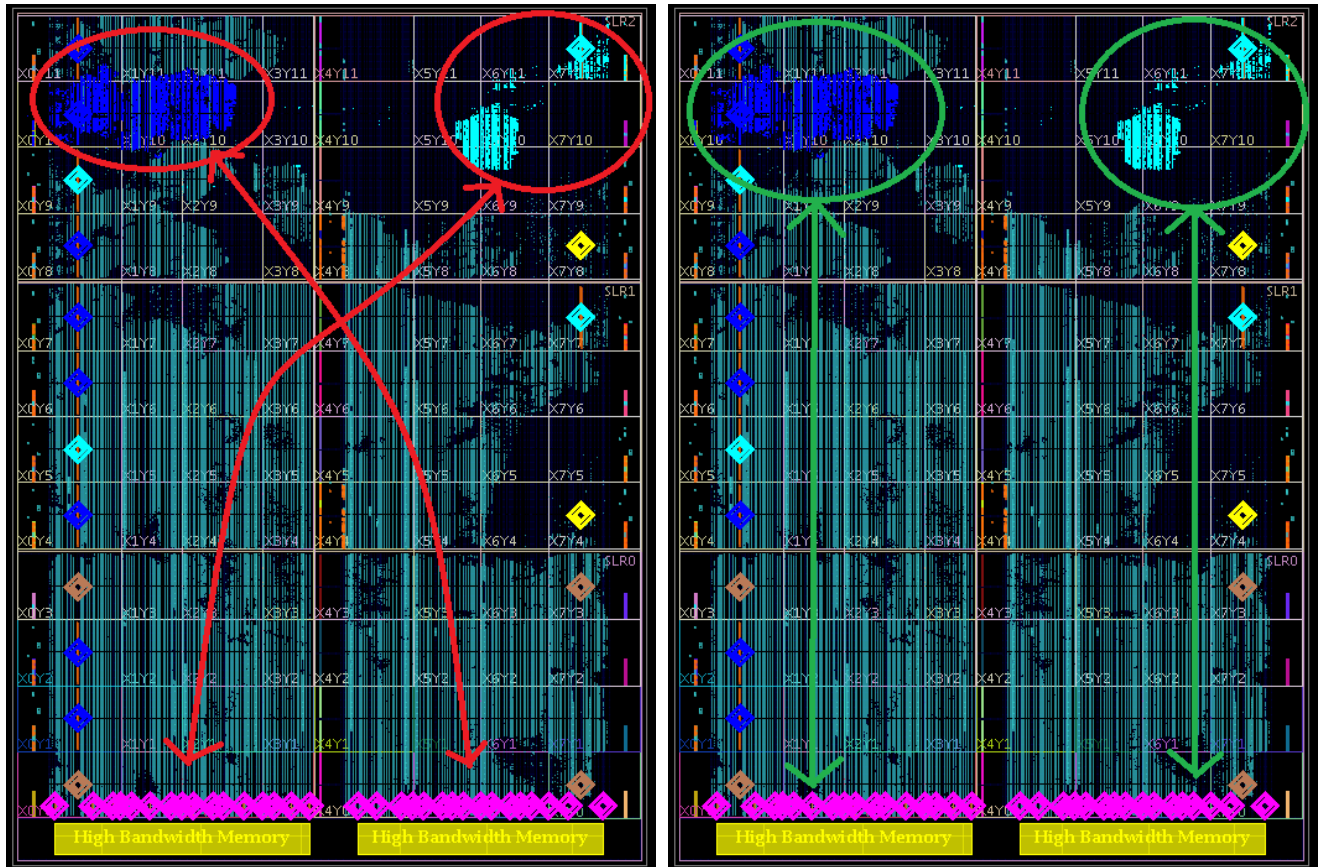
- 
RECOMMENDED: Xilinx recommends keeping the paths from SLR2 and SLR1 vertically aligned to their respective HBM AXI interfaces to avoid crossing the device diagonally.
- 
TIP: Use auto-pipelining (e.g., AXI Register Slice IP) to ensure timing closure between the HBM interfaces and any SLR at 450 MHz.

Figure 14: HBM Sub-Optimal Design Planning (left) versus Optimal Design Planning (right)



X21196-121919

Related Information

[Auto-Pipelining Considerations](#)
[SLR Crossing for Wide Buses](#)

Resource Planning within SLR0

Proper management of the HBM AXI Interfaces and other logic within the SLR0 can provide optimal quality of results (QoR) and minimize routing congestion. Following are some common design planning considerations for the SLR0 in HBM devices:

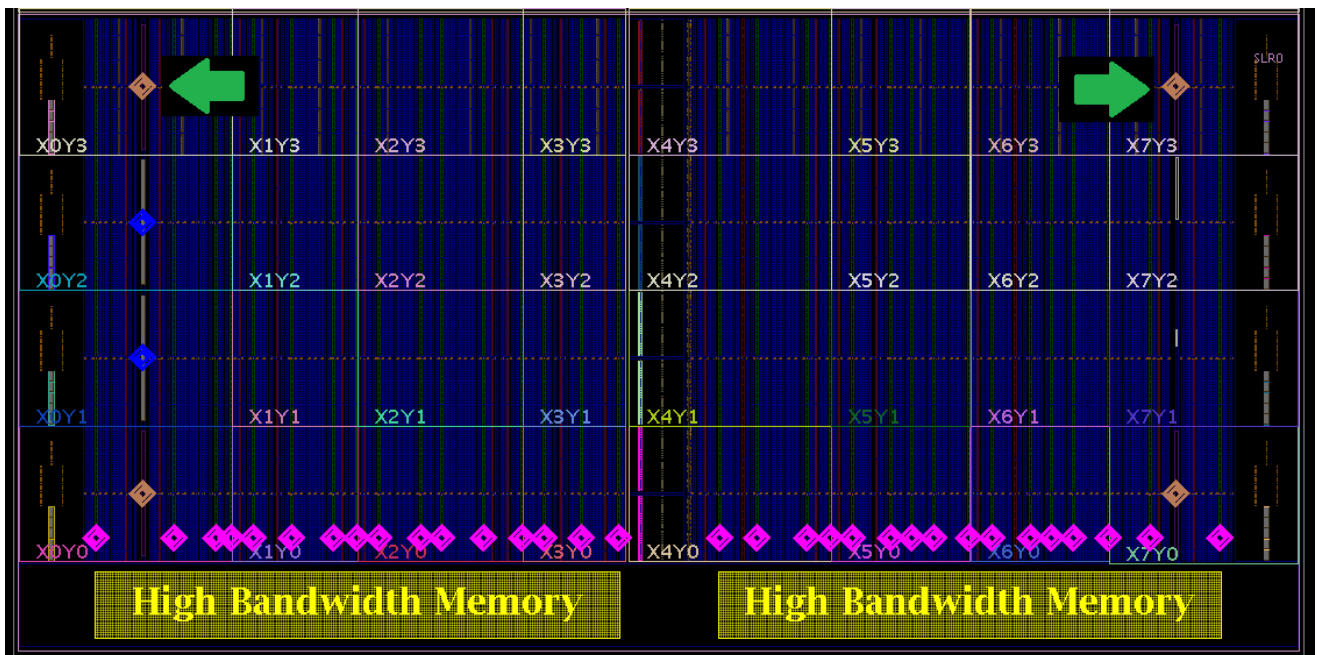
- For designs that heavily utilize the HBM AXI interfaces, budget for lower overall fabric utilization of non-HBM logic in SLR0 to better accommodate the resources required for the HBM AXI interfaces.
- Using MIG IP in the SLR0 might result in timing closure challenges for HBM AXI interfaces located near the I/O columns of the device. When using MIG IP, consider using the I/O columns located in SLR2 or SLR1.

- Be aware of address ranges and the physical location of the HBM AXI interfaces that can impact the latency and bandwidth of the design. To optimize the performance of the HBM, utilize the physical HBM AXI interfaces on the same device side as the addressed HBM stack.

PCIE4C to HBM AXI Paths within SLR0

To achieve optimal timing QoR and minimize routing congestion when designing with HBM and PCIE4C, Xilinx recommends using the PCIE4C sites that are farthest away from the 32 HBM AXI interface in SLR0. In the following figure, these sites are PCIE4CE4_X0Y1 and PCIE4CE4_X1Y1 indicated by the green arrows.

Figure 15: Recommended PCIE4C Sites in SLR0 of a Virtex UltraScale+ HBM vu37p Device



Configuration

Configuration is the process of loading application-specific data into the internal memory of the device. Because Xilinx device configuration data is stored in CMOS configuration latches (CCLs), the configuration data is volatile and must be reloaded each time the device is powered up.

Xilinx devices can load themselves through configuration pins from an external, non-volatile memory device. Devices can also be configured by an external smart source, such as the following:

- Microprocessor

- Microcontroller
- DSP processor
- Personal computer (PC)
- Board tester

When board planning, consider configuration aspects up front, which makes it easier to configure as well as debug. Each device family has a Configuration User Guide that is the primary resource for detailed information about each of the supported configuration modes and their trade-offs on pin count, performance, and cost.

Related Information

[Other Xilinx Documentation](#)

Board Design Tips

When designing a board, it is important to consider which interfaces and pins will assist with debug capability beyond configuration. For example, Xilinx recommends that you ensure the JTAG interface is accessible even when the interface is not the primary configuration mode. The JTAG interface allows you to check the device ID and device DNA information, and you can use the interface to enable indirect flash programming solutions during prototyping.

In addition, signals such as the INIT_B and DONE are critical for device configuration debug. The INIT_B signal has multiple functions. It indicates completion of initialization at power-up and can indicate when a CRC error is encountered. Xilinx recommends that you connect the INIT_B and DONE signals to light-emitting diodes (LEDs) using LED drivers and pull-ups.

For recommended pull-up values, see the configuration user guide for your device:

- *7 Series FPGAs Configuration User Guide* ([UG470](#))
- *UltraScale Architecture Configuration User Guide* ([UG570](#))

To identify and check recommended board-level pin connections, see the schematic checklists:

- *7 Series Schematic Review Recommendations* ([XMP277](#))
- *Kintex UltraScale and Virtex UltraScale FPGAs Schematic Review Checklist* ([XTP344](#))
- *UltraScale+ FPGAs and Zynq Ultrascale+ Devices Schematic Review Checklist* ([XTP427](#))

Design Creation with RTL

After planning your device I/O, planning how to lay out your PCB, and deciding on your use model for the Vivado[®] Design Suite, you can begin creating your design. Design creation includes:

- Planning the hierarchy of your design
- Identifying the IP cores to use and customize in your design
- Creating the custom RTL for interconnect logic and functionality for which a suitable IP is not available
- Creating timing, power, and physical constraints
- Specifying additional constraints, attributes, and other elements used during synthesis and implementation

When creating your design, the main points to consider include:

- Achieving the desired functionality
- Operating at the desired frequency
- Operating with the desired degree of reliability
- Fitting within the silicon resource and power budget

Decisions made at this stage affect the end product. A wrong decision at this point can result in problems at a later stage, causing issues throughout the entire design cycle. Spending time early in the process to carefully plan your design helps to ensure that you meet your design goals and minimize debug time in lab.

Defining a Good RTL Design Hierarchy

The first step in design creation is to decide how to partition the design logically. The main factor when considering hierarchy is to partition a part of the design that contains a specific function. This allows a specific designer to design a piece of IP in isolation as well as isolating a piece of code for reuse.

However, defining a hierarchy based on functionality only does not take into account how to optimize for timing closure, runtime, and debugging. The following additional considerations made during hierarchy planning also help in timing closure.

Add I/O Components Near the Top Level

Where possible, add I/O components near the top level for design readability. When you infer a component, you provide a description of the function you want to accomplish. The synthesis tool then interprets the HDL code to determine which hardware components to use to perform the function. Components that can be inferred are simple single-ended I/O (IBUF, OBUF, OBUFT and IOBUF) and single data rate registers in the I/O.

When using the tool to infer IOBUF or OBUFT components, make sure that the enable logic and the input/output logic are all in the same hierarchy. If the logic is in different hierarchies and there are KEEP_HIERARCHY or DONT_TOUCH attributes between the hierarchies, the tool will not be able to infer these buffers.

I/O components that need to be instantiated, such as differential I/O (IBUFDS, OBUFDS) and double data-rate registers (IDDR, ODDR, ISERDES, OSERDES), should also be instantiated near the top level. When you instantiate a component, you add an instance of that component to your HDL file. Instantiation gives you full control over how the component is used. Therefore, you know exactly how the logic will be used.

Insert Clocking Elements Near the Top Level

Inserting the clocking elements towards the top level allows for easier clock sharing between modules. This sharing may result in fewer clocking resources needed, which helps in resource utilization, improved performance, and power.

Aside from the module the clocks are created in, clock paths should only drive down into modules. Any paths that go through (down from top and then back to top) can create a delta cycle problem in VHDL simulation that is difficult and time consuming to debug.

Register Data Paths at Logical Boundaries

Register the outputs of hierarchical boundaries to contain critical paths within a single module or boundary. Consider registering the inputs also at the hierarchical boundaries. It is always easier to analyze and repair timing paths which lie within a module, rather than a path spanning multiple modules. Any paths that are not registered at hierarchy boundaries should be synthesized with hierarchy rebuilt or flat to allow cross hierarchy optimization. Registering the datapaths at logical boundaries helps to retain traceability (for debug) through the design process because cross hierarchical optimizations are kept to a minimum and logic does not move across modules.

Address Floorplanning Considerations

A floorplan ensures that cells belonging to a specific portion in the design netlist are placed at particular locations on the device. You can use manual floorplanning to accomplish the following:

- Partition logic to a particular SLR when using SSI technology devices.
- Close timing on a design when timing is not met using standard flows.

If the cells are not contained within a level of hierarchy, all objects must be included individually in the floorplan constraint. If synthesis changes the names of these objects, you must update the constraints. A good floorplan is contained at the hierarchy level, because this requires only a one line constraint.

Floorplanning is not always required. Floorplan only when necessary.

For more information on floorplanning, see this [link](#) in the *Vivado Design Suite User Guide: Design Analysis and Closure Techniques (UG906)*.



RECOMMENDED: *Although the Vivado tools allow cross hierarchy floorplans, these require more maintenance. Avoid cross hierarchy floorplans where possible.*

Optimize Hierarchy for Functional and Timing Debug

As discussed earlier in this section, keeping the critical path within the same hierarchical boundary is helpful in debugging and meeting timing requirements. Similarly, for functional debug (and modification) purposes, signals that are related should be kept in the same hierarchy. This allows the related signals to be probed and modified with relative ease, as signal names optimized by synthesis are easier to trace when contained in a single level of hierarchy.

Apply Attributes at the Module Level

Applying attributes at the module level can keep code tidier and more scalable. Instead of having to apply an attribute at the signal level, you can apply the attribute at the module level and have the attribute propagated to all signals declared in the current hierarchy. Applying attributes at the module level also allows you to override global synthesis options.



CAUTION! *Unlike other attributes, the DONT_TOUCH attribute does not propagate from a module to all the signals inside the module. For more information, see this [link](#) in the Vivado Design Suite User Guide: Synthesis (UG901).*

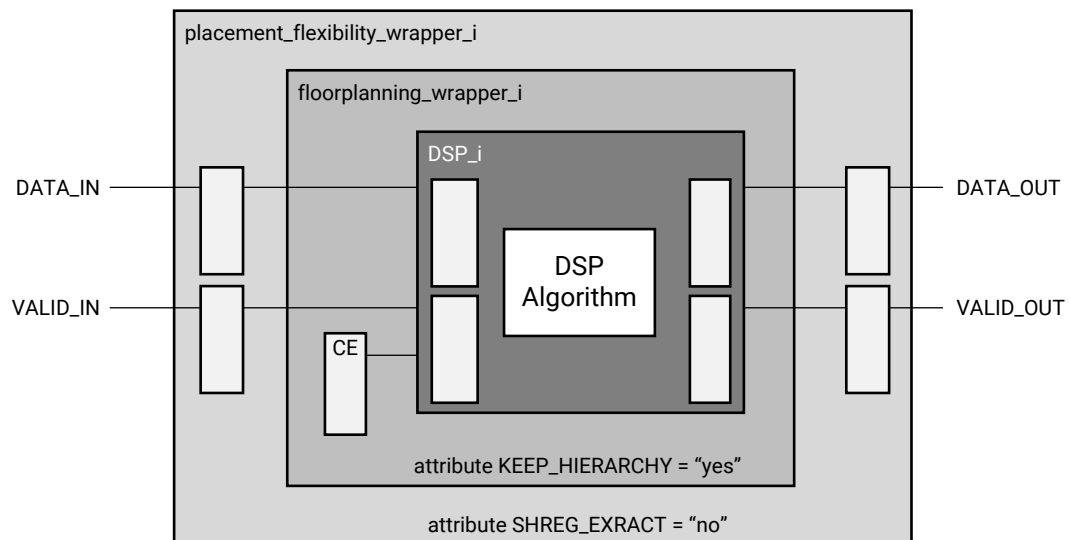
Optimize Hierarchy for Advanced Design Techniques

Advanced design techniques such as bottom-up synthesis, Dynamic Function eXchange (DFX), and out-of-context design require planning at the hierarchical level. The designer must choose the appropriate level of hierarchy for the technique being used. These techniques are not covered in this document. For more information, see this [link](#) in the *Vivado Design Suite User Guide: Hierarchical Design (UG905)*.

Example of Upfront Hierarchical Planning for High Speed DSP Designs

The following example is not applicable to all designs, but demonstrates what can be done with hierarchy. DSP designs generally allow latency to be added to the design. This allows registers to be added to them to be optimized for performance. In addition, registers can be used to increase placement flexibility. This is important because at high clock frequency, signals cannot traverse the die in one clock cycle. Adding registers can allow hard-to-reach areas to be used. The following figure shows how effective hierarchy planning results in faster timing closure.

Figure 16: Effective Hierarchy Planning Example



X13500-122019

There are three levels of hierarchy in this part of the design:

- `DSP_i`

In the `DSP_i` algorithm block, both the inputs and outputs are registered. Because registers are plentiful in a device, it is preferable to use this method to improve the timing budget.

- `floorplanning_wrapper_i`

In `floorplanning_wrapper_i`, there is a CE signal. CE signals are typically heavily-loaded and can present a timing challenge. They should be included in a floorplan. By creating a floorplanning wrapper, this module can be manually floorplanned later if needed.

In addition, `KEEP_HIERARCHY` has been added at the module level to ensure that hierarchy is preserved for floorplanning regardless of any other global synthesis options.

- `placement_flexibility_wrapper_i`

In `placement_flexibility_wrapper_i`, the `DATA_IN`, `VALID_IN`, `DATA_OUT` and `VALID_OUT` signals are registered. Because these signals are not intended to be part of the floorplan, they are outside `floorplanning_wrapper_i`. If they were in the floorplan, they would not be able to fulfill the requirement for placement flexibility.

In addition, more registers can be added later as long as both `DATA_IN` + `VALID_IN` or `DATA_OUT` and `VALID_OUT` are treated as pairs. If more registers are added, the synthesis tool might infer shift register LUTs (SRLs), which will force all registers into one component and not help placement flexibility. To prevent this, `SHREG_EXTRACT` has been added at the module level and set to `NO`.

Working with Intellectual Property (IP)

Pre-validated Intellectual Property (IP) cores significantly reduce design and validation efforts, and ensure a large advantage in time-to-market. See the following resources for more information on working with IP:

- [Vivado Design Suite User Guide: Designing with IP \(UG896\)](#)
- [Vivado Design Suite User Guide: Designing IP Subsystems Using IP Integrator \(UG994\)](#)
- [Vivado Design Suite QuickTake Video: Configuring and Managing Reusable IP in Vivado](#)

Planning IP Requirements

Planning IP requirements is one of the most important stages of any new project:

- Evaluate the IP options available from Xilinx or third-party partners against required functionality and other design goals. For example:
 - Is custom logic more desirable compared to an available IP core?
 - Does it make sense to package a custom design for reuse in multiple projects in an industry standard format?
- Consider the interfaces that are required such as, memory, network, and peripherals.



IMPORTANT! To ensure that the tools process the IP-specific constraints properly, add the .xci or .xcix IP source files to the project. Do not use the IP-generated output DCP files as project sources when working with IP.

AMBA AXI

Xilinx has standardized IP interfaces on the open AMBA[®] 4 AXI4 interconnect protocol. This standardization eases integration of IP from Xilinx and third-party providers, and maximizes system performance. Xilinx has worked with Arm[®] to define the AXI4, AXI4-Lite, and AXI4-Stream specifications for efficient mapping into its device architectures.

AXI4 is targeted at high performance, high clock frequency system designs, and is suitable for high-speed interconnects. AXI4-Lite is a light-weight version of AXI4, and is used mostly for accessing control and status registers.

AXI4-Stream is used for unidirectional streaming of data from Master to Slave. This is typically used for DSP, Video and Communications applications.

Vivado Design Suite IP Catalog

The IP catalog is a single location for Xilinx-supplied IP. In the IP catalog, you can find IP cores for embedded systems, DSP, communication, interfaces, and more.

From the IP catalog, you can explore the available IP cores, and view the Product Guide, Change Log, Product Web page, and Answer Records for any IP.

You can access and customize the cores in the IP catalog through the GUI or Tcl shell. You can also use Tcl scripts to automate the customization of IP cores.

Custom IP

Xilinx uses the industry standard IP-XACT format for delivery of IP, and provides tools (IP packager) to package custom IP. Accordingly, you can also add your own customized IP to the catalog and create IP repositories that can be shared in a team or across a company. IP from third-party providers can also be added to this catalog, provided it is packaged in IP packager, even if it is already in the IP-XACT format.

Selecting IP from the IP Catalog

All Xilinx and third-party vendor IP is categorized based on applications such as communications and networking; video and image processing; and automotive and industrial. Use this categorization to browse the catalog to see which IP is available for your area of interest.

A majority of the IP in the IP catalog is free. However, some high value IP has an associated cost and requires a license. The IP catalog informs you about whether or not the IP requires purchase, as well as the status of the license. To select an IP from the catalog, consider the following key features, based on your design requirements, and what the specific IP offers:

- Silicon Resources required by this IP (found in the respective IP Product Guide)
- Is this IP supported in the device and speed grade being considered (the selection of the IP often drives the speed grade decision)? If supported, what is the max achievable throughput and Fmax?
- External interface standards, needed for your design to talk to its companion chip on board:
 - Industry-standard interfaces such as Ethernet, PCIe® interfaces, etc.
 - Memory interfaces - number of memory interfaces, including their size and performance.
 - Xilinx proprietary interfaces such as Aurora.

Note: You can also choose to design your own custom interface.
- On-chip bus protocol supported by the IP (Application interface)
- On-chip bus protocol, needed for interaction with the rest of your design. Examples:
 - AXI4
 - AXI4-Lite
 - AXI4-Stream
- If multiple protocols are involved, bridging IP cores might have to be chosen using infrastructure IP from the IP catalog. For example:
 - AXI-AHB bridge
 - AXI-AXI Interconnect
 - AXI-PCIe bridge
 - AXI-PLB bridge

Customizing IP

IP can be customized through the GUI or through Tcl scripts.

Related Information

[Using the Customization GUI](#)

[Using a Tcl Script](#)

Using the Customization GUI

Using the graphical interface is the easiest way to find, research, and customize IP. Each IP is customized with its own set of tabs or pages. Related configuration options are grouped together. An example of a customization window is shown in the following figure. A unique customization of an IP can be created, which is represented in an XCI file. From this, the various output products of an IP can be created.

Using a Tcl Script

Almost every GUI action results in the issuance of a Tcl command. The creation of an IP including the setting of all the customization options can be performed in a Tcl script without user interaction.

You would need to know the names of the configuration options, and the values to which they can be set. Typically, you first perform the customization through the GUI, and then create the script from that. Once you see the resulting Tcl script, you can easily modify the script for your needs, such as changing data sizes.

Tcl script based IP creation is useful for automation, for example working with version control system. For information about source management and revision control, see this [link](#) in the *Vivado Design Suite User Guide: Design Flows Overview (UG892)*.

IP Versions and Revision Control

When IP is customized, the tool creates an XCI file containing all the selected parameterization values. Each Vivado IDE version supports only one version of an IP. Xilinx recommends that you use this latest IP version. If you use an older IP version, you must save all the output products for the older version. For information about source management and revision control, see this [link](#) in the *Vivado Design Suite User Guide: Design Flows Overview (UG892)*.



IMPORTANT! For memory IP in 7 series devices, a PRJ file is created in addition to the XCI file. When using revision control with 7 series memory IP, keep the PRJ file in the same directory as the XCI file.

RTL Coding Guidelines

You can create custom RTL to implement glue logic functions as well as functions without suitable IP. For optimal results, follow the coding guidelines in this section. For additional guidelines, see this [link](#) in the *Vivado Design Suite User Guide: Synthesis (UG901)*.

Using Vivado Design Suite HDL Templates

Use the Vivado Design Suite Language Templates when creating RTL or instantiating Xilinx® primitives. The Language Templates include recommended coding constructs for proper inference to the Xilinx device architecture. Using the Language Templates can ease the design process and lead to improved results. To open the Language Templates from the Vivado IDE, select the **Language Templates** option in the Flow Navigator, and select the desired template.

Control Signals and Control Sets

A control set is the grouping of control signals (set/reset, clock enable and clock) that drives any given SRL, LUTRAM, or register. For any unique combination of control signals, a unique control set is formed. This is important, because registers within a 7 series slice all share common control signals, and thus, only registers with a common control set can be packed into the same slice. For example, if a register with a given control set has just one register as a load, the other seven registers in the slice it occupies will be unusable.

Designs with too many unique control sets might have many wasted resources as well as fewer options for placement, resulting in higher power and lower performance. Designs with fewer control sets have more options and flexibility in terms of placement, generally resulting in improved results.

In UltraScale™ devices, there is more flexibility in control set mapping within a CLB. Resets that are undriven do not form part of the control set, because the tie off is generated locally within the slice. However, it is good practice to limit unique control sets to give maximum flexibility in placement of a group of logic.

Resets

Resets are one of the more common and important control signals to take into account and limit in your design. Resets can significantly impact your design's performance, area, and power.

Inferred synchronous code might result in resources such as:

- LUTs
- Registers
- SRLs
- Block or LUT memory
- DSP48 registers

The choice and use of resets can affect the selection of these components, resulting in less optimal resources for a given design. A misplaced reset on an array can mean the difference between inferring one block RAM, or inferring several thousand registers.

Asynchronous resets described at the input or output of a multiplier might result in registers placed in the slices rather than the DSP block. In such situations, additional logic resources are used, which negatively impacts the power consumption and design performance.

When and Where to Use a Reset

Xilinx devices have a dedicated global set/reset signal (GSR). This signal sets the initial value of all sequential cells in hardware at the end of device configuration.

If an initial state is not specified, sequential primitives are assigned a default value. In most cases, the default value is zero. Exceptions are the FDSE and FDPE primitives that default to a logic one. Every register will be at a known state at the end of configuration. Therefore, it is not necessary to code a global reset for the sole purpose of initializing a device on power up.

Xilinx highly recommends that you take special care in deciding when the design requires a reset, and when it does not. In many situations, resets might be required on the control path logic for proper operation. However, resets are generally less necessary on the data path logic. Limiting the use of resets:

- Limits the overall fanout of the reset net.
- Reduces the amount of interconnect necessary to route the reset.
- Simplifies the timing of the reset paths.
- Results in many cases in overall improvement in performance, area, and power.



RECOMMENDED: Evaluate each synchronous block, and attempt to determine whether a reset is required for proper operation. Do not code the reset by default without ascertaining its real need.

Functional simulation should easily identify whether a reset is needed or not.

For logic in which no reset is coded, there is much greater flexibility in selecting the device resources to map the logic.

The synthesis tool can then pick the best resource for that code in order to arrive at a potentially superior result by considering, for example:

- Requested functionality
- Performance requirements
- Available device resources
- Power

Synchronous Reset vs. Asynchronous Reset

If a reset is needed, Xilinx recommends using synchronous resets. Synchronous resets have the following advantages over asynchronous resets:

- Synchronous resets can directly map to more resource elements in the device architecture.

- Asynchronous resets impact the performance of the general logic structures. Because all Xilinx device general-purpose registers can program the set/reset as either asynchronous or synchronous, it might seem like there is no penalty in using asynchronous resets. If a global asynchronous reset is used, it does not increase the control sets. However, the need to route this reset signal to all register elements increases routing complexity.
- Asynchronous resets have a greater probability of corrupting memory contents of block RAMs, LUTRAMs, and SRLs during reset assertion. This is especially true for registers with asynchronous resets that drive the input pins of block RAMs, LUTRAMs, and SRLs.
- Synchronous resets offer more flexibility for control set remapping when higher density or fine tuned placement is needed. A synchronous reset can be remapped to the data path of the register if an incompatible reset is found in the more optimally placed slice. This can reduce routing resource utilization and increase placement density where needed to allow proper fitting and improved performance.
- Some resources such as the DSP48 and block RAM have only synchronous resets for the register elements within the block. When asynchronous resets are used on register elements associated with these elements, those registers may not be inferred directly into those blocks without impacting functionality.

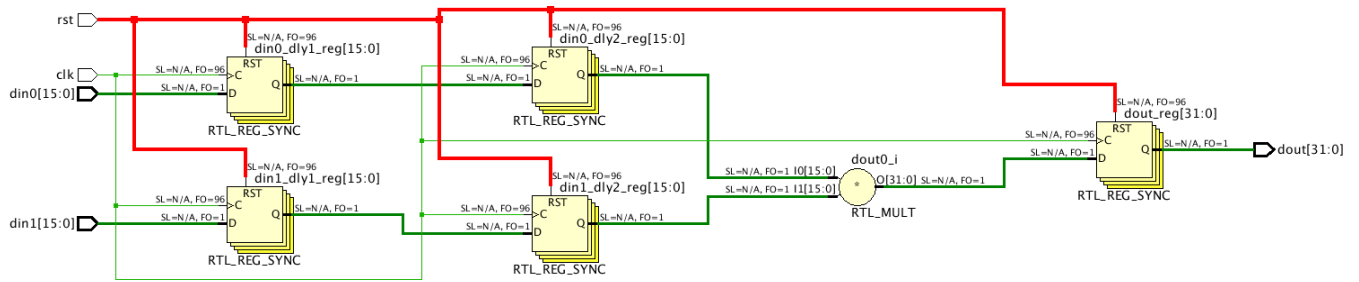
Following are additional considerations:

- The clock works as a filter for small reset glitches for synchronous resets. However, if these glitches occur near the active clock edge, the flip-flop might become metastable.
- Synchronous resets might need to stretch the pulse width to ensure that the reset signal pulse is wide enough for the reset to be present during an active edge of the clock.
- When using asynchronous resets, remember to synchronize the deassertion of the asynchronous reset. Although the relative timing between clock and reset can be ignored during reset assertion, the reset release must be synchronized to the clock. Avoiding the reset release edge synchronization can lead to metastability. During reset release, setup and hold timing conditions must be satisfied for the reset pin relative to the clock pin of a register. A violation of the setup and hold conditions for asynchronous reset (e.g., reset recovery and removal timing) might cause the flip-flop to become metastable, causing design failure due to switching to an unknown state. Note that this situation is similar to the violation of setup and hold conditions for the flip-flop data pin.

Reset Coding Example: Multiplier with Synchronous Reset

To take advantage of the existing DSP primitive features, the following example shows a multiplier with synchronous reset.

Figure 17: Multiplier with Pipeline Registers (Synchronous Reset)



In this circuit, the DSP48 primitive is inferred with all pipeline registers packed within the DSP primitive (AREG/BREG=1, MREG=1, PREG=1).

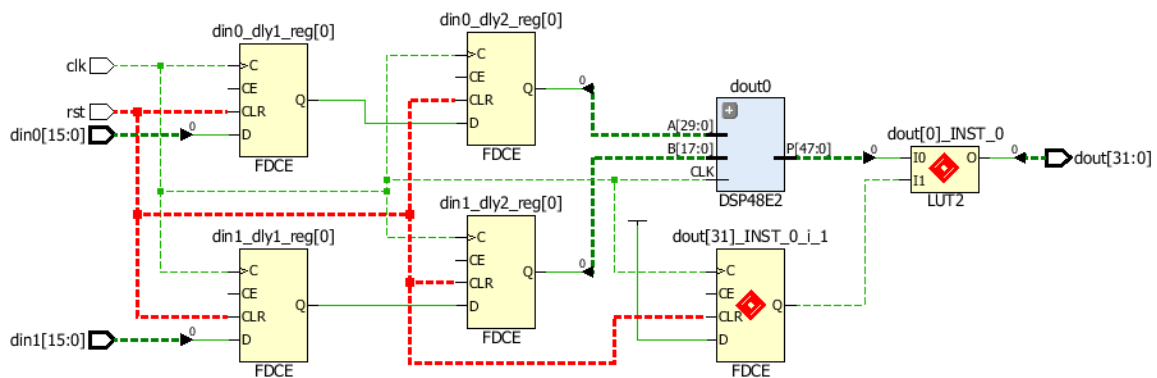
This coding example has the following advantages:

- Optimal resource usage
- Better performance and lower power
- Lower number of endpoints

Reset Coding Example: Multiplier with Asynchronous Reset

The following example illustrates the importance of using registers with synchronous resets for the logic targeting the dedicated DSP resources. The following figure shows a 16x16 bits DSP48-based multiplier using pipeline registers with asynchronous reset. Synthesis must use regular fabric registers for the input stages, as well as an external register and 32 LUT2s (red markers) to emulate the asynchronous reset on the DSP output (DSP48 P registers are enabled but not connected to reset). This costs an extra 65 registers and 32 LUTs, and the DSP48 results in the configuration: AREG/BREG=0, MREG=0, PREG=1.

Figure 18: Multiplier with Pipeline Registers Using Asynchronous Resets




By simply changing the reset definition as shown in the following figure, such that the multiplier pipeline registers use a synchronous reset, synthesis can take advantage of the DSP48 internal registers: AREG/BREG=1, MREG=1, PREG=1.

Figure 19: Changing Asynchronous Reset into Synchronous Reset on Multiplier

```

always @ (posedge clk or posedge rst) begin
    if (rst) begin
        din0_dly1 <= 16'h0;
        din0_dly2 <= 16'h0;
        din1_dly1 <= 16'h0;
        din1_dly2 <= 16'h0;
        dout      <= 32'h0;
    end else begin
        din0_dly1 <= din0;
        din0_dly2 <= din0_dly1;
        din1_dly1 <= din1;
        din1_dly2 <= din1_dly1;
        dout      <= din0_dly2 * din1_dly2;
    end
end
    
```



```

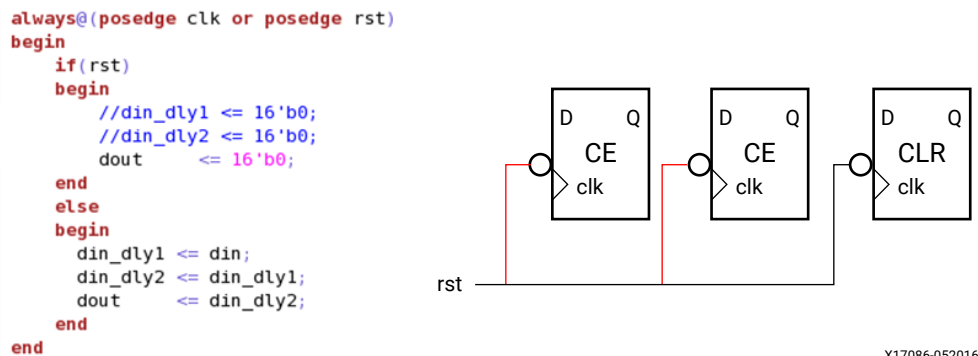
always @ (posedge clk) begin
    if (rst) begin
        din0_dly1 <= 16'h0;
        din0_dly2 <= 16'h0;
        din1_dly1 <= 16'h0;
        din1_dly2 <= 16'h0;
        dout      <= 32'h0;
    end else begin
        din0_dly1 <= din0;
        din0_dly2 <= din0_dly1;
        din1_dly1 <= din1;
        din1_dly2 <= din1_dly1;
        dout      <= din0_dly2 * din1_dly2;
    end
end
    
```

Due to saving fabric resources and taking advantage of all DSP48 internal registers, the design performance and power efficiency are optimal.

Issues When Trying to Eliminate Reset in HDL Code

When optimizing the code to eliminate reset, commenting out the conditions within the reset declaration does not create the desired structures and instead creates issues. For example, the following figure shows three pipeline stages with asynchronous reset used for each. If you attempt to eliminate the reset condition for two of the pipeline stages by commenting out the code with the reset condition, the asynchronous reset becomes enabled (inverted logic of `rst`).

Figure 20: Commenting Out Code with Reset Conditions



The optimal way to remove the resets is to create separate sequential logic procedures with one for reset conditions and the other for non-reset conditions, as shown in the following figure.

Figure 21: Separate Procedural Statements for Registers With and Without Reset

```
always@(posedge clk)
begin
    din_dly1 <= din;
    din_dly2 <= din_dly1;
end

always@(posedge clk or posedge rst)
begin
    if(rst)
        dout <= 16'd0;
    else
        dout <= din_dly2;
end
```



TIP: When using a reset, make sure that all registers in the procedural statement are reset.

Clock Enables

When used properly, clock enables can significantly reduce design power with little impact on area or performance. However, when clock enables are used improperly, they can lead to:

- Increased resource utilization
- Decreased placement density
- Increased power
- Reduced performance

In most cases, low fanout clock enables are the main contributor to the high number of control sets.

Creating Clock Enables

Clock enables are created when an incomplete conditional statement is coded into a synchronous block. A clock enable is inferred to retain the last value when the prior conditions are not met. When this is the desired functionality, it is valid to code in this manner. However, in some cases when the prior conditional values are not met, the output is a don't care. In that case, Xilinx recommends closing off the conditional (that is, use an `else` clause), with a defined constant (that is, assign the signal to a one or a zero).

In most implementations, this does not result in added logic, and avoids the need for a clock enable. The exception to this rule is in the case of a large bus when inferring a clock enable in which the value is held can help in power reduction. The basic premise is that when small numbers of registers are inferred, a clock enable can be detrimental because it increases control set count. However, in larger groups, it can become more beneficial and is recommended.

Reset and Clock Enable Precedence

In Xilinx devices, all registers are built to have set/reset take precedence over clock enable, whether an asynchronous or synchronous set/reset is described. In order to obtain the most optimal result, Xilinx recommends that you always code the set/reset before the enable (if deemed necessary) in the `if/else` constructs within a synchronous block. Coding a clock enable first forces the reset into the data path and creates additional logic.

Related Information

[Clocking Guidelines](#)

Controlling Enable/Reset Extraction with Synthesis Attributes

You can force control set mapping by applying the `DIRECT_RESET` / `DIRECT_ENABLE` / `EXTRACT_RESET` / `EXTRACT_ENABLE` attributes as needed to handle the mapping of control sets for a given structure.

When the design includes a synchronous reset/enable, synthesis creates a logic cone mapped through the CE/R/S pins when the load is equal to or above the threshold set by the `-control_set_opt_threshold` synthesis switch, or creates a logic cone that maps through the D pin if below the threshold. The default thresholds are:

- 7 series devices: 4
- UltraScale devices: 2

Using `DIRECT_ENABLE` and `DIRECT_RESET`

To use control set mapping you can apply attributes to the nets connected to enable/reset signals, which will force synthesis to use the CE/R pin.

In the following figure, the enable signal (`en`) is only connected to one flip-flop. Therefore, the synthesis engine connected the `en` signal to the FDRE/D pin cone of logic. Note that the CE pin is tied to logic 1.

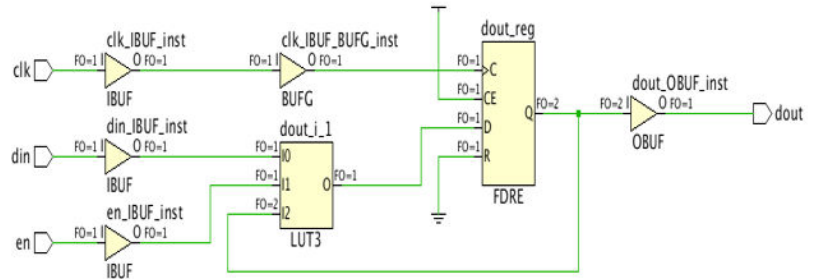
Figure 22: Clock Enable Implementation Using Datapath Logic

```

module test
(
input clk,
input en,
input din,
output reg dout
);

always@(posedge clk)
begin
    if(en)
    begin
        dout <= din;
    end
end

endmodule
    
```



To override this default behavior, you can use the `DIRECT_ENABLE` attribute. For example, the following figure shows how to connect the enable signal (`en`) to the CE pin of the register by adding the `DIRECT_ENABLE` attribute to the port/signal.

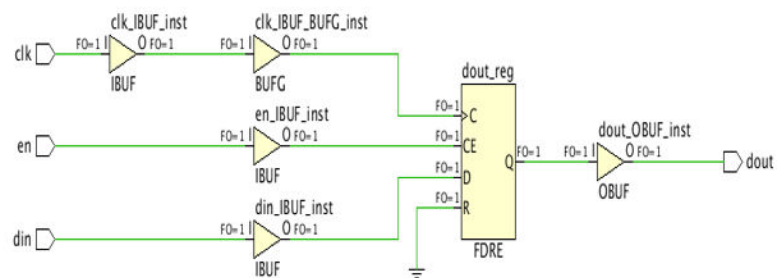
 Figure 23: Dedicated Clock Enable Implementation Using `direct_enable`

```

module test
(
input clk,
(* direct_enable = "true" *) input en,
input din,
output reg dout
);

always@(posedge clk)
begin
    if(en)
    begin
        dout <= din;
    end
end

endmodule
    
```



The following figure shows RTL code in which either `global_rst` or `int_rst` can reset the register. By default, both are mapped to the reset pin cone of logic.

Figure 24: Multiple Reset Conditions Mapped Through Datapath Logic

```

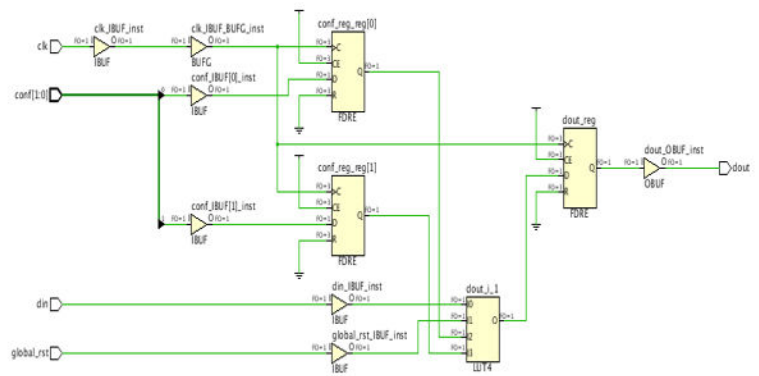
module test (
    input clk,
    input global_rst,
    input [1:0] conf,
    input din,
    output reg dout
);

reg [1:0] conf_reg;

assign int_rst = &conf_reg;

always@(posedge clk)
begin
    conf_reg <= conf;
    if(global_rst || int_rst)
        dout <= 1'b0;
    else
        dout <= din;
end

endmodule
    
```



You can use the `DIRECT_RESET` attribute to specify which reset signal to connect to the register reset pin. For example, the following figure shows how to use the `DIRECT_RESET` attribute to connect only the `global_rst` signal to the register `FDRE/R` pin and connect the `int_rst` signal to the `FDRE/D` cone of logic.

 Figure 25: Dedicated Reset Pin Usage Using `DIRECT_RESET` Attribute

```

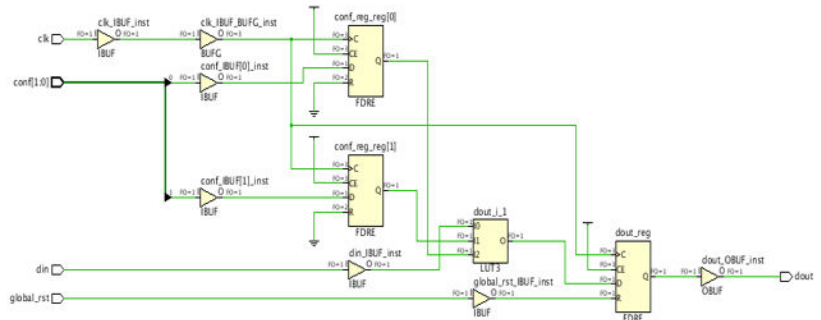
module test (
    input clk,
    (* direct_reset = "true" *) input global_rst,
    input [1:0] conf,
    input din,
    output reg dout
);

reg [1:0] conf_reg;

assign int_rst = &conf_reg;

always@(posedge clk)
begin
    conf_reg <= conf;
    if(global_rst || int_rst)
        dout <= 1'b0;
    else
        dout <= din;
end

endmodule
    
```



Pushing the Logic from the Control Pin to the Data Pin

During analysis of critical paths, you might find multiple paths ending at control pins. You must analyze these paths to determine if there is a way to push the logic into the datapath without incurring penalties, such as extra logic levels. There is less delay in a path to the D pin than CE/R/S pins given the same levels of logic because there is a direct connection from the output of the last LUT to the D input of the FF. The following coding examples show how to push the logic from the control pin to the data pin of a register.

In the following example, the enable pin of `dout_reg[0]` has 2 logic levels, and the data pin has 0 logic levels. In this situation, you can improve timing by moving the enable logic to the D pin by setting the `EXTRACT_ENABLE` attribute to "no" on the `dout` register definition in the RTL file.

Figure 26: Critical Path Ending at Control Pin (Enable) of a Register

```

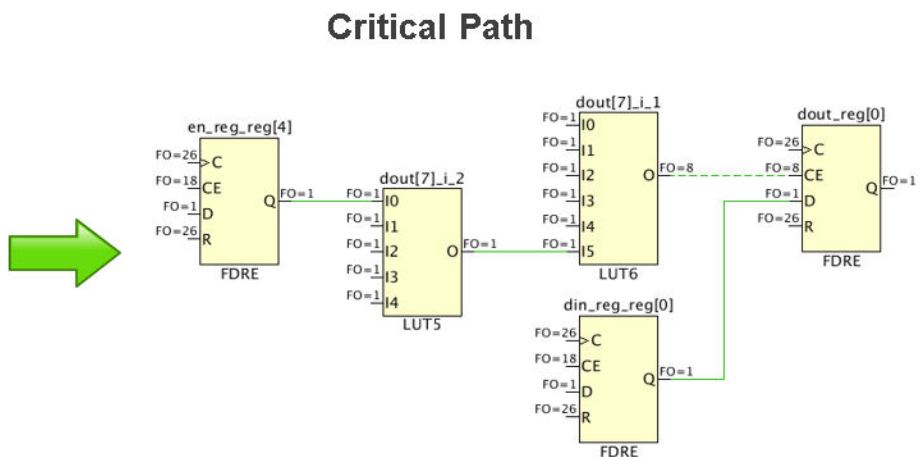
module test
(
input clk,
input [9:0] en,
input [7:0] din,
output reg [7:0] dout
);

wire en_tmp;
reg [7:0] din_reg;
reg [9:0] en_reg;

assign en_tmp = &en_reg;

always@(posedge clk)
begin
en_reg <= en;
din_reg <= din;
if(en_tmp)
dout <= din_reg;
end

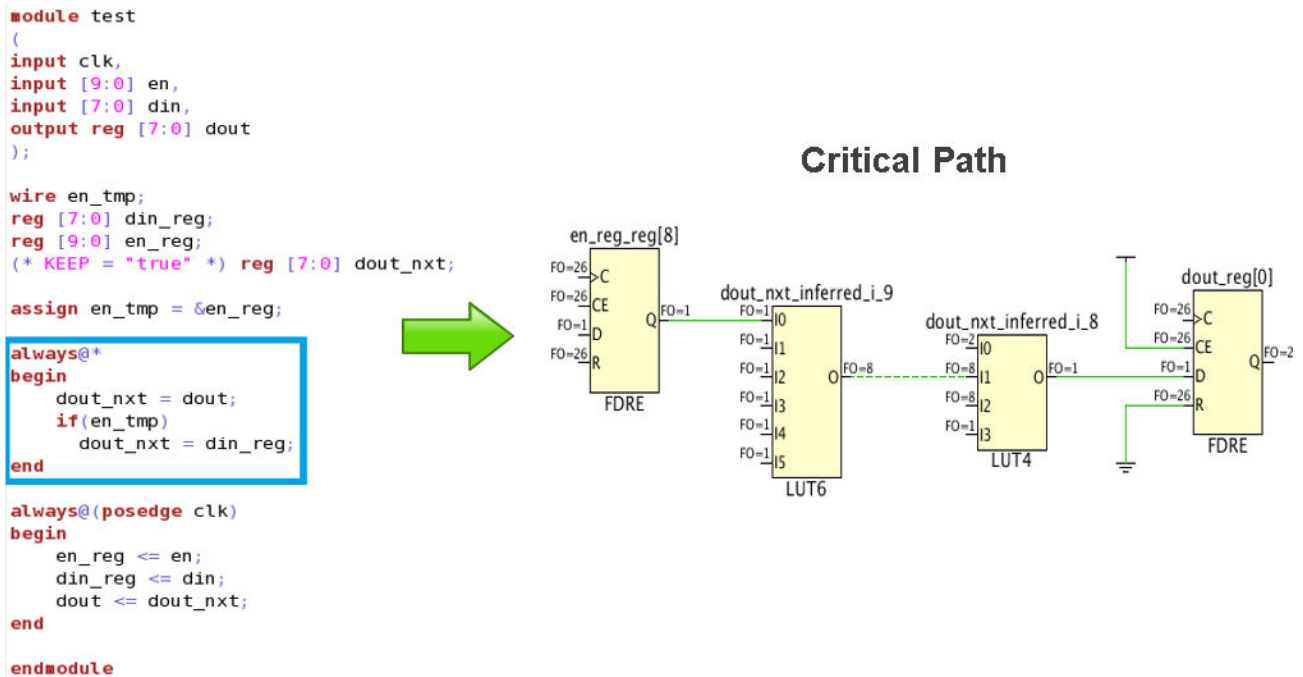
endmodule
    
```



The following example shows how to separate the combinational and sequential logic and map the complete logic in to the datapath. This pushes the logic into the D pin, which still has 2 logic levels.

You can achieve the same structure by setting the `EXTRACT_ENABLE` attribute to "no." For more information on the `EXTRACT_ENABLE` attribute, see the *Vivado Design Suite User Guide: Synthesis (UG901)*.

Figure 27: Critical Path Ending at Data Pin of a Register (Disabling Enable Extraction)



Tips for Control Signals

- Check whether a global reset is really needed.
- Avoid asynchronous control signals.
- Keep clock, enable, and reset polarities consistent.
- Do not code a set and reset into the same register element.
- If an asynchronous reset is absolutely needed, remember to synchronize its deassertion.

Know What You Infer

Your code finally has to map onto the resources present on the device. Make an effort to understand the key arithmetic, storage, and logic elements in the architecture you are targeting. Then, as you code the functionality of the design, anticipate the hardware resources to which the code will map. Understanding this mapping gives you an early insight into any potential problem.

The following examples demonstrate how understanding the hardware resources and mapping can help make certain design decisions:

- For larger than 4-bit addition, subtraction, and add-sub, a carry chain is generally used and one LUT per 2-bit addition is used (that is, an 8-bit by 8-bit adder uses 8 LUTs and the associated carry chain). For ternary addition or in the case where the result of an adder is added to another value without the use of a register in between, one LUT per 3-bit addition is used (that is, an 8-bit by 8-bit by 8-bit addition also uses 8 LUTs and the associated carry chain).

If more than one addition is needed, it may be advantageous to specify registers after every two levels of addition to cut device utilization in half by allowing a ternary implementation to be generated.

- In general, multiplication is targeted to DSP blocks. Signed bit widths less than 18x25 (18x27 in UltraScale devices) map into a single DSP Block. Multiplication requiring larger products might map into more than one DSP block. DSP blocks have pipelining resources inside them.

Pipelining properly for logic inferred into the DSP block can greatly improve performance and power. When a multiplication is described, three levels of pipelining around it generates best setup, clock-to-out, and power characteristics. Extremely light pipelining (one-level or none) might lead to timing issues and increased power for those blocks, while the pipelining registers within the DSP lie unused.

- Two SRLs with depths of 16 bits or less can be mapped into a single LUT, and single SRLs up to 32 bits can also be mapped into a single LUT.
- For conditional code resulting in standard MUX components:
 - A 4-to-1 MUX can be implemented into a single LUT, resulting in one logic level.
 - An 8-to-1 MUX can be implemented into two LUTs and a MUXF7 component, still resulting in effectively one logic (LUT) level.
 - A 16-to-1 MUX can be implemented into four LUTs and a combination of MUXF7 and MUXF8 resources, still resulting in effectively one logic (LUT) level.

A combination of LUTs, MUXF7, and MUXF8 within the same CLB/slice structure results in a very small combinational delay. Hence, these combinations are considered as equivalent to only one logic level. Understanding this code can lead to better resource management, and can help in better appreciating and controlling logic levels for the data paths.

For general logic, take into account the number of unique inputs for a given register. From that number, an estimation of LUTs and logic levels can be achieved. In general, 6 inputs or fewer always results in a single logic level. Theoretically, two levels of logic can manage up to 36 inputs. However, for all practical purposes, you should assume that approximately 20 inputs is the maximum that can be managed with two levels of logic. In general, the larger the number of inputs and the more complex the logic equation, the more LUTs and logic levels are required.



IMPORTANT! Check the availability of hardware resources and how efficiently they are being utilized early in the design cycle to enable easier modifications. This approach yields better results than waiting until late in the design cycle during timing closure.

Inferring RAM and ROM

RAM and ROM may be specified in multiple ways. Each has advantages and disadvantages.

- Inference

Advantages:

- Highly portable
- Easy to read and understand
- Self-documenting
- Fast simulation

Disadvantages:

- Might not have access to all RAM configurations available
- Might produce less optimal results

Because inference usually gives good results, it is the recommended method, unless a given use is not supported, or it is not producing adequate results in performance, area, or power. In that case, explore other methods.

When inferring RAM, Xilinx recommends that you use the HDL Templates provided in the Vivado tools. As mentioned earlier, using asynchronous reset impacts RAM inference, and should be avoided.

- Xilinx Parameterizable Macros (XPMs)

Advantages:

- Portable between Xilinx device families
- Fast simulation
- Support for asymmetric width
- Predictable quality of results (QoR)

Disadvantages:

- Limited to supported XPM options

XPMs are built on inference using fixed templates that you cannot modify. Therefore, they can guarantee QoR and can support features that standard inference does not. When standard inference does not support the features required, Xilinx recommends you use XPMs instead.

Note: When you compile simulation libraries using `compile_simlib`, XPMs are automatically compiled. For more information, see the *Vivado Design Suite User Guide: Logic Simulation (UG900)*.

- Direct Instantiation of RAM Primitives

Advantages:

- Highest level control over implementation
- Access to all capabilities of the block

Disadvantages:

- Less portable code
- Wordier and more difficult to understand functionality and intent
- Core from IP catalog

Advantages:

- Generally more optimized result when using multiple components
- Simple to specify and configure

Disadvantages:

- Less portable code
- Core management

Related Information

[Using Vivado Design Suite HDL Templates](#)

Performance Considerations When Implementing RAM

To efficiently infer memory elements, consider these factors affecting performance:

- Using Dedicated Blocks or Distributed RAMs

RAMs can be implemented in either the dedicated block RAM or within LUTs using distributed RAM. The choice not only impacts resource selection, but can also significantly impact performance and power.

In general, the required depth of the RAM is the first criterion. Memory arrays described up to 64 bits deep are generally implemented in LUTRAMs, where depths of 32 bits and less are mapped 2 bits per LUT and depths up to 64-bits can be mapped one bit per LUT. Deeper RAMs can also be implemented in LUTRAM depending on available resources and synthesis tool assignment.

Memory arrays deeper than 256 bits are generally implemented in block memory. Xilinx devices have the flexibility to map such structures in different width and depth combinations. Familiarize yourself with these configurations to understand the number and structure of block RAMs used for larger memory array declarations in the code.

- Using the Output Pipeline Register

Using an output register is required for high performance designs, and is recommended for all designs. This improves the clock to output timing of the block RAM. Additionally, a second output register is beneficial, as slice output registers have faster clock to out timing than a block RAM register. Having both registers has a total read latency of 3. When inferring these registers, they should be in the same level of hierarchy as the RAM array. This allows the tools to merge the block RAM output register into the primitive.

- Using the Input Pipeline Register

When RAM arrays are large and mapped across many primitives, they can span a considerable area of the die. This can lead to performance issues on address and control lines. Consider adding an extra register after the generation of these signals and before the RAMs. To further improve timing, use `phys_opt_design` later in the flow to replicate this register. Registers without logic on the input will replicate more easily.

Scenarios Preventing Block RAM Output Register Inference

Xilinx recommends that the memory and the output registers are all inferred in a single level of hierarchy, because this is the easiest method to ensure inference is as intended. There are two scenarios that will infer a block RAM output register. The first one is when an extra register exists on the output, and the second is when the read address register is retimed across the memory array. This can only happen using single port RAM. This is illustrated below:

Figure 28: RAM with Extra Read Register for Block RAM Output Register Inference

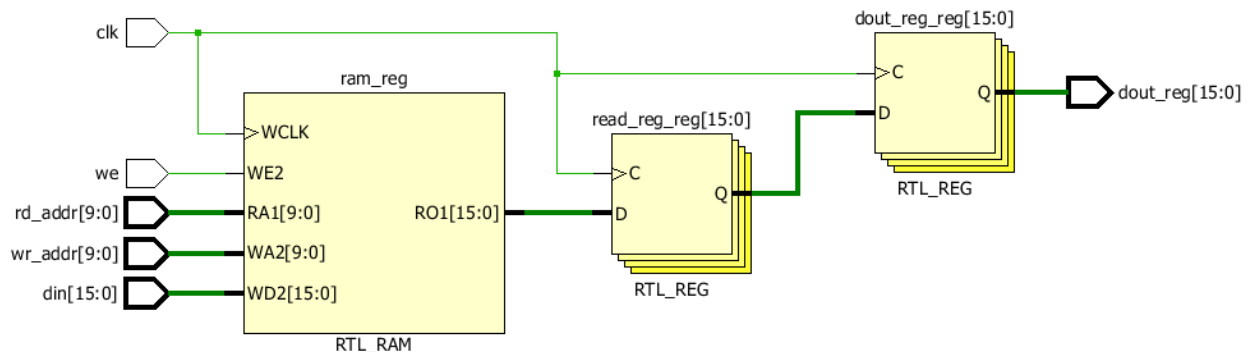
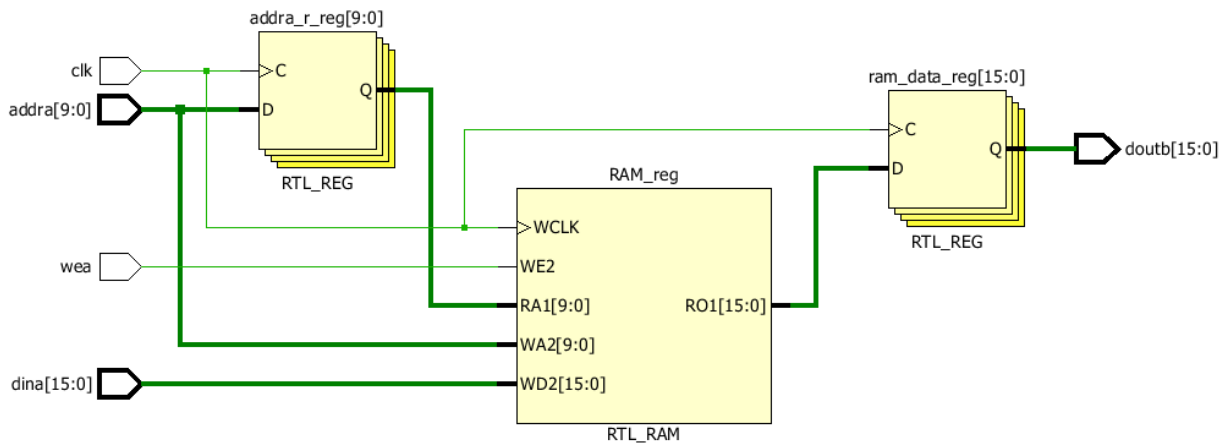


Figure 29: View of RAM Before Address Register Retiming

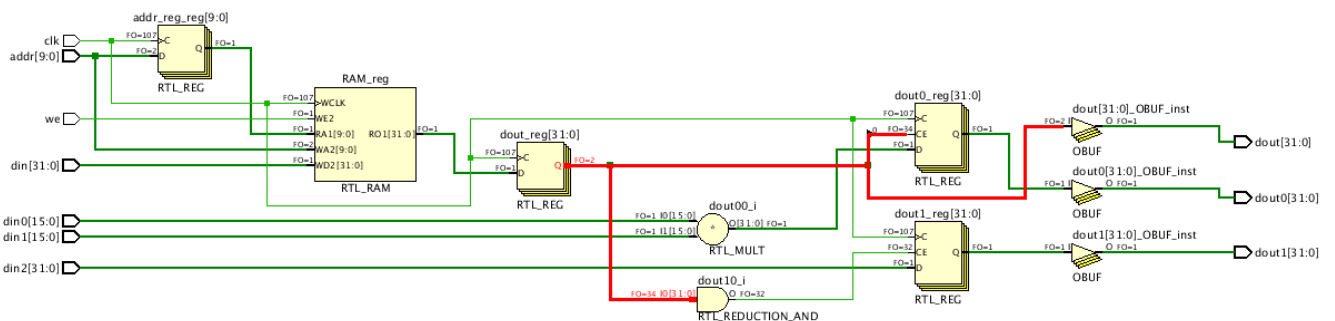


Certain deviations from these examples can prevent the inference of the output register.

Checking for Multi-Fanout on the Output of Read Data Registers

The fanout of the data output bits from the memory array must be 1 for the second register to be absorbed by the RAM primitive. This is illustrated in the following figure.

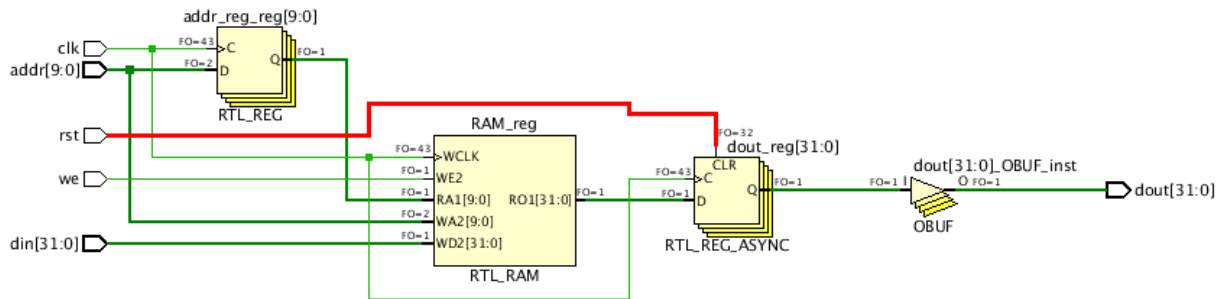
Figure 30: Multiple Fanout Preventing Block RAM Output Register Inference



Checking for Reset Signals on the Address/Read Data Registers

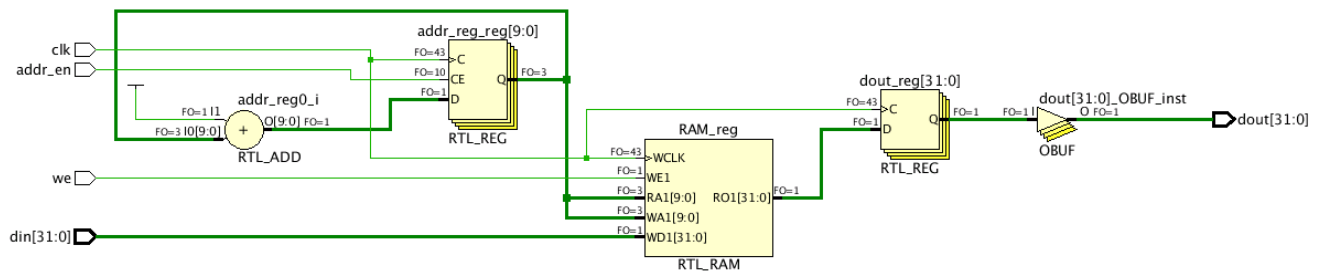
Memory arrays should not be reset. Only the output of the RAM can tolerate a reset. The reset must be synchronous in order for inference of the output register into the RAM primitive. An asynchronous reset will cause the register to not be inferred into the RAM primitive. Additionally, the output signal can only be reset to 0.

The following figure highlights an example of what to avoid to ensure correct inference of RAMs and output registers.

Figure 31: Checking for Reset On Address/Read Data Registers


Checking for Feedback Structures in Registers

Make sure that registers do not have feedback logic, in the example below, because the address requires the current value of register, this logic cannot be retimed and packed in to a block RAM. The resultant circuit is a block RAM without output registers (DOA_REG and DOB_REG set to '0').

Figure 32: Check the Presence of Feedback on Registers Around the RAM Block


Mapping Memories to UltraRAM Blocks

UltraRAM is a 4Kx72 memory block with two ports using a single clock. This primitive is only available in certain UltraScale+™ devices. In these devices, UltraRAM is included in addition to block RAM resources.

UltraRAM can be used in your design using one of the following methods:

- Rely on synthesis to infer UltraRAMs by setting the `ram_style = "ultra"` attribute on a memory declaration in HDL.
- Instantiate Xilinx XPM_MEMORY primitives.
- Instantiate UltraRAM UNISIM primitives.

The following code example shows the instantiation of XPM memory and is available in the HDL Language templates. Highlighted parameters `MEMORY_PRIMITIVE` and `READ_LATENCY` are the key parameters to infer memory as UltraRAM for high performance.

- `MEMORY_PRIMITIVE = "ultra"` specifies the memory is to be inferred as UltraRAM.
- `READ_LATENCY` defines the number of pipeline registers present on the output of the memory.

Larger memories are mapped to an UltraRAM matrix consisting of multiple UltraRAM cells configured as row x column structures.

A matrix can be created with single or multiple columns based on the depth. The current default threshold for UltraRAM column height is 8 and it can be controlled with the attribute `CASCADE_HEIGHT`.

The difference between single column and multiple column UltraRAM matrix is as follows:

- Single column UltraRAM matrix uses the built-in hardware cascade without fabric logic.
- Multiple column UltraRAM matrix uses built-in hardware cascade within each column, plus some fabric logic for connecting the columns. Extra pipelining may be required to maintain performance. This is inferred by increasing the read latency. The Vivado tools automatically pack these registers into UltraRAM as required.

Figure 33: Specifying UltraRAM in RTL Code (via XPM)

```
xpm_memory_spram # (
    // Common module parameters
    .MEMORY_SIZE      (8*(4096*72)), //positive integer
    .MEMORY_PRIMITIVE ("ultra"), //string; "auto", "distributed", "block" or "ultra";
    .MEMORY_INIT_FILE ("none"), //string; "none" or "<filename>.mem"
    .MEMORY_INIT_PARAM (""), //string;
    .USE_MEM_INIT     (0), //integer; 0,1
    .WAKEUP_TIME      ("disable_sleep"), //string; "disable_sleep" or "use_sleep_pin"
    .MESSAGE_CONTROL  (0), //integer; 0,1

    // Port A module parameters
    .WRITE_DATA_WIDTH_A (72), //positive integer
    .READ_DATA_WIDTH_A  (72), //positive integer
    .BYTE_WRITE_WIDTH_A (72), //integer; 8, 9, or WRITE_DATA_WIDTH_A value
    .ADDR_WIDTH_A       (16), //positive integer
    .READ_RESET_VALUE_A ("0"), //string
    .READ_LATENCY_A     (9), //non-negative integer
    .WRITE_MODE_A       ("read_first") //string; "write_first", "read_first", "no_change"

) xpm_memory_spram_inst (
    // Common module ports
    .sleep      (1'b0),

    // Port A module ports
    .clk_a      (clk_a),
    .rst_a      (rst_a),
    .ena        (ena),
    .regcea     (regcea),
    .wea        (wea),
    .addra      (addra),
    .dina       (dina),
    .injectsbiterra (1'b0), //do not change
    .injectdbiterra (1'b0), //do not change
    .douta      (douta),
    .sbiterra   (), //do not change
    .dbiterra   () //do not change

);
```

The preceding example uses a 32 K x 72 memory configuration, which uses eight UltraRAMs. To increase performance of the UltraRAM, more pipelining registers should be added to the cascade chain. This is achieved by increasing the read latency integer.

For more information on inferring UltraRAM in Vivado synthesis, see this [link](#) in the *Vivado Design Suite User Guide: Synthesis (UG901)*.

Coding for Optimal DSP and Arithmetic Inference

The DSP blocks within the Xilinx devices can perform many different functions, including:

- Multiplication
- Addition and subtraction
- Comparators
- Counters

- General logic

The DSP blocks are highly pipelined blocks with multiple register stages allowing for high-speed operation while reducing the overall power footprint of the resource. Xilinx recommends that you fully pipeline the code intended to map into the DSP48, so that all pipeline stages are utilized. To allow the flexibility of use of this additional resource, a set condition cannot exist in the function for it to properly map to this resource.

DSP48 slice registers within Xilinx devices contain only resets, and not sets. Accordingly, unless necessary, do not code a set (value equals logic 1 upon an applied signal) around multipliers, adders, counters, or other logic that can be implemented within a DSP48 slice. Additionally, avoid asynchronous resets, since the DSP slice only supports synchronous reset operations. Code resulting in sets or asynchronous resets may produce suboptimal results in terms of area, performance, or power.

Many DSP designs are well-suited for the Xilinx architecture. To obtain best use of the architecture, you must be familiar with the underlying features and capabilities so that design entry code can take advantage of these resources.

The DSP48 blocks use a signed arithmetic implementation. Xilinx recommends code using signed values in the HDL source to best match the resource capabilities and, in general, obtain the most efficient mapping. If unsigned bus values are used in the code, the synthesis tools may still be able to use this resource, but might not obtain the full bit precision of the component due to the unsigned-to-signed conversion.

If the target design is expected to contain a large number of adders, Xilinx recommends that you evaluate the design to make greater use of the DSP48 slice pre-adders and post-adders. For example, with FIR filters, the adder cascade can be used to build a systolic filter rather than using multiple successive add functions (adder trees). If the filter is symmetric, you can evaluate using the dedicated pre-adder to further consolidate the function into both fewer LUTs and flip-flops and also fewer DSP slices as well (in most cases, half the resources).

If adder trees are necessary, the 6-input LUT architecture can efficiently create ternary addition ($A + B + C = D$) using the same amount of resources as a simple 2-input addition. This can help save and conserve carry logic resources. In many cases, there is no need to use these techniques.

By knowing these capabilities, the proper trade-offs can be acknowledged up front and accounted for in the RTL code to allow for a smoother and more efficient implementation from the start. In most cases, Xilinx recommends inferring DSP resources.

For more information about the features and capabilities of the DSP48 slice, and how to best leverage this resource for your design needs, see the *7 Series DSP48E1 Slice User Guide* ([UG479](#)) and *UltraScale Architecture DSP Slice User Guide* ([UG579](#)).

Coding Shift Registers and Delay Lines

In general, a shift register is characterized by some or all of the following control and data signals:

- Clock
- Serial input
- Asynchronous set/reset
- Synchronous set/reset
- Synchronous/asynchronous parallel load
- Clock enable
- Serial or parallel output

Xilinx devices contain dedicated SRL16 and SRL32 resources (integrated in LUTs). These allow efficiently implemented shift registers without using flip-flop resources. However, these elements support only LEFT shift operations, and have a limited number of I/O signals:

- Clock
- Clock Enable
- Serial Data In
- Serial Data Out

In addition, SRLs have address inputs (A3, A2, A1, A0 inputs for SRL16) determining the length of the shift register. The shift register can be a fixed static length or can be dynamically adjusted. In dynamic mode, each time a new address is applied to the address pins, the new bit position value is available on the Q output after the time delay to access the LUT.

Synchronous and asynchronous set/reset control signals are not available in the SRL primitives. However, if your RTL code includes a reset, the Xilinx synthesis tool infers additional logic around the SRL to provide the reset functionality.

To obtain the best performance when using SRLs, Xilinx recommends that you implement the last stage of the shift register in the dedicated slice register. Slice registers have a better clock-to-out time than SRLs. This allows additional slack for the paths sourced by the shift register logic. Synthesis tools automatically infer this register unless this resource is instantiated or the synthesis tool is prevented from inferring this type of register because of attributes or cross-hierarchy boundary optimization restrictions. To infer the extra register, register the dynamically delayed signal separately in the RTL.

Xilinx recommends that you use the HDL coding styles represented in the Vivado Design Suite HDL Templates.

When using registers to obtain placement flexibility in the chip, turn off SRL inference using the following attribute:

```
SHREG_EXTRACT = "no"
```

For more information about synthesis attributes and how to specify the attributes in the HDL code, see the *Vivado Design Suite User Guide: Synthesis* (UG901).

Initialization of All Inferred Registers, SRLs, and Memories

The GSR net initializes all registers to the specified initial value in the HDL code. If no initial value is supplied, the synthesis tool is at liberty to assign the initial state to either zero or one. Vivado synthesis generally defaults to zero with a few exceptions such as one-hot state machine encodings.

Any inferred SRL, memory, or other synchronous element may also have an initial state defined that will be programmed into the associated element upon configuration.

Xilinx highly recommends that you initialize all synchronous elements accordingly. Initialization of registers is completely inferable by all major device synthesis tools. This lessens the need to add a reset for the sole purpose of initialization, and makes the RTL code more closely match the implemented design in functional simulation, as all synchronous element start with a known value in the device after configuration.

Initial state of the registers and latches VHDL coding example one:

```
signal reg1 : std_logic := '0'; -- specifying register1 to start as a zero
signal reg2 : std_logic := '1'; -- specifying register2 to start as a one
signal reg3 : std_logic_vector(3 downto 0):="1011"; -- specifying INIT
value for
4-bit register
```

Initial state of the registers and latches Verilog coding example two:

```
reg register1 = 1'b0; // specifying register1 to start as a zero
reg register2 = 1'b1; // specifying register2 to start as a one
reg [3:0] register3 = 4'b1011; //specifying INIT value for 4-bit register
```

Another possibility in Verilog is to use an `initial` statement:

```
reg [3:0] register3;
initial begin
    register3= 4'b1011;
end
```

Deciding When to Instantiate or Infer

Xilinx recommends that you have an RTL description of your design; and that you let the synthesis tool do the mapping of the code into the resources available in the device. In addition to making the code more portable, all inferred logic is visible to the synthesis tool, allowing the tool to perform optimizations between functions. These optimizations include logic replications; restructuring and merging; and retiming to balance logic delay between registers.

Synthesis Tool Optimization

When device library cells are instantiated, synthesis tools do not optimize them by default. Even when instructed to optimize the device library cells, synthesis tools generally cannot perform the same level of optimization as with the RTL. Therefore, synthesis tools typically only perform optimizations on the paths to and from these cells but not through the cells.

For example, if an SRL is instantiated and is part of a long path, this path might become a bottleneck. The SRL has a longer clock-to-out delay than a regular register. To preserve the area reduction provided by the SRL while improving its clock-to-out performance, an SRL of one delay less than the actual desired delay is created, with the last stage implemented in a regular flip-flop.

When Instantiation Is Desirable

Instantiation might be desirable when the synthesis tool mapping does not meet the timing, power, or area constraints; or when a particular feature within a device cannot be inferred.

With instantiation, you have total control over the synthesis tool. For example, to achieve better performance, you can implement a comparator using only LUTs, instead of the combination of LUT and carry chain elements usually chosen by the synthesis tool.

Sometimes instantiation may be the only way to make use of the complex resources available in the device. This can be due to:

- HDL Language Restrictions

For example, it is not possible to describe double data rate (DDR) outputs in VHDL because it requires two separate processes to drive the same signal.

- Hardware Complexity

It is easier to instantiate the I/O SerDes elements than to create synthesizable description.

- Synthesis Tools Inference Limitations

For example, synthesis tools currently do not have the capability to infer the hard FIFOs from RTL descriptions. Therefore, you must instantiate them.

If you decide to instantiate a Xilinx primitive, see the appropriate User Guide and Libraries Guide for the target architecture to fully understand the component functionality, configuration, and connectivity.

In case of both inference as well as instantiation, Xilinx recommends that you use the instantiation and language templates from the Vivado Design Suite language templates.

Following are tips:

- Infer functionality whenever possible.
- When synthesized RTL code does not meet requirements, review the requirements before replacing the code with device library component instantiations.

- Consider the Vivado Design Suite language templates when writing common Verilog and VHDL behavioral constructs or if necessary instantiating the desired primitives.

Coding Styles to Improve Maximum Frequency

For high performance designs, the coding techniques discussed in this section can mitigate possible timing hazards.

High Fanouts in Critical Paths

High fanout nets are much easier to deal with early in the design process. What constitutes too high of a fanout is often dictated by performance requirements and the construction of the paths. You can use the following techniques to address issues with high fanout nets.



RECOMMENDED: Identify high fanout nets using the `report_high_fanout_nets` Tcl command after synthesis. Monitor the impact of these nets on design performance as you progress through the implementation process.

Reduce Loads in Portions of the Design That Do Not Require It

For high fanout control signals, evaluate whether all coded portions of the design require that net. Reducing the number of loads can greatly reduce timing problems.

Replicate High Fanout Net Drivers

Register replication can increase the speed of critical paths by making copies of registers to reduce the fanout of a given signal. This gives the implementation tools more flexibility in placing and routing the different loads and associated logic. Synthesis tools use this technique extensively.

Most synthesis tools use a fanout threshold limit to automatically determine whether to duplicate a register. Lowering this global threshold allows automatic duplication of high fanout nets. However, it does not allow control over which registers are duplicated or how their loads are grouped. In addition, the global replication mechanism does not assess timing slack accurately, which can lead to unnecessary replicated cells, logic utilization increase, and potentially higher power consumption.

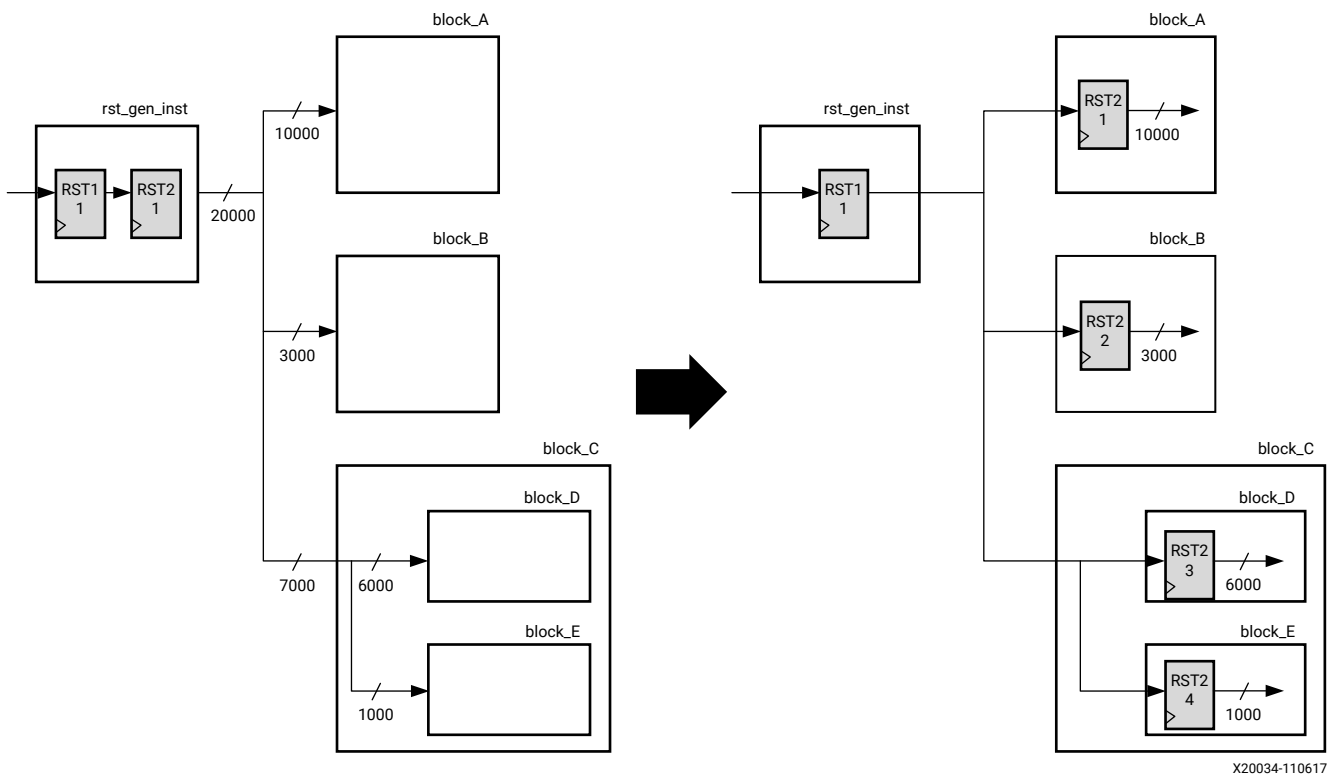
For high frequency designs, a better approach to reducing fanout is to use a balanced tree for the high fanout signals. Consider manually replicating registers based on the design hierarchy, because the cells included in a hierarchy are often placed together. For example, in the balanced reset tree shown in the following figure, the high fanout reset FF RST2 is replicated in RTL to balance the fanout across the different modules. If required, physical synthesis can perform further replication to improve WNS based on placement information.



TIP: To preserve the duplicate registers in synthesis, use a `KEEP` attribute instead of `DONT_TOUCH`. A `DONT_TOUCH` attribute prevents further optimization during physical optimization later in the implementation flow.

Note: If a LUT1 rather than a register is replicated, it indicates that an attribute or constraint is applied incorrectly.

Figure 34: High Fanout Reset Transformed to Balanced Reset Tree



X20034-110617



RECOMMENDED: Using `MAX_FANOUT` attributes on global high fanout signals leads to suboptimal replication similar to when the global fanout limit is lowered in synthesis. For this reason, Xilinx recommends only using `MAX_FANOUT` inside the hierarchies on local signals with medium to low fanout.

Do not replicate registers used for synchronizing signals that cross clock domains. The presence of the `ASYNC_REG` attribute on these registers prevents the tool from replicating these registers. If the synchronizing chain has a very high fanout and replication must meet timing, add an extra register after the synchronization chain that does not have the `ASYNC_REG` constraint.

The following table provides guidelines on the number of fanouts that might be acceptable for your design.

Table 1: Fanout Guidelines for Medium Performance 7 Series Devices

Condition	Fanout > 5000	Fanout > 200	Fanout > 100
Low Frequency 1 to 125 MHz	Few logic levels between synchronous logic <13 levels of logic at maximum frequency	N/A	N/A
Medium Frequency 125 to 250 MHz	If the design does not meet timing, you might need to reduce fanout and/or logic levels.	<6 levels of logic at maximum frequency. (Driver and load types impact performance.)	N/A
High Frequency > 250 MHz	Not recommended for most designs.	Small number of logic levels is typically necessary for higher speeds.	Advance pipelining methods required. Careful logic replication. Compact functions. Low logic levels required. (Driver and load types impact performance.)



TIP: If the timing reports indicate that high-fanout signals are limiting the design performance, consider replicating the signals using the implementation tool options, such as `opt_design - hier_fanout_limit`, `place_design`, and `phys_opt_design`.



TIP: When replicating registers, consider using a naming convention for the registers, such as `<original_name>_a`, `<original_name>_b`, etc., to make it easier to understand intent of the replication and easier to maintain the RTL code.

Pipelining Considerations

Another way to increase performance is to restructure long datapaths with several levels of logic and distribute them over multiple clock cycles. This method allows for a faster clock cycle and increased data throughput at the expense of latency and pipeline overhead logic management.

Because devices contain many registers, the additional registers and overhead logic are usually not an issue. However, the datapath spans multiple cycles, and you must make special considerations for the rest of the design to account for the added path latency.

Consider Pipelining for SSI Devices

When designing high performance register-to-register connections for SLR boundary crossings, the appropriate pipelining must be described in the HDL code and controlled at synthesis. This ensures that the shift register LUT (SRL) inference and other optimizations do not occur in the logic path that must cross an SLR boundary. Modifying the code in this manner along with appropriate use of Pblocks defines where the SLR boundary crossing occurs.

Consider Pipelining Up Front

Considering pipelining up front rather than later on can improve timing closure. Adding pipelining at a later stage to certain paths often propagates latency differences across the circuit. This can make one seemingly small change require a major redesign of portions of the code.

Identifying pipelining opportunities early in the design can often significantly improve timing closure, implementation runtime (due to easier-to-solve timing problems), and device power (due to reduced switching of logic).

Check Inferred Logic

As you code your design, be aware of the logic being inferred. Monitor the following conditions for additional pipelining considerations:

- Cones of logic with large fanin

For example, code that requires large buses or several combinational signals to compute an output.
- Blocks with restricted placement or slow clock-to-out or large setup requirements

For example, block RAMs without output registers or arithmetic code that is not appropriately pipelined.
- Forced placement that causes long routes

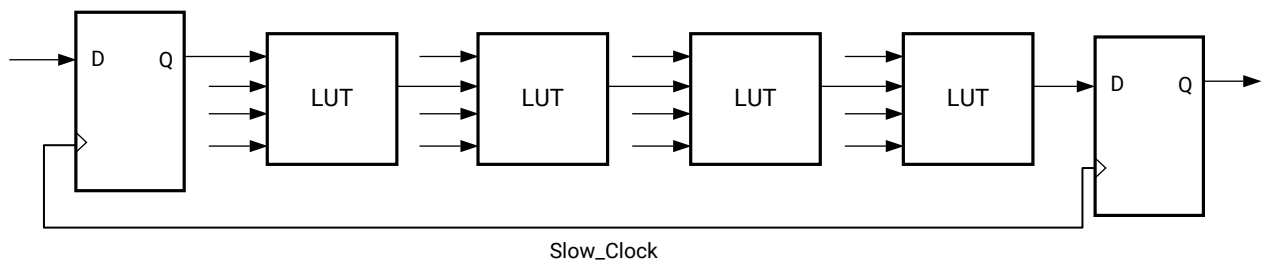
For example, a pinout that forces a route across the chip might require pipelining to allow for high-speed operation.
- Logic comprised of large XOR functions

Large XOR functions often have high switch rates that can generate large dynamic power dissipation. Pipelining these functions can reduce switching, which positively impacts power consumption of the described circuit.

In the following figure the clock speed is limited by:

- Clock-to out-time of the source flip-flop
- Logic delay through four levels of logic
- Routing associated with the four function generators
- Setup time of the destination register

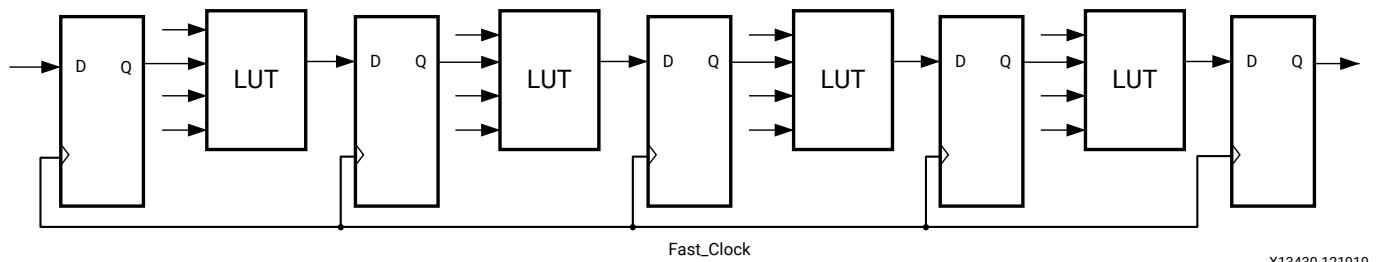
Figure 35: Before Pipelining Diagram



X13429-122019

The following figure is an example of the same data path shown in the Before Pipelining diagram. Because the flip-flop is contained in the same slice as the function generator, the clock speed is limited by the clock-to-out time of the source flip-flop, the logic delay through one level of logic, one routing delay, and the setup time of the destination register. In this example, the system clock runs faster after pipelining than before pipelining.

Figure 36: After Pipelining Diagram



X13430-121919

Determine Whether Pipelining is Needed

A commonly used pipelining technique is to identify a large combinatorial logic path, break it into smaller paths, and introduce a register stage between these paths, ideally balancing each pipeline stage.

To determine whether a design requires pipelining, identify the frequency of the clocks and the amount of logic distributed across each of the clock groups. You can use the `report_design_analysis` Tcl command with the `-logic_level_distribution` option to determine the logic-level distribution for each of the clock groups.



TIP: The design analysis report also highlights the number of paths with zero logic levels, which you can use to determine where to make modifications in your code.

Balance Latency

To balance the latency by adding pipeline stages, add the stage to the control path and not the data path. The data path includes wider buses, which increases the number of flip-flop and register resources used.

For example, if you have a 128-bit data path, 2 stages of registers, and a requirement of 5 cycles of latency, inserting 3 register stages results in an extra $3 \times 128 = 384$ flip-flops. Alternatively, you can use registers to control logic to enable the data path. Use 5 stages of single-bit registers to control the enable signal of datapath flip-flops and multicycle path timing exceptions accordingly.

Note: This example is only possible for certain designs. For example, in cases where there is a fanout from the intermediate data path flip-flops, having only 2 stages does not work.



RECOMMENDED: The optimal LUT:FF ratio in a device is 1:1. Designs with significantly more FFs will increase unrelated logic packing into slices, which will increase routing complexity and can degrade QoR.

Balance Pipeline Depth and SRL Usage

When there are deep register pipelines, map as many registers as possible into the SRLs to avoid significant increases in register utilization. For example, a 9-deep pipeline for a data width of 32 results in 9 registers for each bit, which uses $32 \times 9 = 288$ registers. Mapping the same structure to SRLs uses 32 SRLs. Each SRL has address pins A4 through A0 connected to 5'b01000 to implement a depth of 9 stages.

There are multiple ways to infer SRLs during synthesis, including the following:

- SRL
- REG -> SRL
- SRL -> REG
- REG -> SRL -> REG

You can create these structures using the `srl_style` attribute in the RTL code as follows:

- `(* srl_style = "srl" *)`
- `(* srl_style = "reg_srl" *)`
- `(* srl_style = "srl_reg" *)`
- `(* srl_style = "reg_srl_reg" *)`

A common mistake is to use different enable/reset control signals in deeper pipeline stages. Following is an example of a reset used in a 9-deep pipeline stage with the reset connected to the third, fifth, and eighth pipeline stages. With this structure, the tools map the pipeline stages to registers only, because there is a reset pin on the SRL primitive.

```
FF->FF->FF(reset) -> FF->FF(reset)->FF->FF->FF(reset)->FF
```

To take advantage of SRL inference:

- Ensure there are no resets for the pipeline stages.
- Analyze whether the reset is really required.
- Use the reset on one flip-flop (for example, on the first or last stage of the pipeline).

Avoid Unnecessary Pipelining

For highly utilized designs, too much pipelining can lead to suboptimal results. For example, unnecessary pipeline stages increase the number of flip-flops and routing resources, which might limit the place and route algorithms if the utilization is high.

Note: If there are many paths with 0/1 levels of logic, check to make sure this is intentional.

Consider Pipelining Macro Primitives

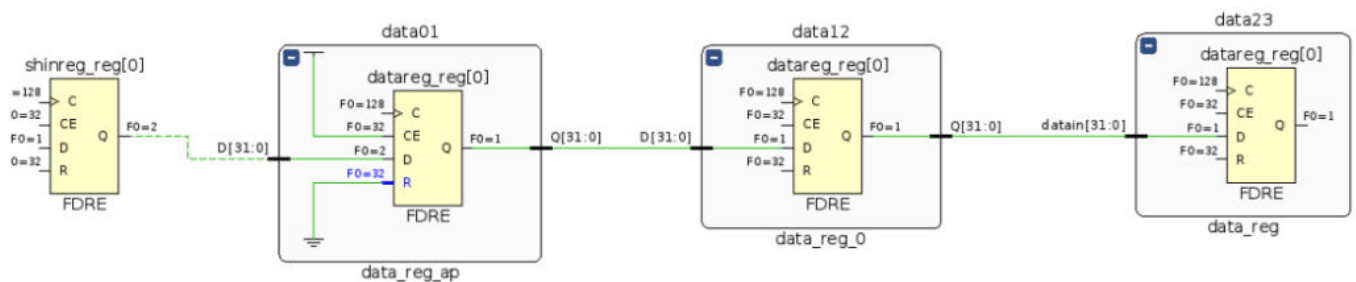
Based on the target architecture, dedicated primitives such as block RAMs and DSPs can work at over 500 MHz if enough pipelining is used. For high frequency designs, Xilinx recommends using all of the pipelines within these blocks.

Auto-Pipelining Considerations

The auto-pipelining feature allows the placer to determine the number of required pipeline stages and their optimal location, which helps timing closure across interface boundaries. You can enable this feature by setting up the auto-pipelining mode of the AXI Register Slice core or by applying the auto-pipelining HDL attribute or XDC constraints for data buses. Because the insertion is timing-driven, always be sure to apply proper timing constraints on the targeted paths. For more information, see this [link](#) in the *Vivado Design Suite User Guide: Implementation (UG904)*.

The following example shows auto-pipelining applied on the interface between the module `data01` and `data12`. The output from `data01` consists of registers with no control sets.

Figure 37: Simple Data Flow Connections Between Modules



Following is the RTL code for this example. The `autopipeline_module` attribute is applied on the hierarchical module `data01`, and the `autopipeline_group/autopipeline_limit/autopipeline_include` attributes are applied on the nets directly driven by the Q pins of the registers.

```
data_reg_ap #( .C_DATA_WIDTH(C_DATA_WIDTH) ) data01 (
    .clk (clk),
    .datain (shinreg),
    .datareg (d1)
);

data_reg #( .C_DATA_WIDTH(C_DATA_WIDTH) ) data12 (
    .clk (clk),
    .datain (d1),
    .datareg (d2)
);

(* autopipeline_module="yes" *)
module data_reg_ap # (
    parameter integer C_DATA_WIDTH = 32
```

```

)
( input wire clk,
input wire [C_DATA_WIDTH-1:0] datain,
(* autopipeline_group="fwd",autopipeline_limit=24 *)
output reg [C_DATA_WIDTH-1:0] datareg
);

always @(posedge clk) begin
datareg <= datain;
end
endmodule
    
```

Following are the XDC constraints for this example, which is an alternative approach to using attributes in the RTL code.

```

# It's suggested to add the USER_SLR_ASSIGNMENT property at the module
level to ensure better logic clustering with its driver and load, see UG912
for more details on this property
set_property USER_SLR_ASSIGNMENT APSRC [get_cells data01]
set_property USER_SLR_ASSIGNMENT APDST [get_cells data12]

set_property AUTOPIPELINE_MODULE TRUE [get_cells data01]
set_property AUTOPIPELINE_GROUP WBUS [get_nets -of [get_pins -filter
REF_PIN_NAME==Q -of [get_cells data01/*]]]
set_property AUTOPIPELINE_LIMIT 10 [get_nets -of [get_pins -filter
REF_PIN_NAME==Q -of [get_cells data01/*]]]
    
```

Coding Styles to Improve Power

Gate Clock or Data Paths

Gating the clock or data paths is a common technique to stop transition when the results of these paths are not used. Gating a clock stops all driven synchronous loads and prevents data path signal switching and glitches from continuing to propagate.

Power optimization (`power_opt_design`) can automatically generate signal gating logic to reduce switching activity. However, you have information about the application, data flow, and dependencies that is not available to the tool, which only you can specify.

Maximize Gating Elements

Maximize the number of elements affected by the gating signal. For example, it is more power efficient to gate a clock domain at its driving source than to gate each load with a clock enable signal.

Use Clock Enable Pins of Dedicated Clock Buffers

When gating or multiplexing clocks to minimize activity or clock tree usage, use the clock enable ports of dedicated clock buffers. Inserting LUTs or using other methods to gate-off clock signals is not efficient for power and timing.

Use Case Block When Priority Encoder Not Needed

When a priority encoding is not needed, use a case block instead of an if-then-else block or ternary operator.

Inefficient coding example:

```
if (reg1)
    val = reg_in1;
else if (reg2)
    val = reg_in2;
else if (reg3)
    val = reg_in3;
else val = reg_in4;
```

Correct coding example:

```
(* parallel_case *) casex ({reg1, reg2, reg3})
1xx: val = reg_in1 ;
01x: val = reg_in2 ;
001: val = reg_in3 ;
default: val = reg_in4 ;
endcase
```

Performance/Power Trade-Off for Block RAMs

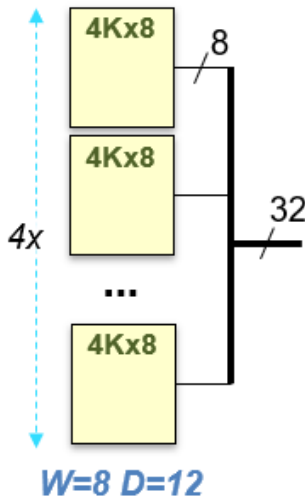
There are multiple ways of breaking a memory configuration to serve a particular requirement. The requirement for a particular design can be performance, power, or a mixture of both.

The following example highlights the different structures that can be generated to achieve your requirements. Synthesis can limit the cascading of the block RAM for the performance/power trade-off using the `CASCADE_HEIGHT` attribute. The usage and arguments for the attribute are described in the *Vivado Design Suite User Guide: Synthesis* ([UG901](#)).

The following figure shows an example of 8Kx32 memory configuration for higher performance (timing).

Note: This example applies to UltraScale and UltraScale+ devices only.

Figure 38: RTL Representation of 4Kx32 Using 4Kx8 and CASCADE_HEIGHT=1



```

module test(
input clk,
input we,
input [31:0] din,
input [11:0] addr,
output reg [31:0] dout
);

(* ram_style = "block", cascade height = 1 *)
reg [31:0] mem [(2**12)-1:0];
reg [11:0] addr_reg;

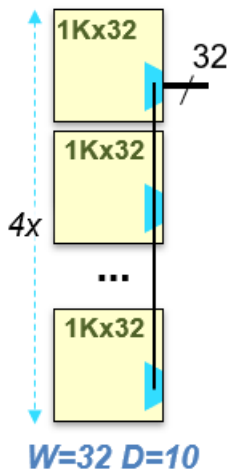
always @(posedge clk)
begin
    addr_reg <= addr;
    dout <= mem[addr_reg];
    if (we)
        mem[addr_reg] <= din;
end

endmodule
    
```

In this implementation, all block RAMs are always enabled (for each read or write) and consume more power.

The following figure shows an example of cascading all the block RAMs for low power.

Figure 39: RTL Representation of 4Kx32 Using 1Kx32 and CASCADE_HEIGHT=4



```

module test(
input clk,
input we,
input [31:0] din,
input [11:0] addr,
output reg [31:0] dout
);

(* ram_style = "block", cascade height = 4 *)
reg [31:0] mem [(2**12)-1:0];
reg [11:0] addr_reg;

always @(posedge clk)
begin
    addr_reg <= addr;
    dout <= mem[addr_reg];
    if (we)
        mem[addr_reg] <= din;
end

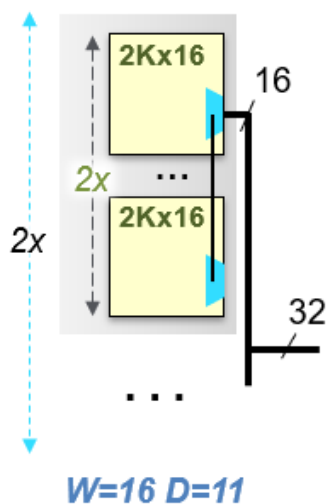
endmodule
    
```

In this implementation, because one block RAM at a time is selected (from each unit), the dynamic power contribution is almost half. Block RAMs have a dedicated cascade MUX and routing structure that allows the construction of wide, deep memories requiring more than one block RAM primitive to be built in a very power efficient configuration.

The following figure shows an example of how to limit the cascading and gain both power and performance at the same time, often with no trade-off in performance.

Note: This example applies to UltraScale and UltraScale+ devices only.

Figure 40: RTL Representation of 4Kx32 Using 2Kx16 and CASCADE_HEIGHT=2



```

module test(
input clk,
input we,
input [31:0] din,
input [11:0] addr,
output reg [31:0] dout
);

(* ram_style = "block", cascade height = 2 *)
reg [31:0] mem [(2**12)-1:0];
reg [11:0] addr_reg;

always @(posedge clk)
begin
    addr_reg <= addr;
    dout <= mem[addr_reg];
    if (we)
        mem[addr_reg] <= din;
end

endmodule
    
```

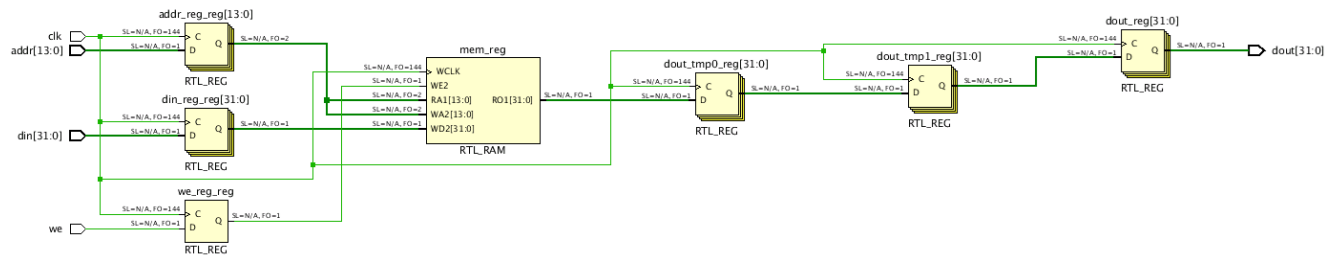
Because two block RAMs are selected at a time in this implementation, the dynamic power contribution is better than for the high performance structure, but not as good as for the low power structure. The advantage with this structure compared to a low power structure is that it uses only two block RAMs in the cascaded path, which has impact on the target frequency when compared to four block RAMs in the critical path for the low power structure.

Decomposing Deeper Memory Configurations for Balanced Power and Performance

When working with deeper memory configurations, you can use the `RAM_DECOMP` synthesis attribute in the RTL to reduce power by improving memory composition. When the `RAM_DECOMP` attribute is applied to a memory array, the memory logic is mapped to a wider array of block RAM primitives. To balance power and performance, you can control cascading using the `CASCADE_HEIGHT` attribute along with the `RAM_DECOMP` attribute. This approach requires more address decoding logic but helps to reduce the number of block RAMs that are enabled for each read operation, which helps to reduce power.

For example, the following figure shows a 32x16K memory configuration.

Figure 41: 32x16K Memory Configuration



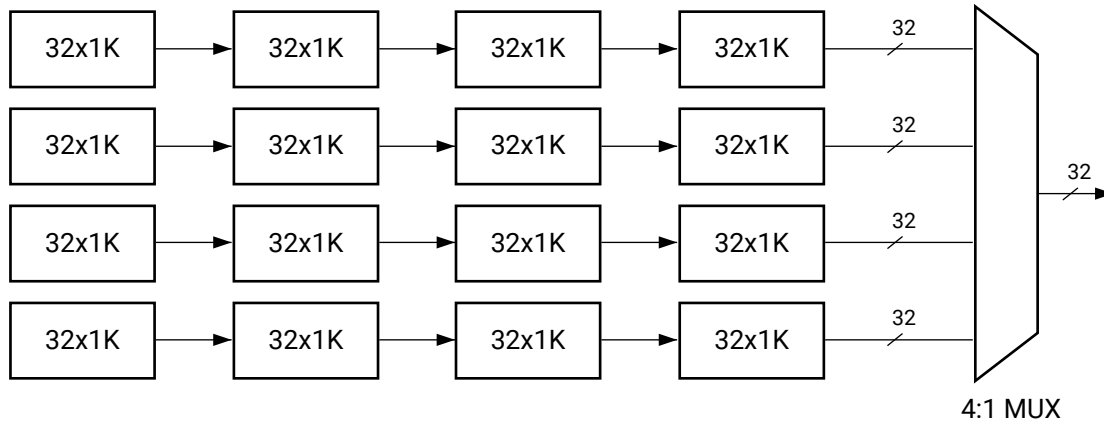
If you apply the following attributes:

```
ram_decomp = "power"
cascade_height = 4
```

16 RAMB36E2 is inferred and the memory is decomposed as follows:

- The base primitive is 32x1K.
- 4 block RAMs are cascaded to create a 32x4K configuration.
- 4 parallel structures create a 16K deep memory.
- The outputs are multiplexed to generate the output data.

Figure 42: Generated Structure for 32x16K Memory Configuration Example Using CASCADE_HEIGHT and RAM_DECOMP Attributes



X19283-121919

The following RTL code example shows the use of the CASCADE_HEIGHT and RAM_DECOMP attributes.

Figure 43: RTL Code for 32x16K Memory Configuration Using the CASCADE_HEIGHT and RAM_DECOMP Attributes

```

module test
(
    input clk,
    input we,
    input [13:0] addr,
    input [31:0] din,
    output reg [31:0] dout
);

(* ram_style = "block", ram_decomp = "power", cascade_height = 4 *) reg [31:0] mem [(16*1024)-1:0];
reg [13:0] addr_reg;
reg [31:0] dout_tmp0;
reg [31:0] dout_tmp1;
reg [31:0] din_reg;
reg we_reg;

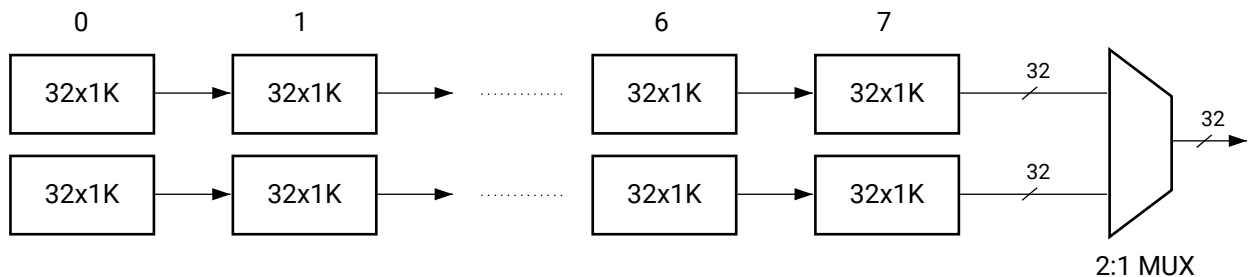
always @(posedge clk)
begin
    addr_reg <= addr;
    din_reg <= din;
    we_reg <= we;
    dout_tmp0 <= mem[addr_reg];
    dout_tmp1 <= dout_tmp0;
    dout <= dout_tmp1;
    if (we_reg)
        mem[addr_reg] <= din_reg;
end
endmodule
    
```

If you apply only the `ram_decomp = "power"` attribute, 16 RAMB36E2 are inferred and the memory is decomposed as follows:

- The base primitive is 32x1K.
- 8 block RAMs are cascaded to create a 32x8K configuration.
- 2 parallel structures create a 16K deep memory.

- The outputs are multiplexed into a 2:1 MUX to generate the output data.

Figure 44: **Generated Structure for 32x16K Memory Configuration Using the RAM_DECOMP Attribute**



X19284-050517

The following RTL code example shows the use of the RAM_DECOMP attribute.

Figure 45: **RTL Code for 32x16K Memory Configuration Using the RAM_DECOMP Attribute**

```

module test
(
    input clk,
    input we,
    input [13:0] addr,
    input [31:0] din,
    output reg [31:0] dout
);

(* ram_style = "block", ram_decomp = "power"*) reg [31:0] mem [(16*1024)-1:0];
reg [13:0] addr_reg;
reg [31:0] dout_tmp0;
reg [31:0] dout_tmp1;
reg [31:0] din_reg;
reg we_reg;

always @(posedge clk)
begin
    addr_reg <= addr;
    din_reg <= din;
    we_reg <= we;
    dout_tmp0 <= mem[addr_reg];
    dout_tmp1 <= dout_tmp0;
    dout <= dout_tmp1;
    if (we_reg)
        mem[addr_reg] <= din_reg;
end

endmodule
    
```

If you use only the RAM_DECOMP attribute, the overall power savings is similar to using both the RAM_DECOMP and CASCADE_HEIGHT attributes together, because only one block RAM is active at a time. Creating a 4-deep cascaded block RAM chain is better for performance when compared to an 8-deep cascaded block RAM chain.

For more information, see this [link](#) in the *Vivado Design Suite User Guide: Synthesis (UG901)*.

Running RTL DRCs

A set of RTL DRC rules identify potential coding issues with your HDL. You can perform these checks on the elaborated views, which you can open by clicking **Open Elaborated Design** in the Flow Navigator. You can run these DRC checks by selecting **RTL Analysis → Report Methodology** in the Flow Navigator or by executing `report_methodology` at the Tcl command prompt.

Clocking Guidelines

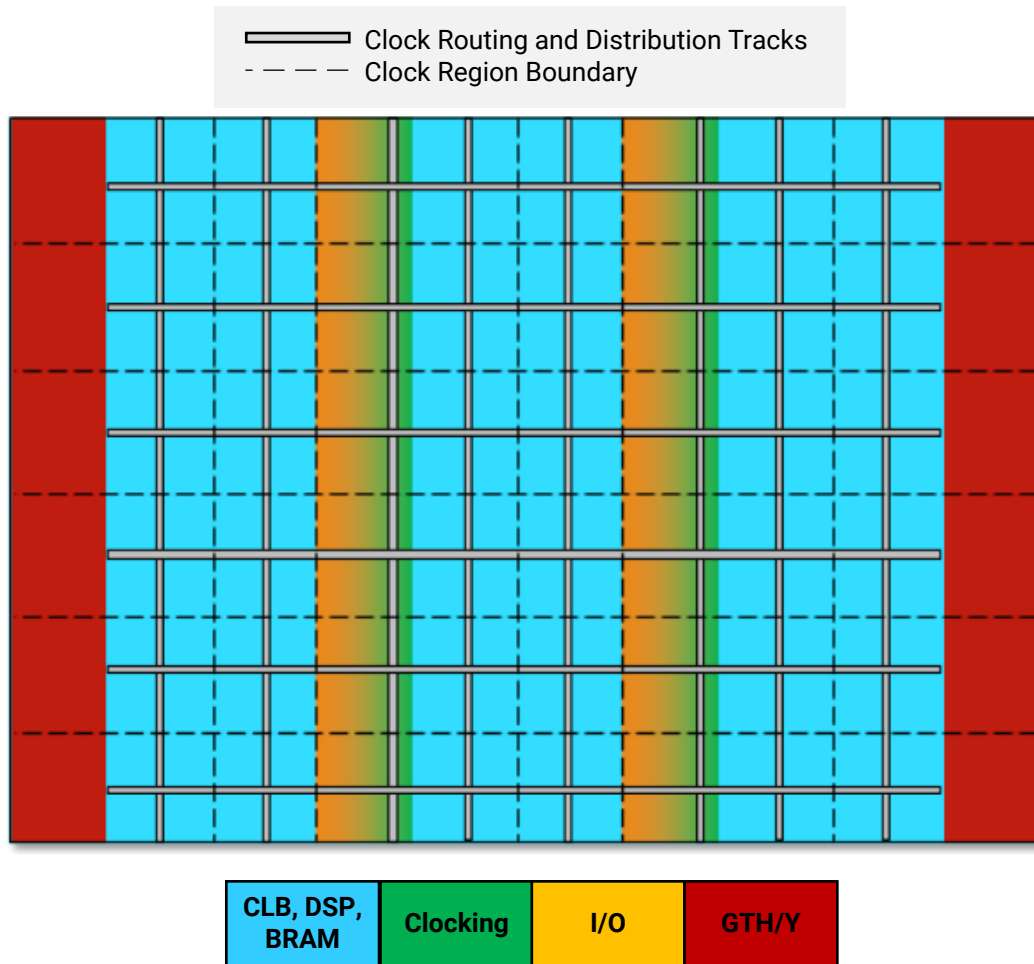
Each device architecture has some dedicated resources for clocking. Understanding the clocking resources for your device architecture can allow you to plan your clocking to best utilize those resources. Most designs might not need you to be aware of these details. However, if you can control the placement and have a good idea of the fanout on each of the clocking domains, you can explore alternatives based on the following clocking details. If you decide to exploit any of these clocking resources, you need to explicitly instantiate the corresponding clocking element.

UltraScale Device Clocking

UltraScale devices have a different clocking structure from previous device architectures, which blurs the line between global versus regional clocking. UltraScale devices do not have regional clock buffers like 7 series devices and instead use a common buffer and clock routing structure whether the loads are local/regional or global.

UltraScale devices feature smaller clock regions of a fixed size across devices, and the clock regions no longer span half of the device width in the horizontal direction. The number of clock regions per row varies per UltraScale device. Each clock region contains a clock network routing that is divided into 24 vertical and horizontal routing tracks and 24 vertical and horizontal distribution tracks. The following figure shows a device with 36 clock regions (6 columns x 6 rows). The equivalent 7 series device has 12 clock regions (2 columns x 6 rows).

Figure 46: UltraScale Device Clock Region Tiles



X15241-122019

The clocking architecture is designed so that only the clock resources necessary to connect clock buffers and loads for a given placement are used, and no resource is wasted in clock regions with no loads. The efficient clock resource utilization enables support for more design clocks in the architecture while improving clock characteristics for performance and power. Following are the main categories of clock types and associated clock structures grouped by their driver and use:

- High-Speed I/O Clocks

These clocks are associated with the high-speed SelectIO™ interface bit slice logic, generated by the PLL, and routed via dedicated, low-jitter resources to the bit slice logic for high-speed I/O interfaces. In general, this clocking structure is created and controlled by Xilinx IP, such as memory IP or the High Speed SelectIO Wizard, and is not user specified.

- General Clocks

These clocks are used in most clock tree structures and can be sourced by a GCIO package pin, an MMCM/PLL, or fabric logic cells (not generally suggested). The general clocking network must be driven by BUFGCE/BUFGCE_DIV/BUFGCTRL buffers, which are available in any clock region that contains an I/O column. Any given clock region can support up to 24 unique clocks, and most UltraScale devices can support over 100 clock trees depending on their topology, fanout, and load placement.

- Gigabit Transceiver (GT) Clocks

Transmit, receive, and reference clocks of gigabit transceivers (GTH or GTY) use dedicated clocking in the clock regions that include the GTs. You can use GT clocks to achieve the following:

- Drive the general clocking network using the BUFG_GT buffers to connect any loads in the fabric
- Share clocks across several transceivers in the same or different Quad

Clock Primitives

Most clocks enter the device through a global clock-capable I/O (GCIO) pin. These clocks directly drive the clock network via a clock buffer or are transformed by a PLL or MMCM located in the clock management tile (CMT) adjacent to the I/O column.

The CMT contains the following clocking resources:

- Clock generation blocks
 - 2 PLLs
 - 1 MMCM
- Global clock buffers
 - 24 BUFGCEs
 - 8 BUFGCTRLs
 - 4 BUFGCE_DIVs

Note: Clocking resources in CMTs that are adjacent to I/O columns with unbonded I/Os are available for use.

The GT user clocks drive the global clock network via BUFG_GT buffers. There are 24 BUFG_GT buffers per clock region adjacent to the GTH/GTY columns.

Following is summary information for each of the UltraScale device clock buffers:

- BUFGCE

The most commonly used buffer is the BUFGCE. This is a general clock buffer with a clock enable/disable feature equivalent to the 7 series BUFHCE.

- BUFGCE_DIV

The BUFGCE_DIV is useful when a simple division of the clock is required. It is considered easier to use and more power efficient than using an MMCM or PLL for simple clock division. When used properly, it can also show less skew between clock domains as compared to an MMCM or PLL when crossing clock domains. The BUFGCE_DIV is often used as replacement for the BUFR function in 7 series devices. However, because the BUFGCE_DIV can drive the global clock network, it is considered more capable than the BUFR component.

- BUFGCTRL (also BUFGMUX)

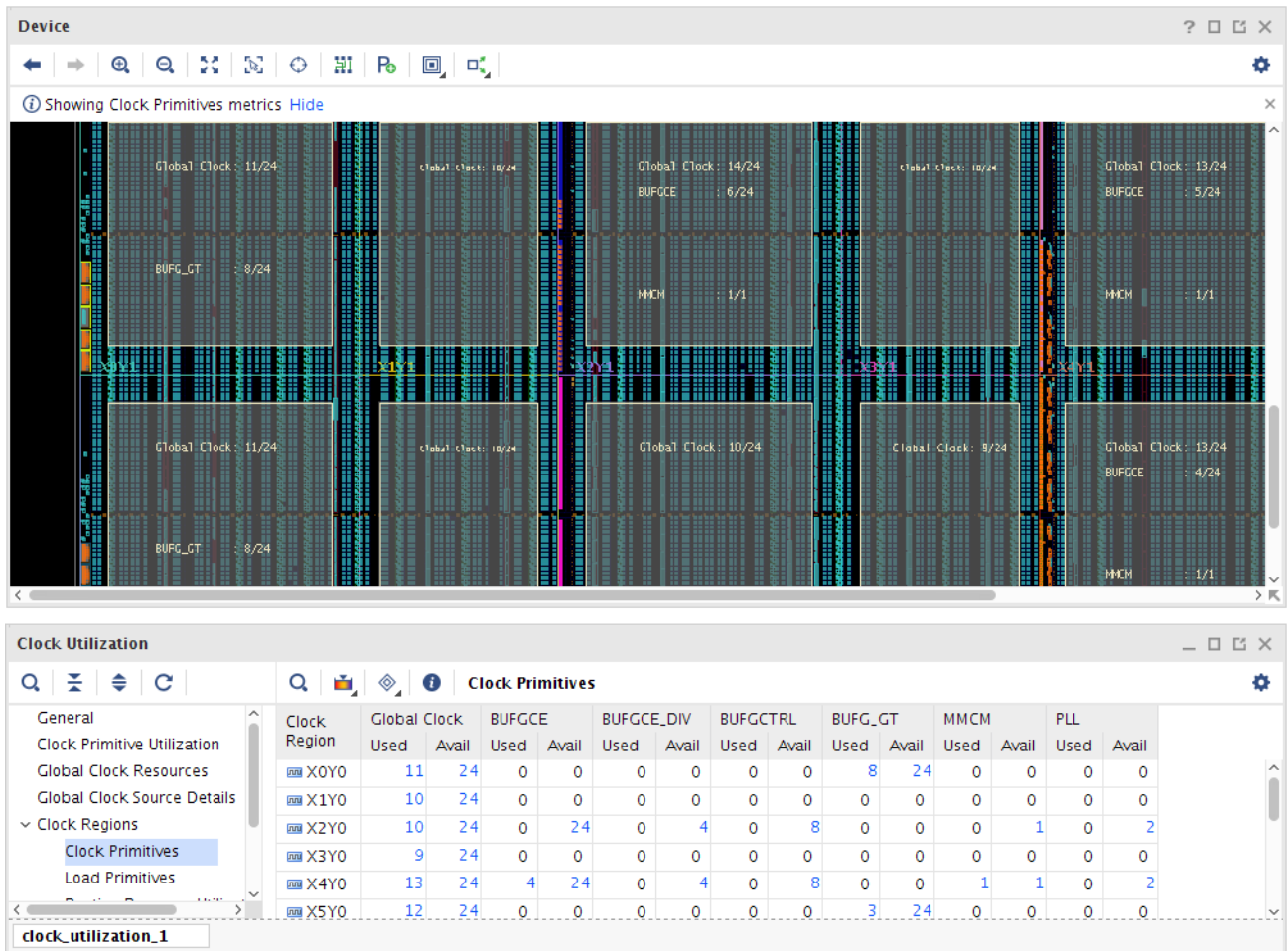
The BUFGCTRL can be instantiated as a BUFGMUX and is generally used when multiplexing two or more clock sources to a single clock network. As with the BUFGCE and BUFGCE_DIV, it can drive the clock network for either regional or global clocking.

- BUFG_GT

When using clocks generated by GTs, the BUFG_GT clock buffer allows connectivity to the global clock network. In most cases, the BUFG_GT is used as a regional buffer with its loads placed in one or two adjacent clock regions. The BUFG_GT has built-in dynamic clock division capability that you can use in place of an MMCM for clock rate changes.

You can use the Clock Utilization Report in the Vivado IDE to visually analyze clocking resource utilization and clock routing. The following figure shows the clock resource utilization per clock region overlaid in the Device window. For more information on this report, see the *Vivado Design Suite User Guide: Design Analysis and Closure Techniques* ([UG906](#)).

Figure 47: Clock Utilization Report



For more information on the BUFCE, BUFCE_DIV, and BUFCTRL buffers, see the *UltraScale Architecture Clocking Resources User Guide (UG572)*. For details on connectivity and use of the BUF_GT buffer, see the appropriate UltraScale Architecture Transceiver User Guide:

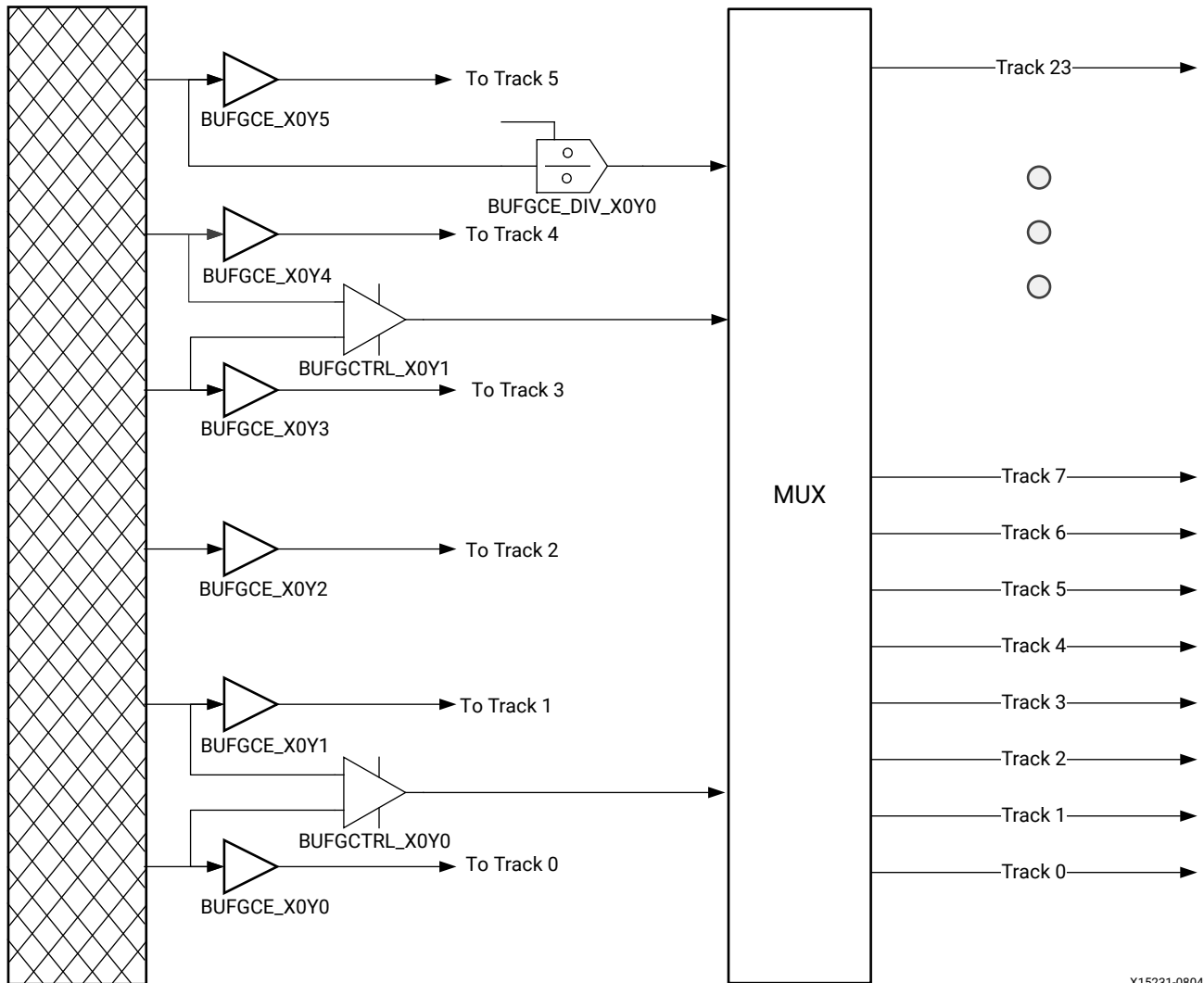
- *UltraScale Architecture GTH Transceivers User Guide (UG576)*
- *UltraScale Architecture GTY Transceivers User Guide (UG578)*

Global Clock Buffer Connectivity and Routing Tracks

Each of the 24 BUFCE buffers in a clock region can only drive a specific clock routing track. However, the BUFCTRL and BUFCE_DIV outputs can use any of the 24 tracks by going through a MUX structure. Each BUFCE_DIV shares the input connectivity with a specific BUFCE site, and each BUFCTRL shares input connectivity with two specific BUFCE sites. Consequently, when BUFCE_DIV or BUFCTRL buffers are used in the clock region, use of the BUFCE buffers is limited. The following figure shows the bottom 6 BUFCE in a clock region, which are replicated 4 times within a clock region.

Note: A global clock net is assigned to a specific track ID in the device for all the vertical, horizontal routing, and distribution resources the clock uses. A clock *cannot* change track IDs unless the clock goes through another clock buffer.

Figure 48: BUFGCE, BUFGCE_DIV, and BUFGCTRL Shared Inputs and Output Multiplexing



X15231-080420

Clock Routing, Root, and Distribution

To properly understand the clocking capacity of an UltraScale device and the clocking utilization of a design, it is important to know how the clock routes use the dedicated routing resources:

- From the clock buffer to the clock root, the clock signal goes through one or several segments of vertical and horizontal routing. Each segment must use the same track ID (between 0 and 23).

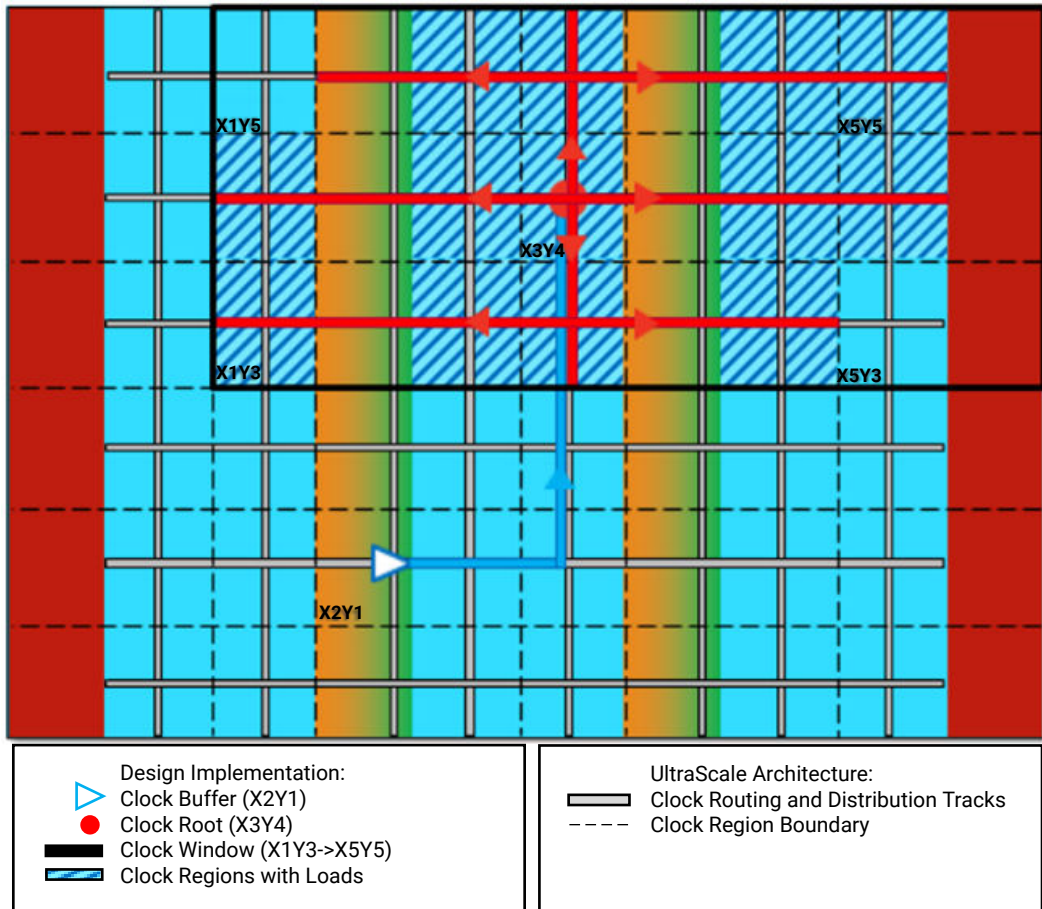
- At the clock root, the clock signal transitions from the routing track to the distribution track with the same track ID. To reduce skew, the clock root is usually in the clock region located in the center of the clock window. The clock window is the rectangular area that includes all the clock regions where the clock net loads are placed. For skew optimization reasons, the Vivado IDE might move the clock root to off center.
- From the clock root to the CLB columns where the loads are located, the clock signal travels on the vertical distribution (both up and down the device as needed) and then onto the horizontal distribution (both to the left and right as needed).
- The CLB columns are split into two halves, which are located above and below the horizontal distribution resources. Each half of the CLB column contains several leaf clock routing resources that can be reached by any of the horizontal distribution tracks.

In some cases, a clock buffer can directly drive onto the clock distribution track. This usually happens when the clock root is located in the same clock region as the clock buffer or when the clock buffer only drives non-clock pins (for example, high fanout nets).

Because clock routing resources are segmented, only the routing and distribution segments used to traverse a clock region or to reach a load in a clock region are consumed.

The following figure shows how a clock buffer located in clock region X2Y1 reaches its loads placed inside the clock window, which is formed by a rectangle of clock regions from X1Y3 to X5Y5.

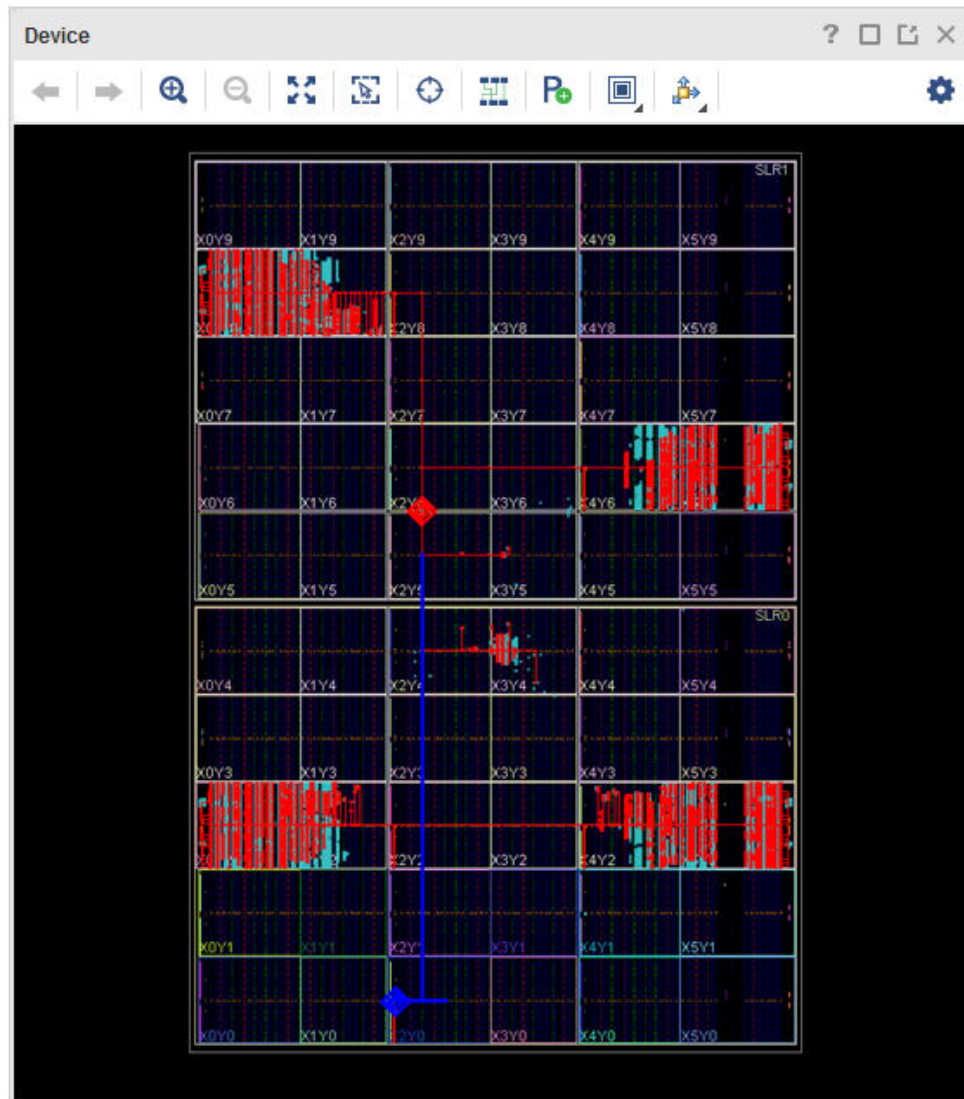
Figure 49: UltraScale Device Clock Routing from Driver to Loads



X15389-120619

In the following figure, a routed device view shows an example of a global clock that spans most of the device. The clock buffer driving the network is marked in blue in clock region X2Y0 and drives onto the horizontal routing in that clock region. The net then transitions from the horizontal routing onto the vertical routing in clock region X2Y0 reaching the clock root in clock region X2Y5. All clock routing is marked in blue. The clock root is marked in red in the clock region X2Y5. From the clock root in X2Y5, the net transitions onto the vertical distribution and then the horizontal distribution to the clock leaf pins. The distribution layer and the leaf clock routing resources in the CLB columns are marked in red.

Figure 50: Routed Device View of a Routed Clock Network



Clock Tree Placement and Routing

During the following phases, the Vivado placer determines the placement of MMCM/PLLs, global clock buffers, and the clock root while honoring the physical XDC constraints:

1. I/O and clock placement

The placer places I/O buffers and MMCM/PLLs based on connectivity rules and user constraints. The placer assigns clock buffers to clock regions but not to individual sites unless constrained using the LOC property. Only the clock buffers that only drive non-clock loads can move to a different clock region later in the flow based on the placement of their driver and loads.

Any placer error at this phase is due to conflicting connectivity rules, user constraints, or both. The log file shows extensive information about the possible root cause of the error, which you must review in detail to make the appropriate design or constraint change.

2. SLR partitioning (SSI technology devices only) and global placement

The placer performs the initial clock tree implementation based on early driver and load placements. Each clock net is associated with a clock window. The excessive overlap of clock windows can lead to placer errors due to anticipated clock routing contention.

When a clock partitioning error occurs, the log file shows the last clock budgeting solution for each clock net as well as the number of unique clock nets present in each clock region. Review the log file in detail to determine which clocks to remove from the overutilized clock regions. You can remove clocks using the following methods:

- Reduce the number of clocks in the design by combining identical synchronous clocks, removing unnecessary MMCM feedback clocks, or consolidating lower fanout clocks with high fanout clocks.
- Move clock primitives to different clock regions, especially those without connectivity-based placement rules.
- Add floorplanning constraints on clock loads to keep clocks with smaller fanout closer to their driver or away from the highly utilized clock regions.

The placer refines the clock tree implementation several times to help improve timing QoR. For example, during the later placement optimization phases, the placer analyzes each challenging clock to determine a better clock root location.

3. Clock tree pre-routing

The placer guides the subsequent implementation steps and provides accurate delay estimates for post-place timing analysis.

After placement, the Vivado tools can modify the clock tree implementation as follows:

- The Vivado physical optimizer can replicate and move cells to clock regions without associated clocks.
- The Vivado router can make adjustments to improve timing QoR and legalize the clock routing.

The following table summarizes the placement rules for the main clock topologies and how constraints affect these rules.

Table 2: Topologies with and without Placement Rules

Constrained Source	Unconstrained Destination	Behavior
GCIO	BUFGCE, BUFGCTRL, BUFGCE_DIV, PLL/MMCM	Automatically placed in same clock region.
PLL/MMCM	BUFGCE, BUFGCTRL, BUFGCE_DIV	Automatically placed in same clock region.
GT*_CHANNEL	BUFG_GT	Automatically placed in same clock region.

Table 2: Topologies with and without Placement Rules (cont'd)

Constrained Source	Unconstrained Destination	Behavior
BUFGCTRL	BUFGCTRL	Automatically placed in same clock region. Note: You can override placement within same clock region using the CLOCK_REGION constraint.
BUFG*	BUFG*	Unpredictable placement of unconstrained destination BUFG. Recommend constraining destination BUFG* using the CLOCK_REGION constraint. Note: This excludes BUFGCTRL > BUFGCTRL.
BUFG*	MMCM/PLL	Unpredictable placement of unconstrained destination MMCM/PLL. Recommend constraining MMCM/PLL using a LOC constraint. Recommend CLOCK_DEDICATED_ROUTE constraint when the route spans adjacent or multiple clock regions.

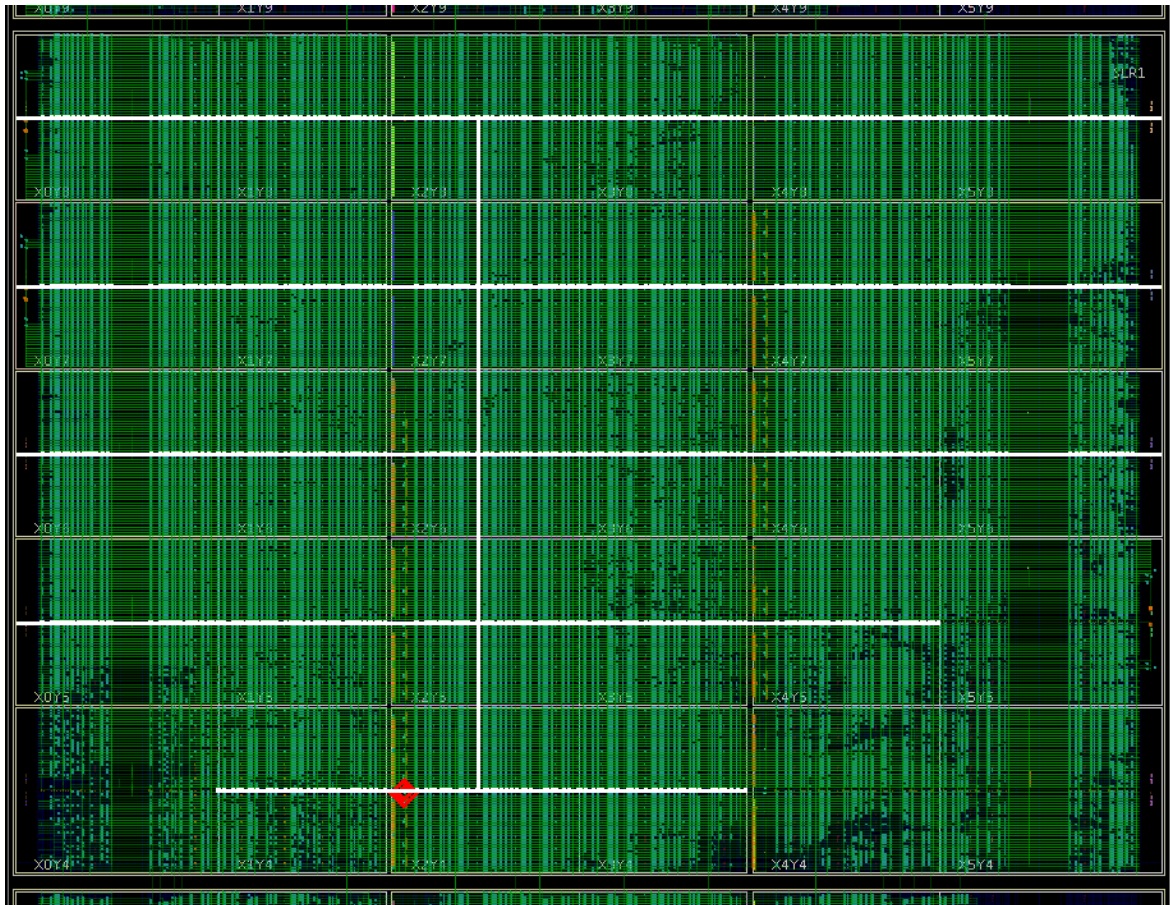
Clocking Capability

Clock planning must be based on the total number of high fanout clocks and low fanout clocks in the target device.

High Fanout Clocks

A high fanout clock spans almost an entire SLR of an SSI technology device or almost all clock regions of a monolithic device. The following figure shows a high fanout clock that spans almost an entire SLR with the BUFGCE driver shown in red.

Figure 51: High Fanout Clock Spanning an SLR



Note: Using more than 24 clocks in a design might cause issues that require special design considerations or other up-front planning.

★ **IMPORTANT!** In ZHOLD and BUF_IN compensation modes, the MMCM feedback clock path matches the CLKOUT0 clock path in terms of routing track, clock root location, and distribution tracks. Therefore, the feedback clock can be considered a high fanout clock when the clock buffer and clock root are far apart.

Related Information

[I/O Timing with MMCM ZHOLD/BUF_IN Compensation](#)

Low Fanout Clocks

In most cases, a low fanout clock is a clock net that is connected to less than 5,000 clock pins, which are placed in 3 or fewer horizontally adjacent clock regions. The clock routing, clock root, and clock distribution are all contained within the localized area.

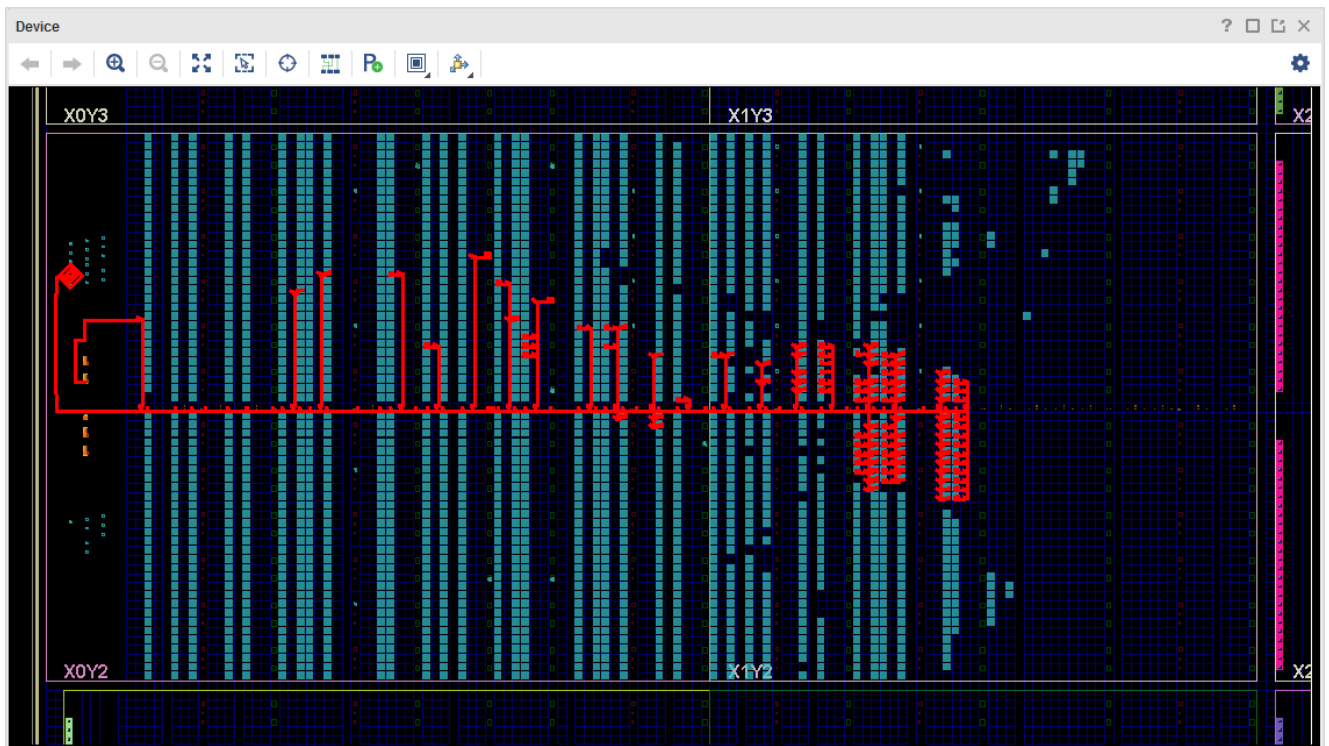
In some cases, the placer is expected to identify a low fanout clock but fails. This can be caused by design size, device size, or physical XDC constraints, such as a LOC constraint or Pblock, which prevent the placer from placing the loads in a local area. To address this issue, you might need to guide the tool by manually creating a Pblock or modifying the existing physical constraints.

Clocks driven by BUFG_GTs are an example of a low fanout clock. The Vivado placer automatically identifies these clock nets and contains the loads to the clock regions adjacent to the GT interface. The following figure shows a low fanout clock contained in two clock regions with the BUFG_GT driver shown in red.



TIP: To contain a low fanout clock to a single clock region, you can use the `CLOCK_LOW_FANOUT` XDC constraint.

Figure 52: Low Fanout Clock Contained in Two Clock Regions



Related Information

[Using the CLOCK_LOW_FANOUT Constraint](#)

Balanced Utilization of High and Low Fanout Clocks

UltraScale devices support more clocks than previous Xilinx device families. This enables a wide range of clocking utilization scenarios, such as the following:

- 24 clocks or less

Unless conflicting user constraints exist, all clocks can be treated as high fanout clocks without risking placement or routing contention.

- Almost 300 clocks

For a design that targets a device with 6 clock region rows and includes only low fanout clocks with each clock included in 3 clock regions at most, the following clocks are required: 6 rows x 2 clock windows per row x 24 clocks per region = 288 clocks.

Low fanout clock windows do not have a fixed size but are usually between 1 and 3 clock regions. High fanout clocks rarely span the entire device or an entire SLR.

The following method shows how to balance high fanout clocks and low fanout clocks, assuming that a few low fanout clocks come from I/O interfaces and most from GT interfaces. You can apply the same method for each SSI technology device SLR.

- High fanout clocks
 - Up to 12 for monolithic devices
 - Up to 24 for SSI technology devices (assuming some high fanout clocks are only present in 1 SLR)
- Low fanout clocks
 - Up to 12 plus 8 per GT utilized Quad
 - Alternatively, up to 12 plus 6 per GT interface (group of GT channels that share the RXUSRCLK and TXUSRCLK)

Clock Constraints

Physical XDC constraints drive the implementation of clock trees and control the use of high fanout clocking resources. Because UltraScale device clocking is more flexible than clocking with previous architectures and includes additional architectural constraints, it is important to understand how to properly constrain your clocks for implementation.

Using LOC Constraints for IO/MMCM/PLL/GT

To constrain clocks, you can assign placement constraints as follows:

- On a clock input at the I/O port

Assigning a PACKAGE_PIN constraint for a clock on a GCIO or assigning a LOC to an IOB affects the clock network. The MMCM/PLL and clock buffers directly connected to the input port must be placed in the same clock region.

- On an MMCM or PLL

The clock buffers directly connected to the MMCM or PLL outputs and the input clock ports connected to the MMCM or PLL inputs are automatically placed in the same clock region. If an input clock port and an MMCM or PLL are directly connected and constrained to different clock regions, you must manually insert a clock buffer and set a `CLOCK_DEDICATED_ROUTE` constraint on the net connected to the MMCM or PLL.

- On a `GT*_CHANNEL` or `IBUFDS_GT*` cell

The `BUFG_GT`s driven by the cell are placed in the same clock region.



CAUTION! Xilinx does not recommend using LOC constraints on the clock buffer cells. This method forces the clock onto a specific track ID, which can result in placement that cannot be legally routed. Only use LOC constraints to place high fanout clock buffers in UltraScale devices when you understand the entire clock tree of the design and when placement is consistent in the design. Even after taking these precautions, collisions might occur during implementation due to design or constraint changes.

Using the `CLOCK_REGION` Property on Clock Buffers

You can use the `CLOCK_REGION` constraint to assign a clock buffer to a clock region without specifying a site. This gives the placer more flexibility when optimizing all the clock trees and when determining the appropriate buffer sites to successfully route all clocks.

You can also use a `CLOCK_REGION` constraint to provide guidance on the placement of cascaded clock buffers or clock buffers driven by non-clocking primitives, such as fabric logic.

In the following example, the XDC constraint assigns the `clkgen/clkout2_buf` clock buffer to the `CLOCK_REGION X2Y2`.

```
set_property CLOCK_REGION X2Y2 [get_cells clkgen/clkout2_buf]
```

Note: In most cases, the clock buffers are directly driven by input clock ports, MMCMs, PLLs, or `GT*_CHANNEL`s that are already constrained to a clock region. If this is the case, the clock buffers are automatically placed in the same clock region, and you do not need to use the `CLOCK_REGION` constraint.

Using a Pblock to Restrict Clock Buffer Placement

When a clock buffer does not need to be placed in a specific clock region, you can use a Pblock to specify a range of clock regions. For example, use a Pblock when a `BUFGCTRL` is needed to multiplex two clocks that are located in different areas. You can assign the `BUFGCTRL` to a Pblock that includes the clock regions between the two clock drivers and let the placer identify a valid placement.

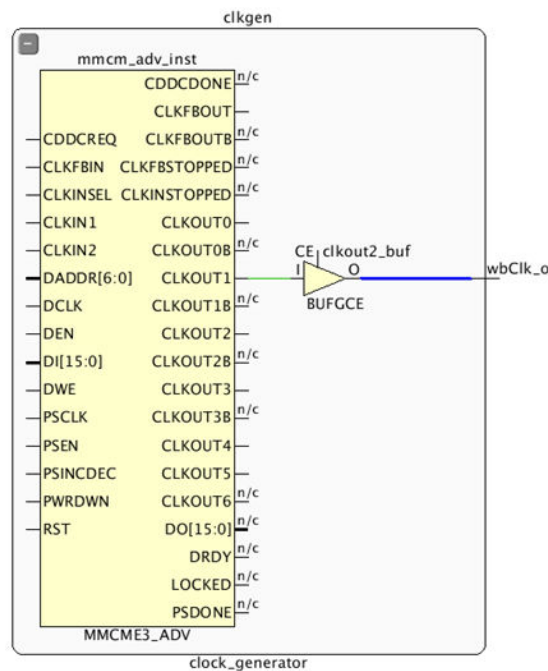
Note: Xilinx does not recommend using a Pblock for a single clock region.

Using the USER_CLOCK_ROOT Property on a Clock Net

You can use the USER_CLOCK_ROOT property to force the clock root location of a clock driven by a clock buffer. Specifying the USER_CLOCK_ROOT property influences the design placement, because it impacts both insertion delay and skew by modifying the clock routing. The USER_CLOCK_ROOT value corresponds to a clock region, and you must set the property on the net segment directly driven by the high fanout clock buffer. Following is an example:

```
set_property USER_CLOCK_ROOT X2Y3 [get_nets clkgen/wbClk_o]
```

Figure 53: USER_CLOCK_ROOT Applied on the Net Segment Driven by the Clock Buffer



After placement, you can use the CLOCK_ROOT property to query the actual clock root as shown in the following example. The CLOCK_ROOT reports the assigned root whether it was user assigned or automatically assigned by the Vivado tools.

```
get_property CLOCK_ROOT [get_nets clkgen/wbClk_o]
=> X2Y3
```

Another way to review the clock root assignments of your implemented design is to use the report_clock_utilization Tcl command. For example:

```
report_clock_utilization -clock_roots_only
```

The following figure shows this report.

Figure 54: report_clock_utilization Clock Root Assignments

Index	Clock Net	Root Clock Region
1	clkgen/clkfbout_buf	X4Y1
2	clkgen/cpuClk_o	X4Y1
3	clkgen/fftClk_o	X3Y2
4	clkgen/phyClk0_o	X3Y3
5	clkgen/phyClk1_o	X3Y2
6	clkgen/usbClk_o	X3Y3

Using the CLOCK_DELAY_GROUP Constraint on Several Clock Nets

You can use the CLOCK_DELAY_GROUP constraint to match the insertion delay of multiple, related clock networks driven by different clock buffers. This constraint is commonly used to minimize skew on synchronous CDC timing paths between clocks originating from the same MMCM, PLL, or GT source. You must set the CLOCK_DELAY_GROUP constraint on the net segment directly connected to the clock buffer. The following example shows the clk1_net and clk2_net clock nets, which are directly driven by the clock buffers:

```
set_property CLOCK_DELAY_GROUP grp12 [get_nets {clk1_net clk2_net}]
```

Related Information

[Synchronous CDC](#)

Using the CLOCK_DEDICATED_ROUTE Constraint

The CLOCK_DEDICATED_ROUTE constraint is typically used when driving from a clock buffer in one clock region to an MMCM or PLL in another clock region. By default, the CLOCK_DEDICATED_ROUTE constraint is set to TRUE, and the buffer/MMCM or PLL pair must be placed in the same clock region.

Note: Using the 7 series CLOCK_DEDICATED_ROUTE value of BACKBONE on an UltraScale device results in the same behavior as SAME_CMT_COLUMN.

The following table summarizes the different CLOCK_DEDICATED_ROUTE constraint values, use, and behavior.

Table 3: UltraScale Device CLOCK_DEDICATED_ROUTE Constraint Summary

Value	Use	Behavior
TRUE	Default value on clock nets	Global clock buffer and MMCM/PLLs must be placed in the same clock region. This value ensures the net is routed using only global clock resources.

Table 3: UltraScale Device CLOCK_DEDICATED_ROUTE Constraint Summary (cont'd)

Value	Use	Behavior
SAME_CMT_COLUMN (BACKBONE)	Net driven by a global clock buffer Example: <pre>set_property CLOCK_DEDICATED_ROUTE SAME_CMT_COLUMN \ [get_nets -of [get_pins BUFGCE_inst/O]]</pre>	MMCM/PLLs must be placed in a clock region in the same vertical column. This value ensures the net is routed using only global clock resources. For optimal results, Xilinx recommends using a LOC constraint on the MMCM/PLL to control placement of the MMCM/PLL within in the same vertical column.
ANY_CMT_COLUMN	Net driven by a global clock buffer Examples: <pre>set_property CLOCK_DEDICATED_ROUTE ANY_CMT_COLUMN \ [get_nets -of [get_pins BUFGCE_inst/O]] set_property CLOCK_DEDICATED_ROUTE ANY_CMT_COLUMN \ [get_nets -of [get_pins BUFGCE_DIV_inst/O]] set_property CLOCK_DEDICATED_ROUTE ANY_CMT_COLUMN \ [get_nets -of [get_pins BUFGCTRL_inst/O]]</pre>	MMCM/PLLs can be placed in any clock region with available resources. This value ensures the net is routed using only global clock resources. For optimal results, Xilinx recommends using a LOC constraint on the MMCM/PLL to control placement of the MMCM/PLL within the device.
FALSE	Clock net not driven by a global clock buffer but part of the clock network (for example, nets driven by the output of an IBUF or nets directly connected to output clock pins of an MMCM) Examples: <pre>set_property CLOCK_DEDICATED_ROUTE FALSE \ [get_nets -of [get_pins MMCME4_ADV_inst/CLKOUT0]] set_property CLOCK_DEDICATED_ROUTE FALSE \ [get_nets -of [get_pins IBUF_inst/O]]</pre>	Net is routed using fabric and global clock resources. This can adversely affect the timing and performance of the clock network. <hr/> <p>IMPORTANT! For UltraScale devices, the FALSE value must only be used when a clock normally routed with global clock resources needs to be routed with fabric resources for special design reasons.</p> <hr/>

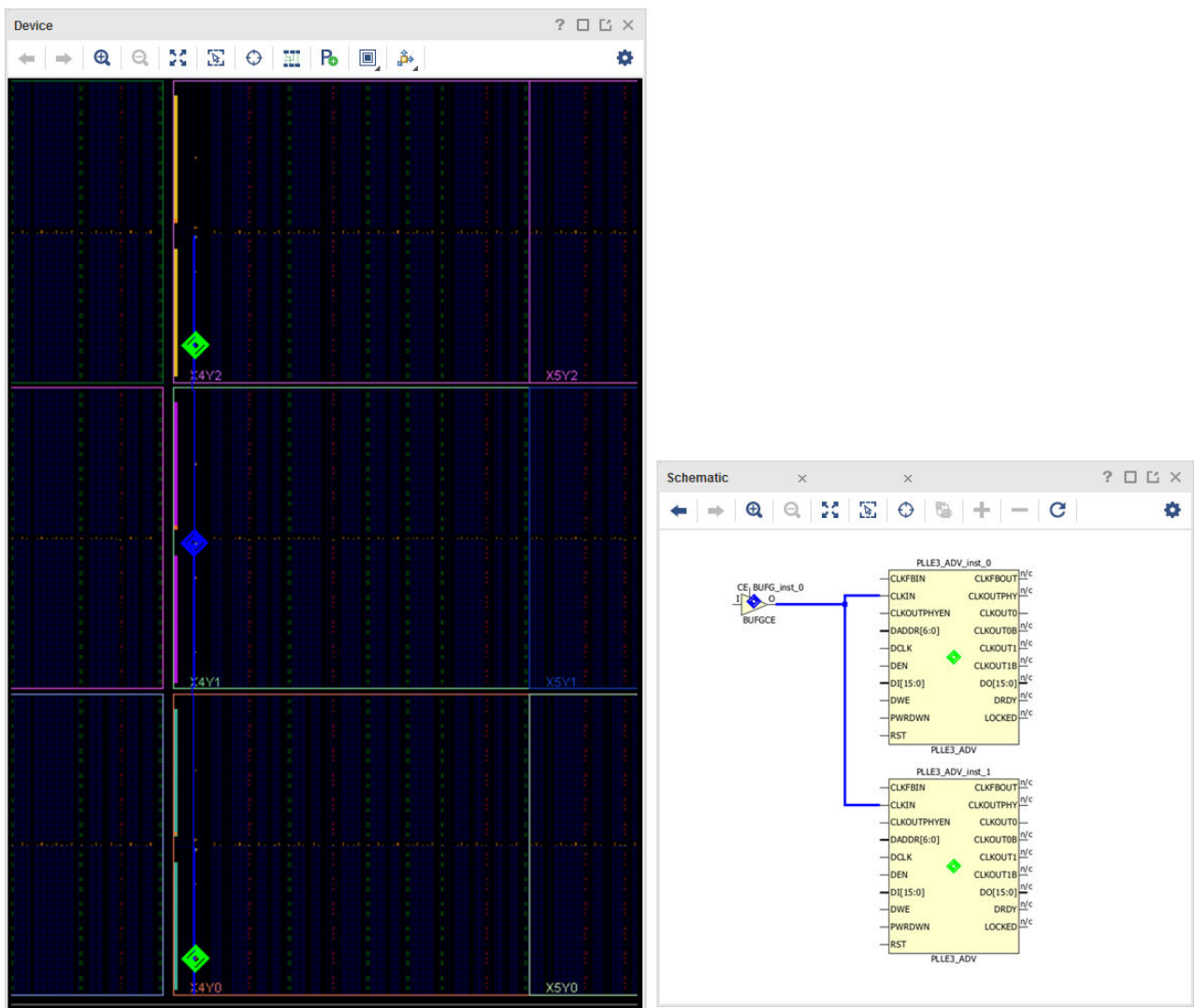
Note: When working with UltraScale devices, do not apply the CLOCK_DEDICATED_ROUTE property to the net driven directly by a port. Instead, apply the CLOCK_DEDICATED_ROUTE property to the output of the IBUF.

Constraint Example for Vertically Adjacent Clock Regions

When driving from a clock buffer in one clock region to a MMCM or PLL in a vertically adjacent clock region, you must set the `CLOCK_DEDICATED_ROUTE` to `BACKBONE` for 7 series devices or to `SAME_CMT_COLUMN` for UltraScale devices. This prevents implementation errors and ensures that the clock is routed with global clock resources only. The following example and figure show a clock buffer driving two PLLs in vertically adjacent clock regions.

```
set_property CLOCK_DEDICATED_ROUTE SAME_CMT_COLUMN [get_nets -of [get_pins BUFG_inst_0/O]]
set_property LOC PLLE3_ADV_X0Y0 [get_cells PLLE3_ADV_inst_0]
set_property LOC PLLE3_ADV_X0Y4 [get_cells PLLE3_ADV_inst_1]
```

Figure 55: `CLOCK_DEDICATED_ROUTE` Constraint Set to `SAME_CMT_COLUMN`

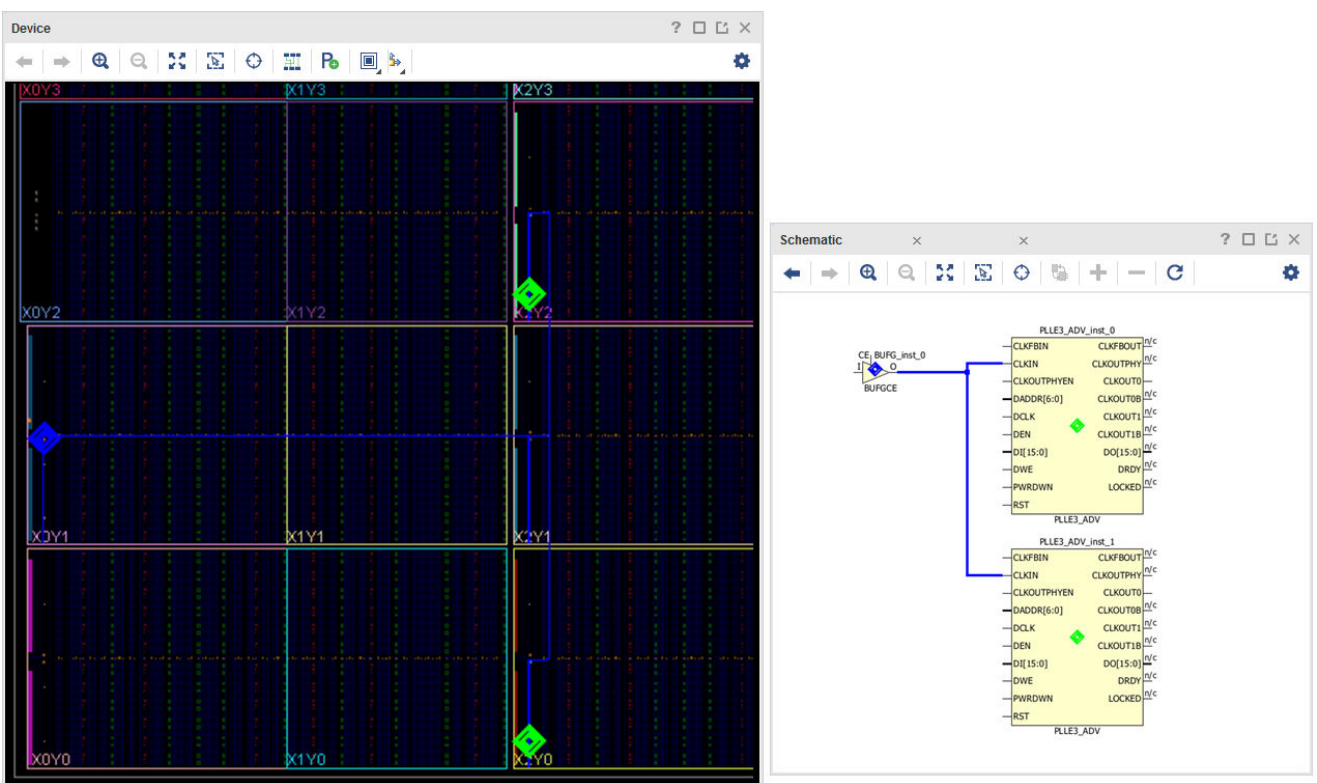


Constraint Example for Non-Vertically Adjacent Clock Regions

When driving from a clock buffer to other clock regions that are not vertically adjacent, you must set the `CLOCK_DEDICATED_ROUTE` to `FALSE` for 7 series devices or to `ANY_CMT_COLUMN` for UltraScale devices. This prevents implementation errors and ensures that the clock is routed with global clock resources only. The following example and figure show a `BUFGCE` driving two PLLs that are not located on the same clock region column as the input buffer.

```
set_property CLOCK_DEDICATED_ROUTE ANY_CMT_COLUMN [get_nets -of [get_pins BUFG_inst_0/O]]
set_property LOC PLLE3_ADV_X1Y0 [get_cells PLLE3_ADV_inst_0]
set_property LOC PLLE3_ADV_X1Y4 [get_cells PLLE3_ADV_inst_1]
```

Figure 56: `CLOCK_DEDICATED_ROUTE` Set to `ANY_CMT_COLUMN`



Using the `CLOCK_LOW_FANOUT` Constraint

You can use the `CLOCK_LOW_FANOUT` constraint to contain the loads of a clock buffer in a single clock region. You can set the `CLOCK_LOW_FANOUT` constraint on a clock net segment directly driven by a global clock buffer or on a list of flip-flops.

Note: The `CLOCK_LOW_FANOUT` constraint takes lower precedence when used with other clocking constraints. If `CLOCK_LOW_FANOUT` is in conflict with other clock constraints, such as `USER_CLOCK_ROOT`, `CLOCK_DELAY_GROUP`, or `CLOCK_DEDICATED_ROUTE`, `CLOCK_LOW_FANOUT` is not obeyed.

Constraint Example for Flip-Flops

Setting the `CLOCK_LOW_FANOUT` constraint on a list of flip-flops driven by a global clock buffer causes `opt_design` to create a new parallel global clock buffer to isolate the flip-flops. During `place_design`, the isolated flip-flops that are driven by the newly created parallel global clock buffer are contained to a single clock region.

The following example shows the `CLOCK_LOW_FANOUT` constraint applied to a list of flip-flops that are used as part of a clock gating synchronization circuit to control the clock enable of a global clock buffer.

```
set_property CLOCK_LOW_FANOUT TRUE [get_cells safeClockStartup_reg[*]]
```

In the design, an always-on clock network initially drives more than 2000 loads, including the flip-flops that are part of the clock gating synchronization circuit used to clock gate other logic. The following schematics show the clock gating synchronization circuit and additional logic connected to the always-on clock network before and after `opt_design` creates a new parallel global clock buffer to isolate the clock gating synchronization circuit.

Figure 57: Schematic Before `opt_design` Transform with `CLOCK_LOW_FANOUT` Applied to Flip-Flops

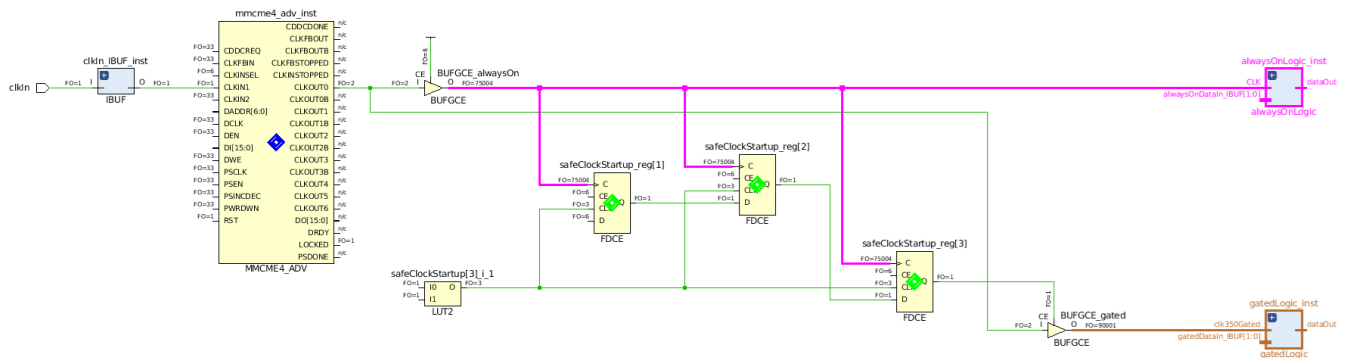
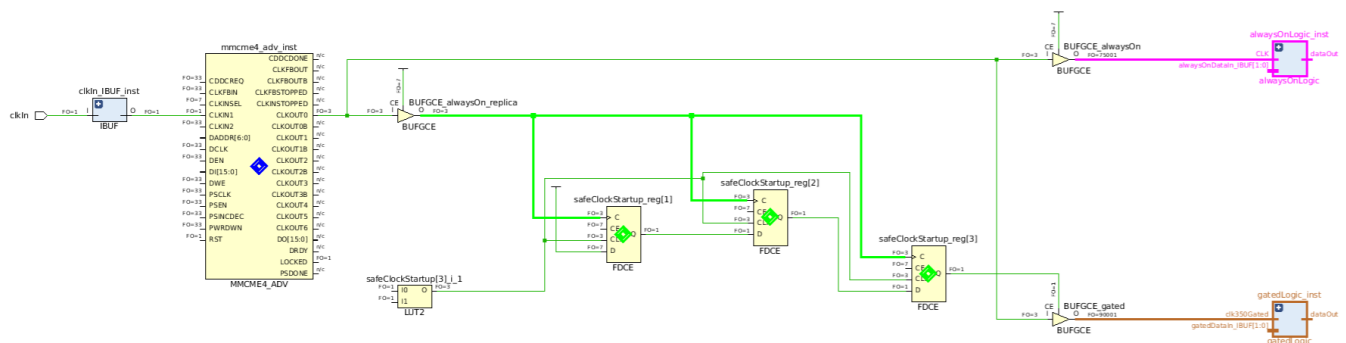
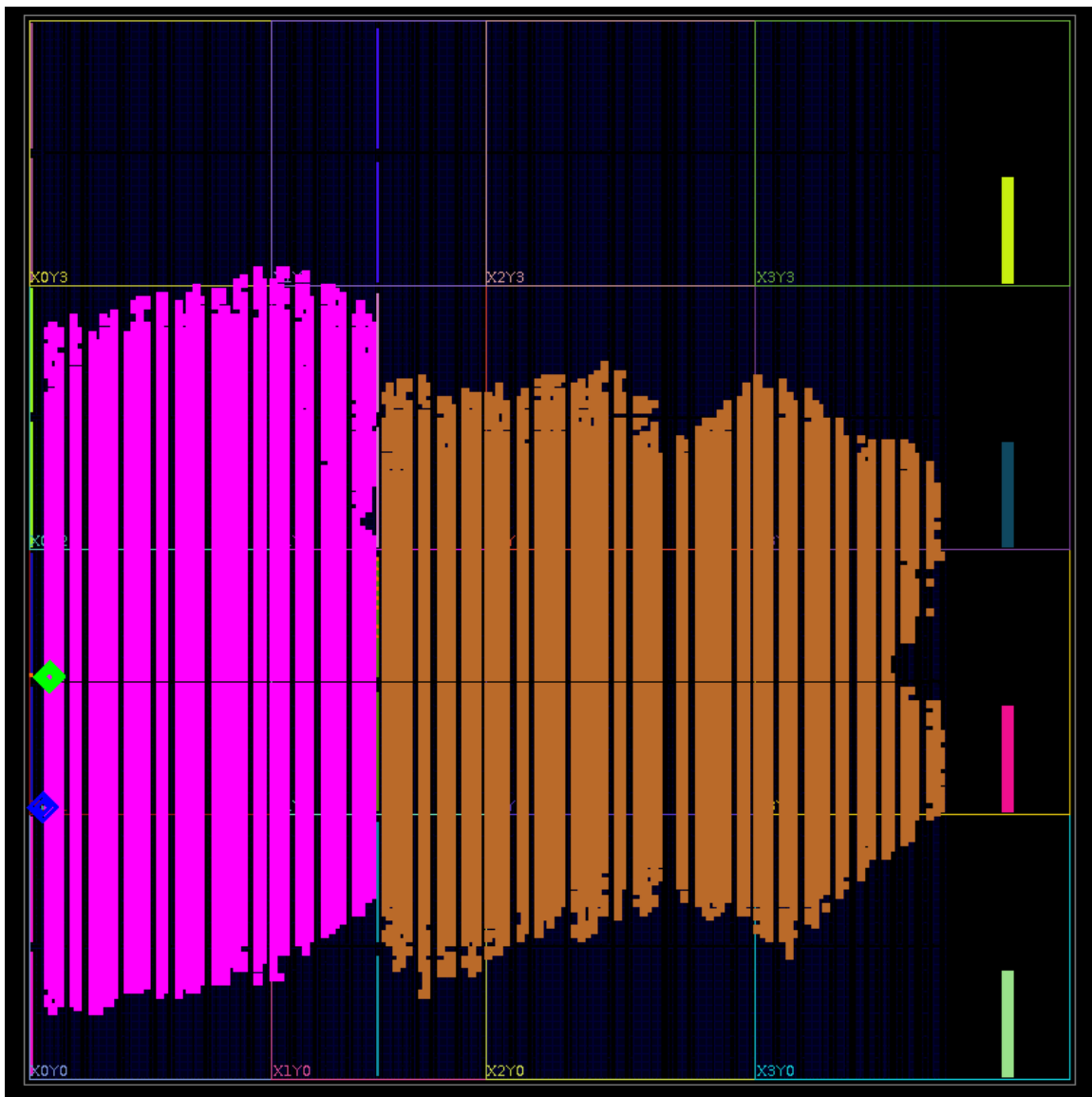


Figure 58: Schematic After `opt_design` Transform with `CLOCK_LOW_FANOUT` Applied to Flip-Flops



The Device window of the fully implemented design shows the clock gating synchronization circuit with green markers along with the always-on logic and clock-gated logic. The clock gating synchronization circuit is placed in the same CLOCK_REGION as the MMCM, close to the global clock buffers.

Figure 59: Fully Implemented Design with Placement of Clock Gating Synchronization Circuit



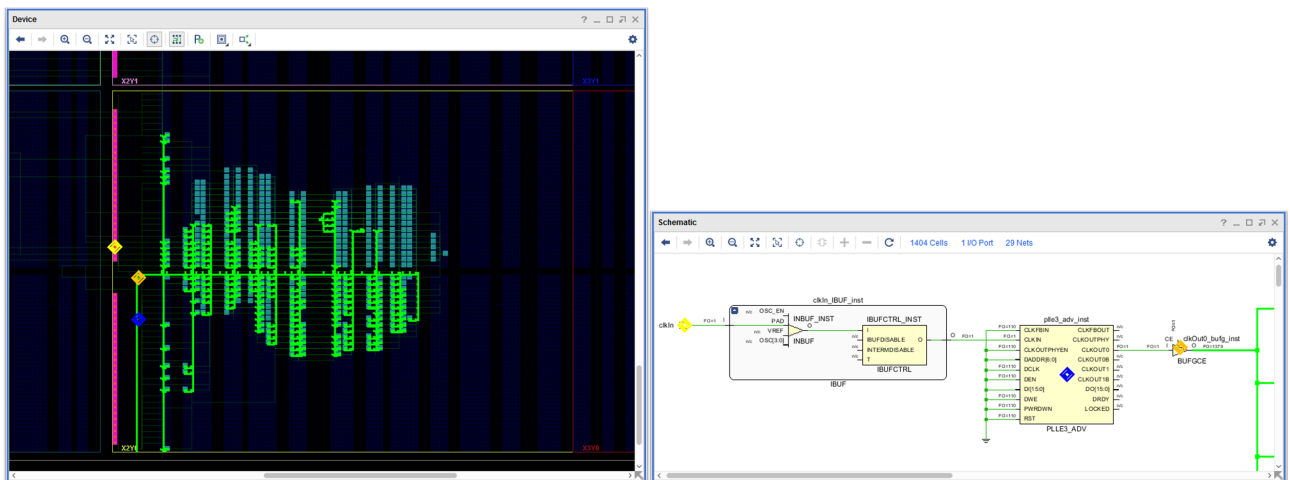
Constraint Example for Clock Nets

If you set the `CLOCK_LOW_FANOUT` property on a clock net segment directly driven by a global clock buffer and the fanout of the global clock buffer is less than 2000 loads, the placement of the loads is contained to a single clock region.

The following example shows the `CLOCK_LOW_FANOUT` constraint applied to the clock net segment directly driven by a global clock buffer. The clock network drives less than 2000 loads and is contained to a single clock region. The input clock port, `clkIn` has a `PACKAGE_PIN` assignment to a GCI0 located in the `CLOCK_REGION X2Y0` and drives a `PLLE3_ADV`. The `PLLE3_ADV` drives a global clock buffer that subsequently drives the clock network with 1379 loads. The loads of the global clock buffer are all placed in the `CLOCK_REGION X2Y0`.

```
# PACKAGE_PIN AF9 - IOBank 64 - CLOCK_REGION X2Y0
set_property PACKAGE_PIN AF9 [get_ports clkIn]
set_property IOSTANDARD LVCMOS18 [get_ports clkIn]
set_property CLOCK_LOW_FANOUT TRUE [get_nets -of [get_pins
clkOut0_bufg_inst/O]]
```

Figure 60: `CLOCK_LOW_FANOUT` Example in the Device Window and Schematic Window



Clocking Topology Recommendations

Xilinx recommends using simple clock tree topologies with the minimum number of clock buffers required for the design. Using extra clock buffers requires more routing tracks, which can lead to placement errors or routing conflicts in clock regions where the clock routing requirement is high and is close to the maximum capacity.

Following are clocking topology recommendations for `BUFGCE`/`BUFGCTRL`/`BUFGCE_DIV` connectivity.

Parallel Clock Buffers

Use parallel clock buffers to achieve the following:

- Ensure predictable placement across implementation runs.

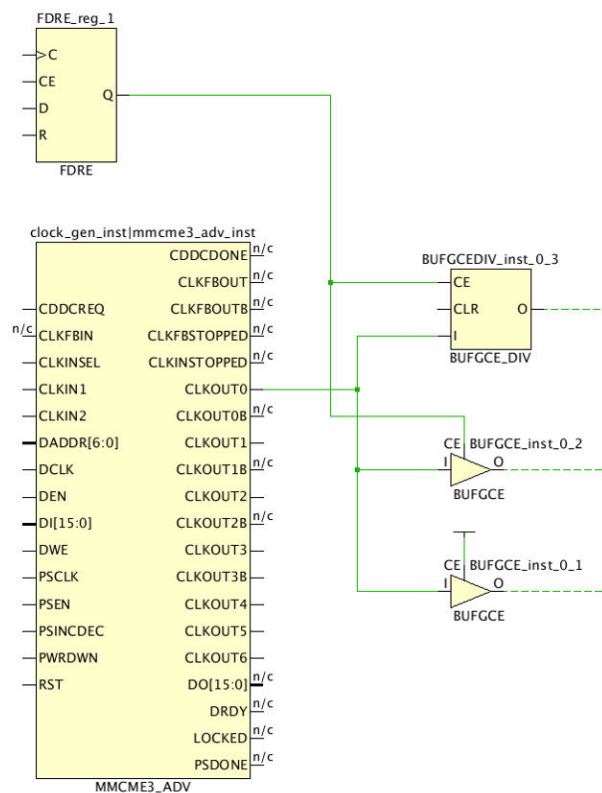
When the parallel clock buffers are directly driven by the same input clock port, MMCM, PLL, or GT*_CHANNEL, the buffers are always placed in the same clock region as their driver regardless of the netlist changes or logic placement variation.

- Match the insertion delays between parallel branches of the clock tree.

Xilinx recommends parallel buffers over cascaded clock buffers, especially when there are synchronous paths between the branches. When using cascaded buffers, the clock insertion delay is *not* matched between the branches of the clock trees even when using the CLOCK_DELAY_GROUP or USER_CLOCK_ROOT constraints. This can result in high clock skew, which makes timing closure challenging if not impossible.

The following figure shows three parallel BUFGE buffers driven by the MMCM CLKOUT0 port.

Figure 61: Parallel BUFGE on MMCM Output



Cascaded Clock Buffers

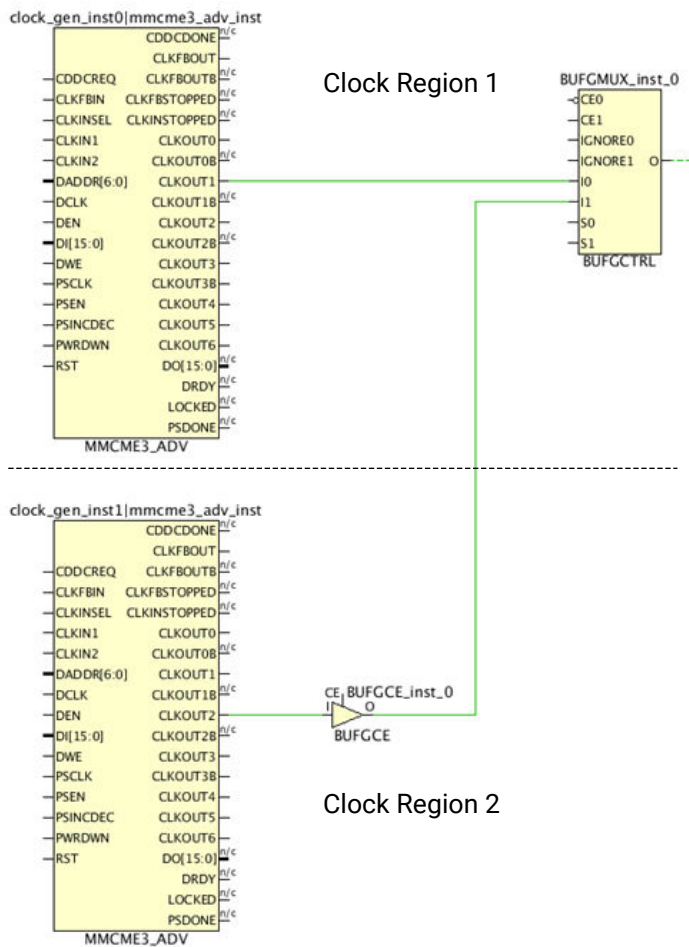
In general, Xilinx does not recommend using cascaded buffers to artificially increase the delay and reduce the skew between unrelated clock trees branches. Unlike connections between BUFGECTRLs, other clock buffer connections do not have a dedicated path in the architecture. Therefore, the relative placement of clock buffers is not predictable, and all placement rules take precedence over placing unconstrained cascaded buffers.

However, you can use cascaded clock buffers to achieve the following:

- Route the clock to another clock buffer located in a different clock region.

This method is typical when using a clock multiplexer for clocks generated by MMCMs located in different clock regions. Although one of the MMCMs can directly drive the BUFGCTRL (BUFGMUX), the other MMCM requires an intermediate clock buffer to route the clock signal to the other region. The following figure shows an example.

Figure 62: Routing the Clock to Another Clock Region



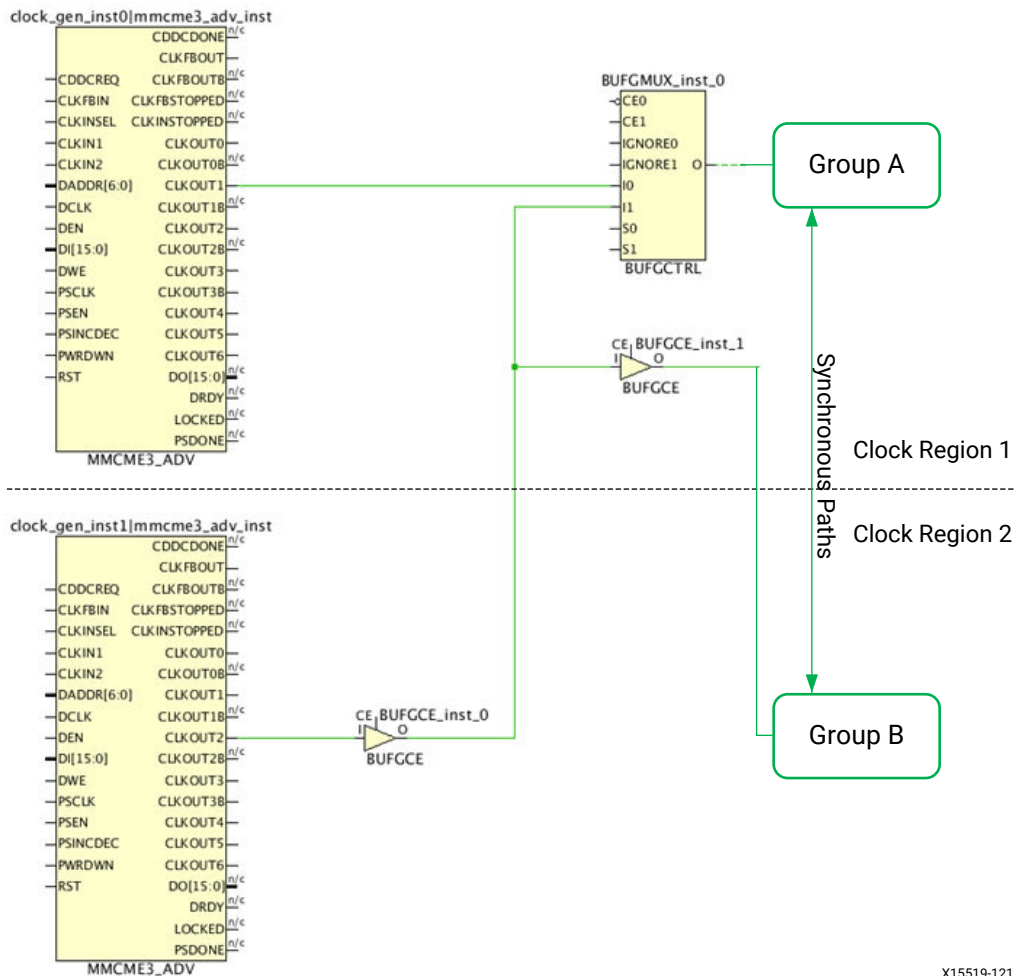
X15518-121919

- Balance the number of clock buffer levels across the clock tree branches when there is a synchronous path between those branches.

For example, consider an MMCM clock called clk0 that drives both group A (sequential cells driven via a BUFGCTRL located in a different clock region) and group B (sequential cells). To better match the delay between the branches, insert a BUFGCE for group B and place it in the same clock region as the BUFGCTRL. This ensures that the synchronous paths between group A and group B have a controlled amount of skew. The following figure shows an example.

Note: The Vivado logic optimization command `opt_design` is not aware of the timing relationship between timing clocks and clock network branches. As a result, `opt_design` removes as many cascaded or redundant clock buffers as possible. In this example, `opt_design` removes `BUFGCE_inst_1` unless you set a `DONT_TOUCH = "TRUE"` property on it. If there are only asynchronous paths between the clock tree branches, the branches do not need to be balanced as long as there is proper synchronization circuitry on the receiving clock domain.

Figure 63: Balancing Clock Trees for Synchronous Paths Between Clock Regions



X15519-121919

- Build clock multiplexers as described in [Clock Multiplexing](#).

To reduce the variation of insertion delays and skew, Xilinx recommends the following when using cascaded clock buffers:

- Keep the cascaded buffers in the same or adjacent clock regions.
- When clock tree branches are balanced, assign all the clock buffers of the same level to the same clock region.

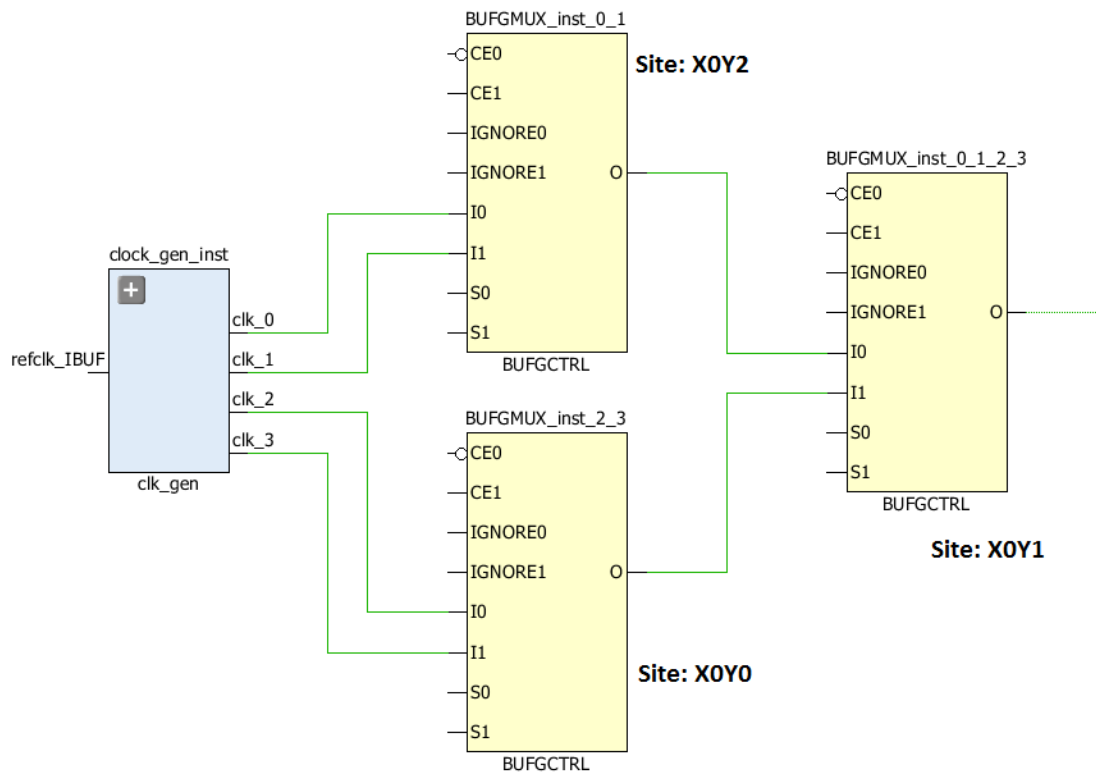
Note: If absolutely required, Xilinx recommends using two cascaded BUFGCTRLs instead of cascaded BUFGCEs. Using dedicated routing, you can cascade two adjacent BUFGCTRLs with minimum delay when both BUFGCTRLs are placed inside the same clock region.

Clock Multiplexing

You can build a clock multiplexer using a combination of parallel and cascaded BUFGCTRLs. The placer finds the optimal placement based on the clock buffer site availability. If possible, the placer places BUFGCTRLs in adjacent sites to take advantage of the dedicated cascade paths. If that is not possible, the placer will attempt to place the BUFGCTRLs from the same level in the adjacent clock regions.

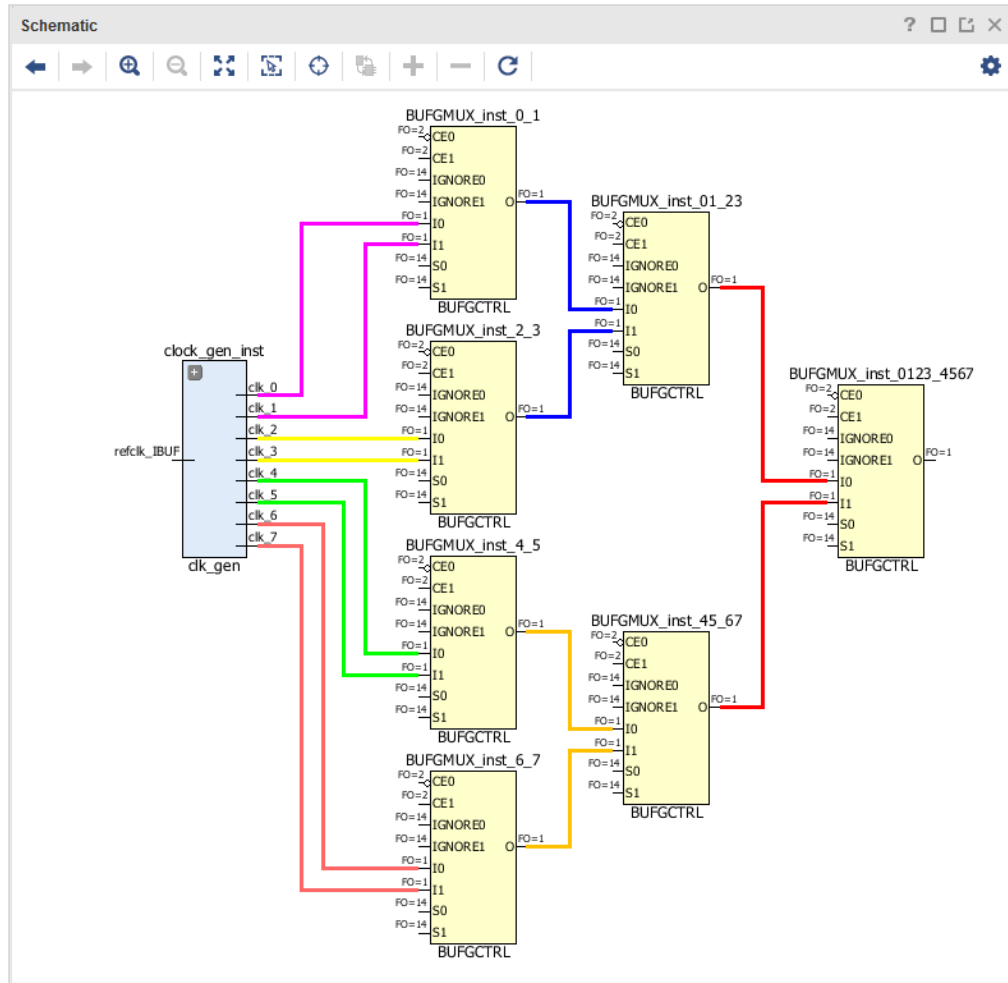
The following figure shows a 4:1 MUX with balanced cascading. The first level of BUFGCTRL buffers are both placed in the directly adjacent sites (X0Y2, X0Y0) of the last BUFGCTRL (X0Y1). This configuration ensures a comparable insertion delay for all the clocks reaching the last BUFGCTRL. You can use an equivalent structure for a 3:1 MUX.

Figure 64: 4:1 MUX Using Parallel BUFGCTRL



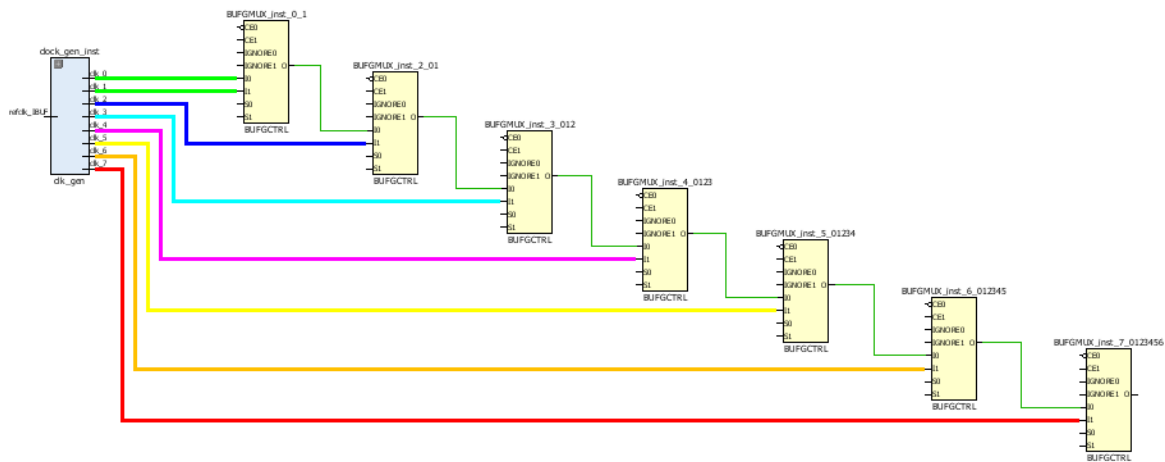
When creating a 5:1 or larger clock MUX structure, it is common to create a symmetrical clock structure as shown in the following figure. However, this is a suboptimal solution, because each BUFGCTRL only has one cascade path to the two adjacent BUFGCTRLs, which does not provide minimal delay for all connections between the BUFGCTRLs.

Figure 65: Non-Recommended 8:1 Balanced Clock MUX Structure



To support larger clock multiplexers (from 5:1 to 8:1 MUX), Xilinx recommends using cascaded BUFCTRL buffers as shown in the following figure. This figure shows an optimal 8:1 MUX that uses 7 BUFCTRL buffers.

Figure 66: 8:1 MUX Using Cascaded BUFCTRL



Note: When using wide BUFCTRL-based clock multiplexers, the clock insertion delays cannot be balanced because some paths are longer than other paths in hardware. Therefore, this method is recommended for multiplexing asynchronous clocks only.

PLL/MMCM Feedback Path and Compensation Mode

PLLs do not support delay compensation and always operate in INTERNAL compensation mode, which means they do not need a feedback path. Similarly, MMCMs set to INTERNAL compensation mode do not need a feedback path. In both cases, the Vivado tools do not always automatically remove unnecessary feedback clock buffers. You must remove the clock buffers manually to reduce the amount of high fanout clock resource utilization. This is especially important for designs with high clocking usage where clock contention might occur.

When the MMCM compensation is set to ZHOLD or BUF_IN, the placer assigns the same clock root to the nets driven by the feedback buffer and by all buffers directly connected to the CLKOUT0 pin. This ensures that the insertion delays are matched so that the I/O ports and the sequential cells connected to CLKOUT0 are phase-aligned and hold time is met at the device interface. The Vivado tools consider all the loads of these nets to optimally define the clock root.

The Vivado tools do not automatically match the insertion delay with the other MMCM outputs. To match the insertion delay for the nets driven by other MMCM output buffers, use the following properties:

- **CLOCK_DELAY_GROUP**

Apply the same CLOCK_DELAY_GROUP property value to the nets directly driven by feedback clock buffer, the CLKOUT0 buffers, and the other MMCM output buffers as needed. This is the preferred method.

- **USER_CLOCK_ROOT**

If you need to force a specific clock root, use the same USER_CLOCK_ROOT property value on the nets driven by the feedback clock buffer, the CLKOUT0 buffers, and the other MMCM output buffers as needed.

BUFG_GT Divider

The BUFG_GT buffers can drive any loads in the fabric and include an optional divider you can use to divide the clock from the GT*_CHANNEL. This eliminates the need to use an extra MMCM or BUFG_DIV to divide the clock.

SelectIO Clocking

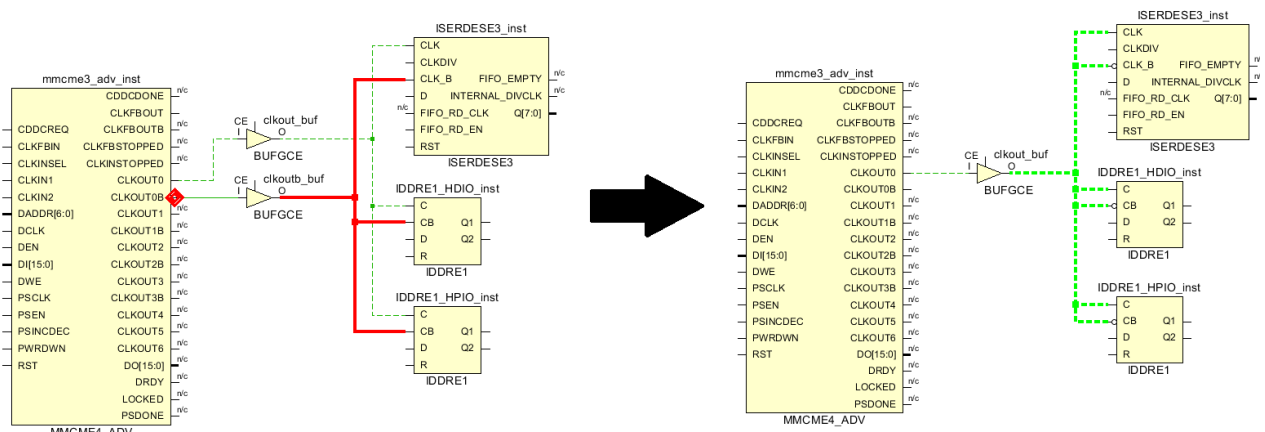
The UltraScale device SelectIO primitives have maximum skew requirements between clock pins. Using the optimal clocking topology for the SelectIO primitives prevents maximum skew violations, improves interface timing between the UltraScale device and the fabric logic, and uses fewer clocking resources.

ISERDESE3 and IDDRE1 Clocking

For ISERDESE3 and IDDRE1 clocking in UltraScale and UltraScale+ devices, maximum skew requirements exist between the clock and inverted clock pins. To meet the maximum skew requirements, Xilinx recommends using a single net for the clock and inverted clock pins when using the local inversion.

In the following figure, the left side shows a suboptimal configuration that uses the CLKOUT0B output of the MMCM. The right side of the figure shows the optimal configuration that uses the local inversion on the CLK_B and CB pins of the ISERDESE3 and IDDRE1. Using the optimal configuration guarantees that the maximum skew requirement is met while using fewer global clock resources.

Figure 67: Suboptimal to Optimal Clocking Topologies for ISERDESE3 and IDDRE1



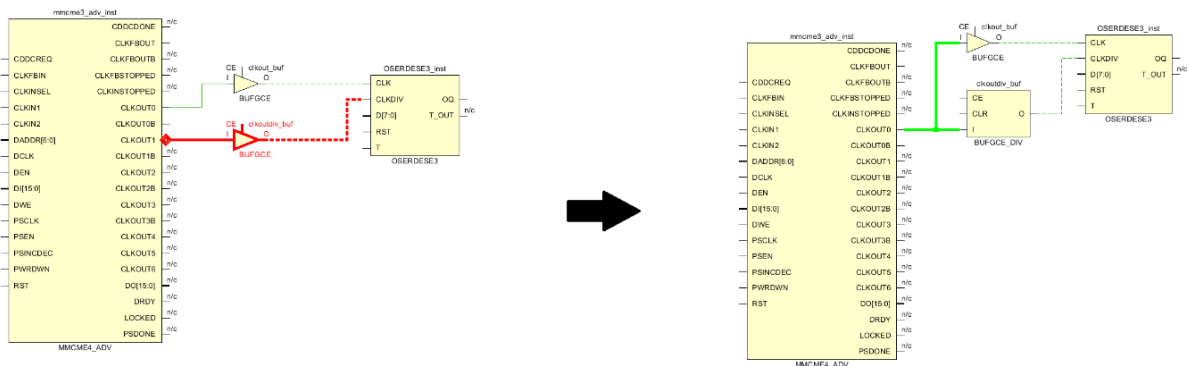
OSERDESE3 Clocking

For OSERDESE3 clocking in UltraScale and UltraScale+ devices, maximum skew requirements exist between the high-speed clock and divided clock pins. To meet the maximum skew requirements, Xilinx recommends using parallel global clock buffers where one of the global clock buffers is a BUFGCE_DIV. This removes the additional clock uncertainty between the two outputs of the MMCM.

In the following figure, the left side shows a suboptimal configuration that uses two separate outputs of the MMCM. The right side of the figure shows the optimal configuration that uses a single MMCM output and the BUFGCE_DIV cell, which provides the divided clock using the BUFGCE_DIVIDE property.

Note: The high-speed clock does not need to be driven by a BUFGCE. Alternatively, you can use BUFGCE_DIV with a BUFGCE_DIVIDE property setting of 1.

Figure 68: Suboptimal to Optimal Clocking Topologies for OSERDESE3



I/O Timing with MMCM ZHOLD/BUF_IN Compensation

ZHOLD compensation indicates that the MMCM is configured to provide a negative hold for all I/O registers of an entire I/O column. When a clock capable I/O (CCIO) drives a single MMCM that is configured in ZHOLD compensation mode, the placer will attempt to place the MMCM with the CCIO in the same clock region. In this case, the CCIO can drive the MMCM directly without going through a BUFG. This allows the ZHOLD compensation of the MMCM to remain in effect.

However, if a CCIO drives an MMCM configured in ZHOLD mode in addition to another MMCM, logic optimization will attempt to legalize the clock routing to the MMCMs by inserting a BUFG after the CCIO. Because the MMCM with ZHOLD compensation is no longer driven directly by a CCIO, the compensation is changed to BUF_IN. To avoid this, ensure that the CCIO drives the MMCM configured in ZHOLD mode directly and drives the additional MMCM through a BUFG. In addition, set the CLOCK_DEDICATED_ROUTE property for the net driven by the BUFG to ANY_CMT_COLUMN.

Because the clock insertion delay varies with the clock root locations and the clock root placement depends on placement of the loads, there might be variability between runs. This variability affects the timing inside the device as well as the I/O timing.

When dealing with high-frequency I/Os, you might want more control over the I/O timing and less variability between runs. One way to achieve this is to force the clock root placement. You can run the tool in automated mode and look at the clock root region. If the I/O timing is satisfactory, you can force the clock root placement on the buffer nets associated with I/O timing. To determine the placement of the clock roots, use the `report_clock_utilization [-clock_roots_only]` Tcl command.

In the following example, the I/O ports are located in the X0Y0 region. The Vivado placer determined the placement of the clock roots in X1Y2 based on the I/O placement as well as placement of other loads.

Figure 69: Clock Utilization Summary with Unconstrained Clock Root

Index	Clock Net	Root Clock Region	Clock Root Node
1	mmcn/inst/clk0	X1Y2	RCLK_BRAM_L_X30Y209/CLK_VDISTR_B0T0
2	mmcn/inst/clkfbout_buf_mmcn_zhold	X1Y2	RCLK_CLEL_R_L_X25Y209/CLK_VDISTR_B0T

The following summary shows the I/O timing when the clock root is unconstrained.

Figure 70: Timing Summary with Unconstrained Clock Root

Setup	Hold
Worst Negative Slack (WNS): -0.279 ns	Worst Hold Slack (WHS): 0.036 ns
Total Negative Slack (TNS): -5.394 ns	Total Hold Slack (THS): 0.000 ns
Number of Failing Endpoints: 47	Number of Failing Endpoints: 0

In the following example, the clock roots are moved next to the I/O registers in X0Y0, which reduces the clock insertion delays and timing pessimism and therefore, improves the I/O timing.

Figure 71: Clock Utilization Summary with User Constrained Clock Root

Index	Clock Net	Root Clock Region	Clock Root Node
1	mmcn/inst/clk0	X0Y0	XIPHY_L_X0Y60/CLK_VDISTR_B0T20
2	mmcn/inst/clkfbout_buf_mmcn_zhold	X0Y0	XIPHY_L_X0Y60/CLK_VDISTR_B0T14


The following summary shows the I/O timing when the clock root is moved.

Figure 72: Timing Summary with User Constrained Clock Root

Setup	Hold
Worst Negative Slack (WNS): -0.217 ns	Worst Hold Slack (WHS): 0.060 ns
Total Negative Slack (TNS): -1.488 ns	Total Hold Slack (THS): 0.000 ns
Number of Failing Endpoints: 16	Number of Failing Endpoints: 0

Synchronous CDC

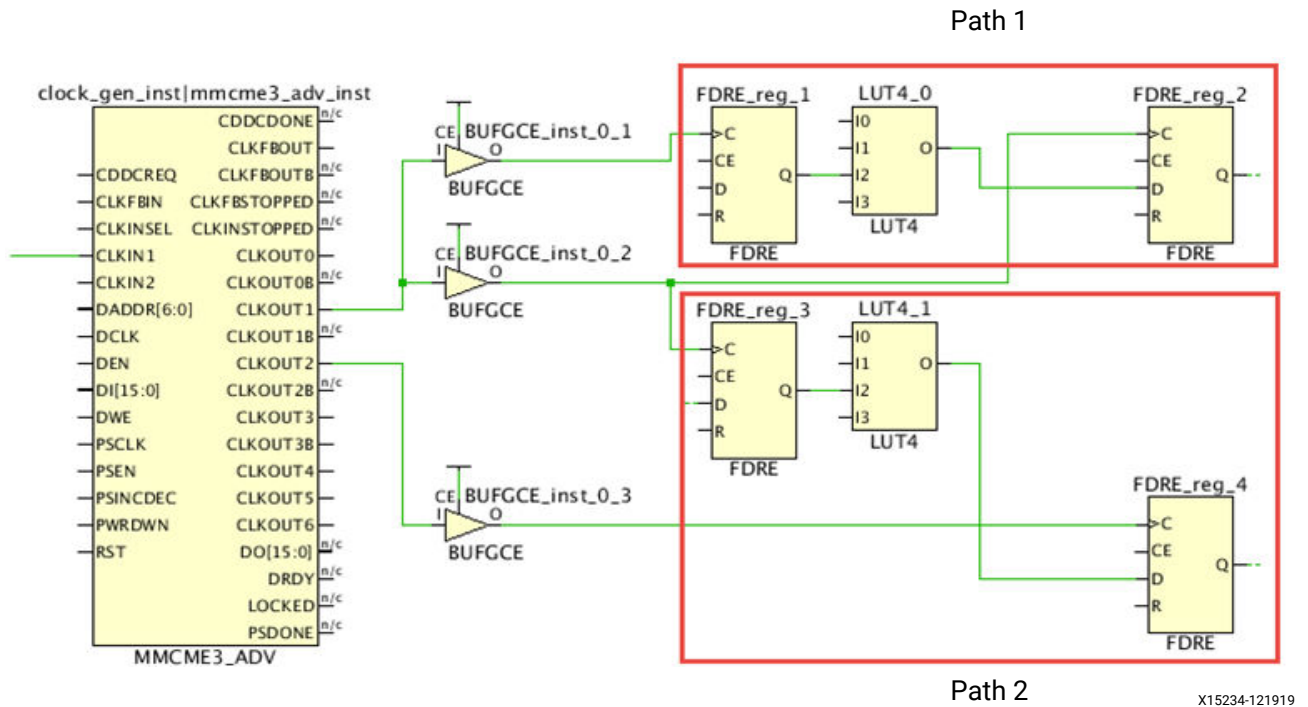
When the design includes synchronous CDC paths between clocks that originate from the same MMCM/PLL, you can use the following techniques to better control the clock insertion delays and skew and therefore, the slack on those paths.

 **IMPORTANT!** *If the CDC paths are between clocks that originate from different MMCM/PLLs, the clock insertion delays across the MMCMs/PLLs are more difficult to control. In this case, Xilinx recommends that you treat these clock domain crossings as asynchronous and make design changes accordingly.*

When a path is timed between two clocks that originate from different output pins of the same MMCM/PLL, the MMCM/PLL phase error adds to the clock uncertainty for the path. For designs using high clock frequencies, the phase error can cause issues with timing closure both for setup and hold.

The following figure shows an example of paths both with and without the phase error. Path 1 is a CDC path clocked by two buffers connected to the same MMCM output and does not include the phase error. Path 2 is clocked by two clocks that originate from two different MMCM outputs and does include the phase error.

Figure 73: MMCM and Phase Error



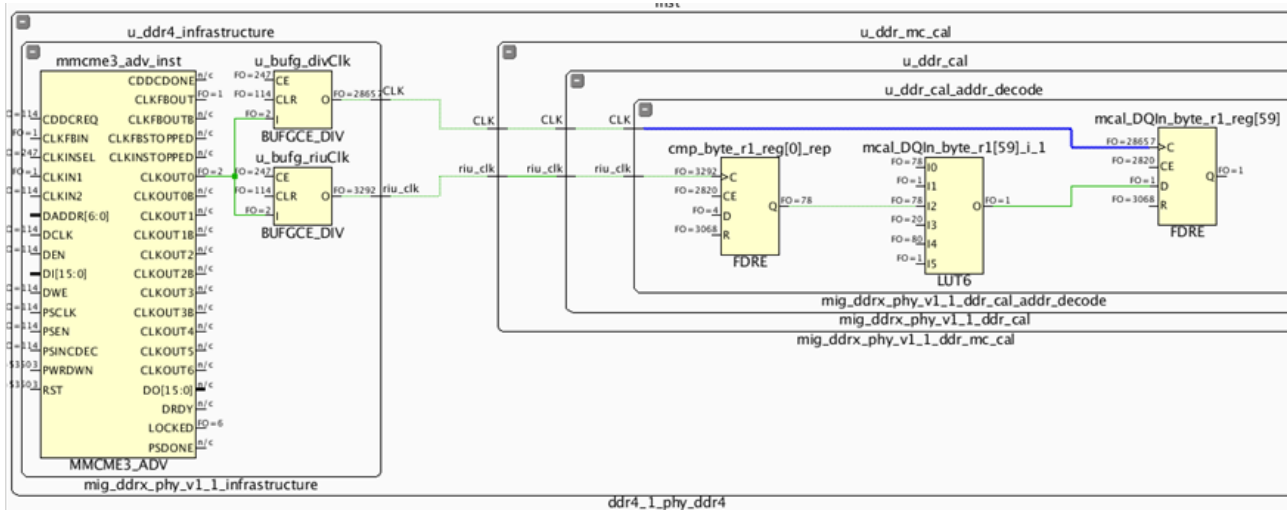
When two synchronous clocks from the same MMCM/PLL have a simple period ratio ($/2$ $/4$ $/8$), you can prevent the phase error between the two clock domains using a single MMCM/PLL output connected to two BUFSGCE_DIV buffers. The BUFSGCE_DIV buffer performs the clock division ($/1$ $/2$ $/4$ $/8$). Other ratios are possible ($/3$ $/5$ $/6$ $/7$) but this requires modifying the clock duty cycle and making mixed edge timing paths more challenging.

Note: Because the BUFSGCE and BUFSGCE_DIV do not have the same cell delays, Xilinx recommends using the same clock buffer for both synchronous clocks (two BUFSGCE or two BUFSGCE_DIV buffers).

The following figure shows two BUFSGCE_DIVs that divide the CLKOUT0 clock by 1 and by 2 respectively.

★ **IMPORTANT!** To ensure safe timing between parallel BUFSGCE_DIV cells where the BUFSGCE_DIVIDE property is set to a value greater than 1, both buffers must use the same enable signal (CE) and the same reset signal (RST). Otherwise, the divided clocks might become phase shifted from one another in hardware, which is not reported by the Vivado tools.

Figure 74: MMCM Synchronous CDC with BUFGE_DIVs Connected to One MMCM Output



To automatically balance several clocks that originate from the same MMCM or PLL, set the same `CLOCK_DELAY_GROUP` property value on the nets driven by the clock buffers that need to be balanced. Following are additional recommendation:

- Avoid setting the `CLOCK_DELAY_GROUP` constraint on too many clocks, because this stresses the clock placer resulting in suboptimal solutions or errors.
- Review the critical synchronous CDC paths in the Timing Summary Report to determine which clocks must be delay matched to meet timing.
- Limit the use of the `CLOCK_DELAY_GROUP` on groups of synchronous clocks with tight requirements and with identical clocking topologies.

IMPORTANT! Xilinx recommends using the Clocking Wizard for creating optimal clocking structures, which use a mix of BUFGEs and BUFGE_DIVs along with related clock grouping constraints.

GT Interface Clocking

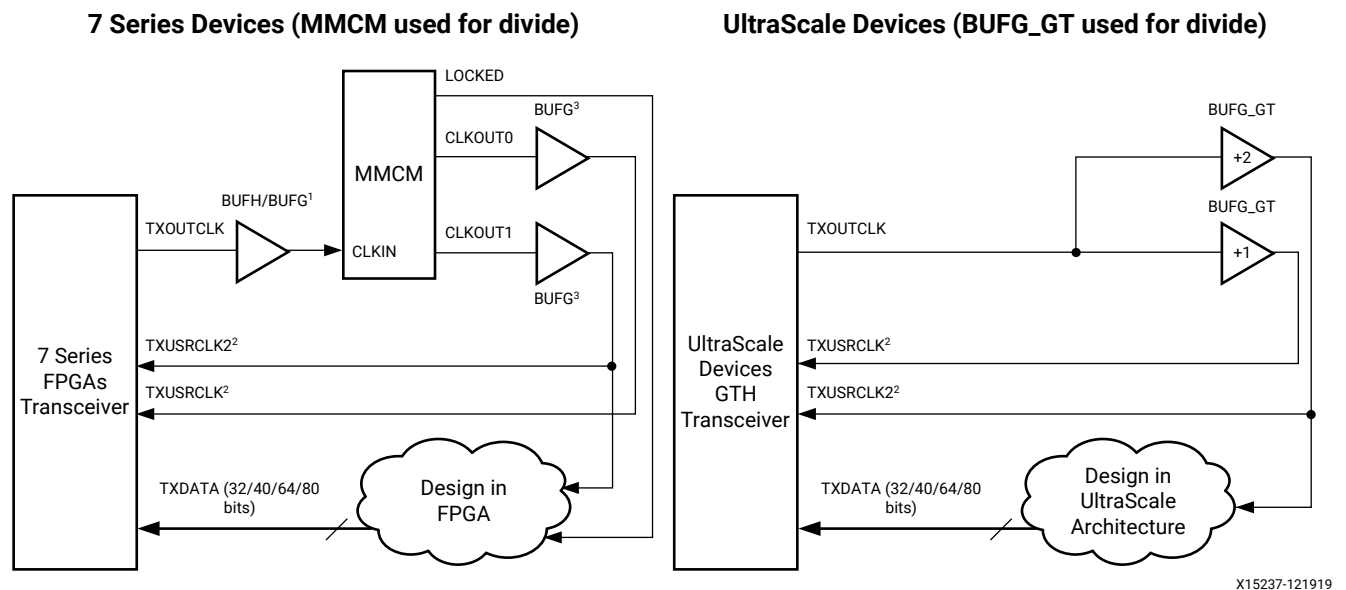
Each GT interface requires several clocks, including some clocks that are shared across bonded `GT*_CHANNEL` cells located in one or several GT quads. UltraScale devices provide up to 128 `GT*_CHANNEL` sites, which can lead to the use of several hundreds of clocks in a design. Most GT clocks have a low fanout with loads placed locally in the clock region next to the associated `GT*_CHANNEL`. Some GT clocks drive loads across the entire device and require the utilization of clock routing resource in many clock regions. The UltraScale architecture includes the following enhancements to efficiently support the high number of GT clocks required.

BUFG_GT with Dynamic Divider

In UltraScale devices, the BUFG_GT buffer simplifies GT clocking. Because the BUFG_GT includes dynamic division capabilities, MMCMs are no longer required to perform simple integer divides on GT output clocks. This saves clocking resources and provides an improved low skew clock path when both a divided GT*_CHANNEL output clock and full-rate clock are required.

You can use the BUFG_GT global clock buffer for GT interfaces where the user logic operates at half the clock frequency of the internal PCS logic or for PCIe interfaces where the GT*_CHANNEL needs to generate multiple clock frequencies for user_clk, sys_clk, and pipe_clk. The following figure compares clocking requirements between 7 series and UltraScale devices for a single-lane GT interface where the frequency of TXUSRCLK2 is equal to half of the frequency of TXUSRCLK.

Figure 75: Clocking Requirements Comparison



You can use any output clock of the GT*_CHANNELS within a Quad or any reference clock generated by an IBUFDS_GTE3/ODIV2 pin within a Quad to drive any of the 24 BUFG_GT buffers located in the same clock region. A BUFG_GT_SYNC is always required to synchronize reset and clear of BUFG_GTs driven by a common clock source.

Note: The Vivado tools automatically insert the BUFG_GT_SYNC primitive if it is not present in the design.

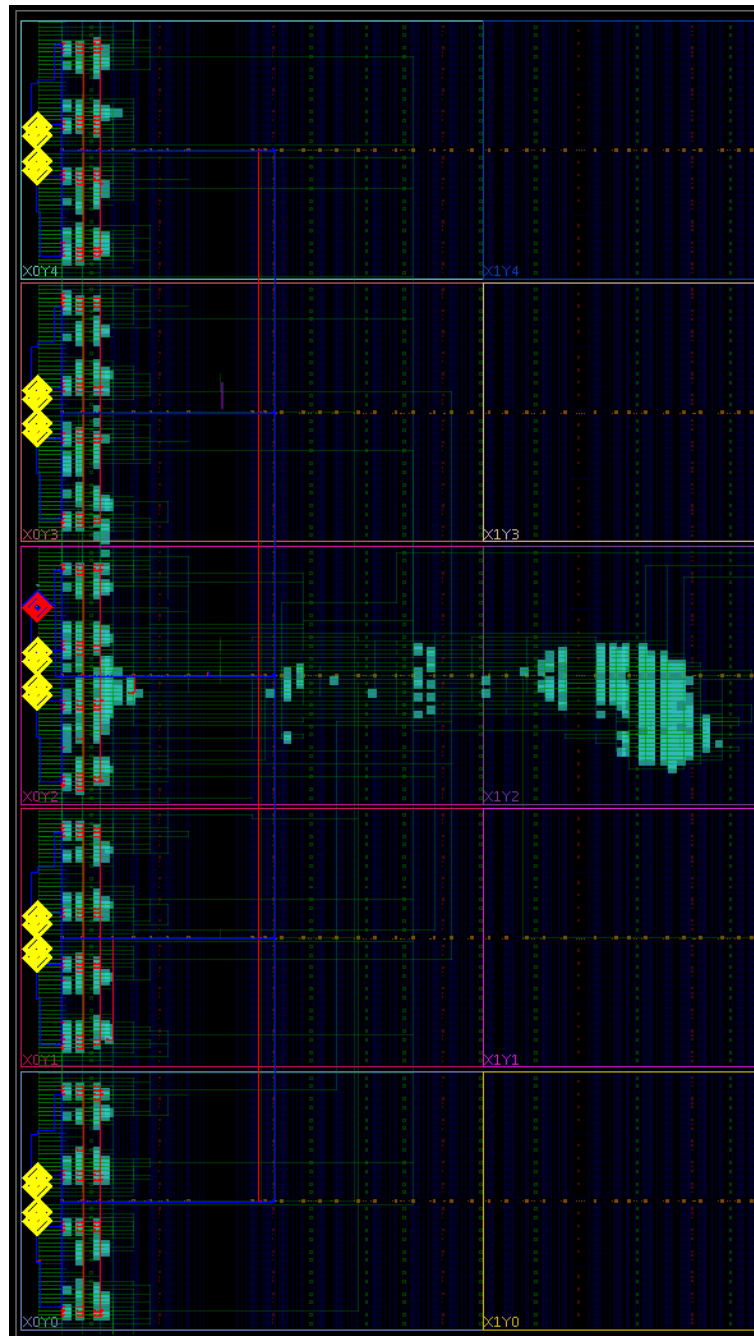
Some applications still require the use of an MMCM to generate complex non-integer clock division of the GT output clocks or the IBUFDS_GTE3/ODIV2 reference clock. In these cases, a BUFG_GT must directly drive the MMCM. By default, the placer tries to place the MMCM on the same clock region row as the BUFG_GT. If other MMCMs try to use the same MMCM site, you must verify that the automated MMCM placement is still as close as possible to the BUFG_GT to avoid wasting clocking resources due to long routes.

Single Quad vs. Multi-Quad Interface

In a multi-channel interface, a master channel can generate [RT]XUSRCLK[2] for all the GT*CHANNELs of the interface. If a multi-channel interface spans multiple quads, the maximum allowed distance for a GT*CHANNEL from the reference clock source is 2 clock regions above and below.

The following figure shows a multi-quad interface. The GT*CHANNELs are marked in yellow, the TXUSRCLK is highlighted in blue, and the TXUSRCLK2 is highlighted in red. The BUFG_GTs driving both TXUSRCLK and TXUSRCLK2 are located in the center quad and are marked in blue and red.

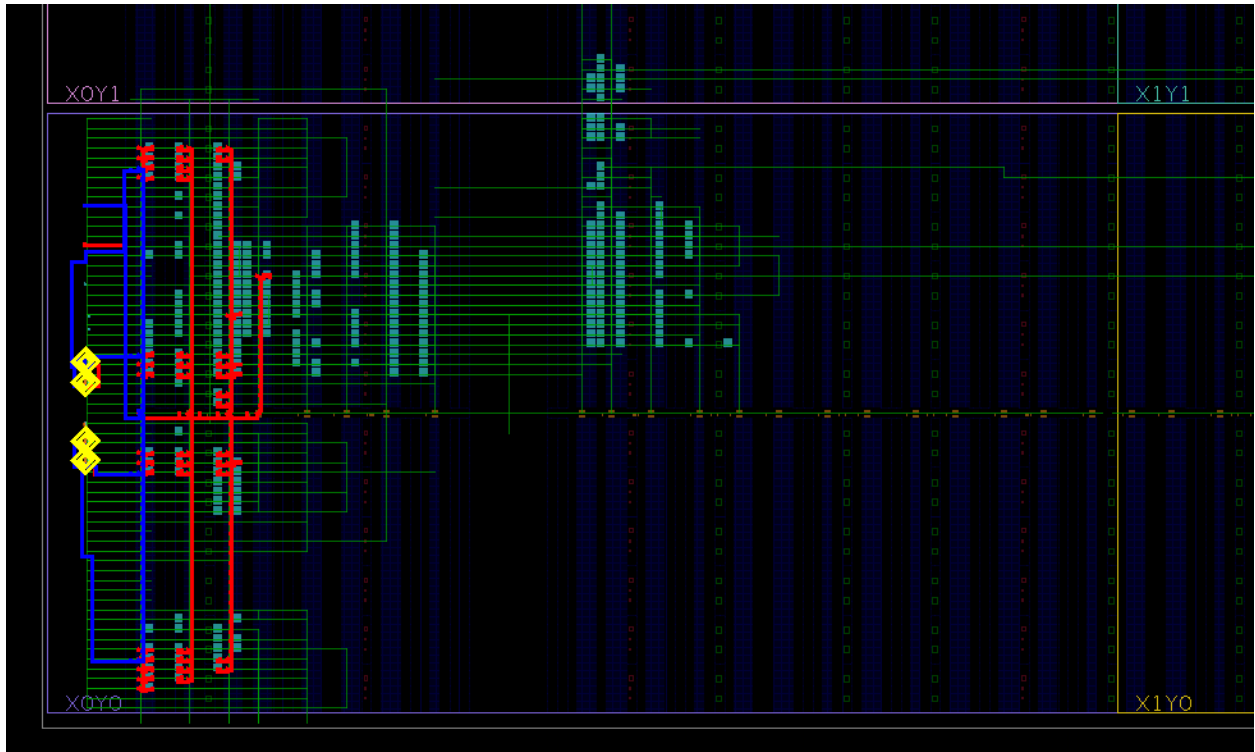
Figure 76: TXUSRCLK/TXUSRCLK2 Clock Routing for a Multi-Quad Interface



If the GT interface is contained within a single Quad, the placer treats the BUFG_GT clocks as local clocks. In this case, the placer attempts to place the BUFG_GT clock loads in the clock regions horizontally adjacent to the BUFG_GT, starting with the clock region that contains the BUFG_GT and potentially using up to half the width of the device.

To override the placer regional clock constraint, assign any of the BUFG_GT clock loads to a Pblock. The following figure shows a single-quad interface. The GT*CHANNELs are marked in yellow, the TXUSRCLK is highlighted in blue, and the TXUSRCLK2 is highlighted in red. All the TXUSRCLK2 loads are placed in the same clock region as the GT*CHANNELs.

Figure 77: TXUSRCLK/TXUSRCLK2 Clock Routing for a Single-Quad Interface



[RT]XUSRCLK/[RT]XUSRCLK2 Skew Matching

When [RT]XUSRCLK2 operates at half the frequency of [RT]XUSRCLK (i.e., separate BUFG_GTs with divide by 1 and divide by 2), a tight skew requirement exists between the [RT]XUSRCLK/[RT]XUSRCLK2 pair at each GT*CHANNEL of a GT interface. To meet the skew requirement, GT*CHANNELs can be a maximum of 2 clock regions above or below the master channel that generates the [RT]XUSRCLK/[RT]XUSRCLK2 pair. In addition, the placer tightly controls skew as follows:

- Assigns the BUFG_GT pairs to the upper or lower 12 BUFG_GTs in a Quad
- Assigns the clock root for both clocks close to the clock region containing the BUFG_GTs



RECOMMENDED: To avoid skew violations, Xilinx highly recommends following this clocking topology when [RT]XUSRCLK2 operates at half the frequency of [RT]XUSRCLK.

Integrated Block for PCI Express CORECLK/PIPECLK/USERCLK Skew Matching

The UltraScale Integrated Block for PCI Express® requires three clocks: CORECLK, USERCLK, and PIPECLK. The three clocks are sourced by BUFG_GTs driven by the TXOUTCLK pin of one of the GT*_CHANNELs of the physical interface. A tight skew requirement exists between the CORECLK and PIPECLK pins and the CORECLK and USERCLK pins. To meet the skew requirement, the placer tightly controls skew as follows:

- Assigns the BUFG_GTs that drive the three PCIe clocks in groups to the upper or lower 12 BUFG_GTs in a Quad
- Assigns the clock root for all three clocks to the same clock region

Note: For more information on PCIe clocking requirements, see the *UltraScale Devices Gen3 Integrated Block for PCI Express LogiCORE IP Product Guide* ([PG156](#)).

7 Series Device Clocking

Note: This section uses Virtex®-7 clocking resources as an example. The clocking resources for Virtex-6 devices are similar. If you are using a different architecture, see the *7 Series FPGAs Clocking Resources User Guide* ([UG472](#)) or the *UltraScale Architecture Clocking Resources User Guide* ([UG572](#)) depending on your device.

Virtex-6 and Virtex-7 devices contain thirty-two global clock buffers known as BUFGs. BUFGs can serve most clocking needs for designs with less demanding needs in terms of number of clocks, design performance, and clocking control. Global clocking resources include BUFG, BUFGCE, BUFGMUX, and BUFGCTRL primitives, which each have their own features. For more information on the features of these global clock components, see the *Clocking Resources Guide (7 Series FPGAs Clocking Resources User Guide)* ([UG472](#)) or *UltraScale Architecture Clocking Resources User Guide* ([UG572](#)) and *Libraries Guide (Vivado Design Suite 7 Series FPGA and Zynq-7000 SoC Libraries Guide)* ([UG953](#)) or *UltraScale Architecture Libraries Guide* ([UG974](#)) for your device.



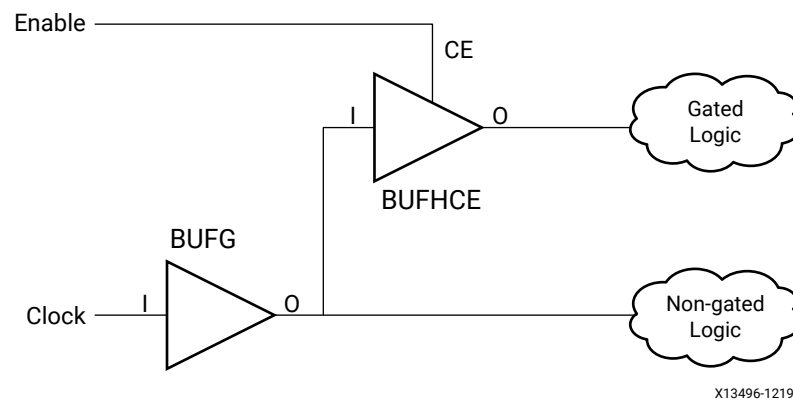
RECOMMENDED: *If clocking demands exceed the number of BUFGs, or if better overall clocking characteristics are desired, analyze the clocking needs against the available clocking resources, and select the best resource for the task.*

In addition to global clocking resources, regional clocking resources are also available, which allow tighter control of clock networks. Regional clocking resources include the Horizontal Clock Region Buffers (BUFH, BUFHCE), Regional Clock Buffer (BUFR), I/O Clock Buffer (BUFIO), and Multi-Regional Clock Buffer (BUFMR). For more information on the features of these regional clock components, see the *Clocking Resources Guide (7 Series FPGAs Clocking Resources User Guide)* ([UG472](#)) or *UltraScale Architecture Clocking Resources User Guide* ([UG572](#)) and *Libraries Guide (Vivado Design Suite 7 Series FPGA and Zynq-7000 SoC Libraries Guide)* ([UG953](#)) or *UltraScale Architecture Libraries Guide* ([UG974](#)) for your device.

Using Horizontal Clock Region Buffers for Clock Gating

You can use the Horizontal Clock Region Buffer (BUFHCE) with BUFGs to perform a medium-grained clock gating function. For portions of a clock domain ranging from a few hundred to a few thousand loads in which you want to stop clocking intermittently, the BUFHCE can be an effective clocking resource. A BUFg can drive multiple BUFHCEs in the same or different clock regions, which allows you to individually control clocking in several low clock skew domains.

Figure 78: Horizontal Clock Region Buffers



When used independently, all loads connected to the BUFH must reside in the same clock region. This makes it well-suited for very high-speed, more fine-grained (fewer loads) clocking needs. BUFHCE can be used to achieve medium-grained clock-gating within the specific clock region. You must ensure that the resources driven by the BUFH do not exceed the available resources in the clock region and that no other conflicts exist.

The phase relationship might be different between the BUFH and clock domains driven by BUFGs, other BUFHs, or any other clocking resource. The single exception is when two BUFHs are driven to horizontally adjacent regions. In this case, the skew between left and right clock regions when both BUFHs driven by the same clock source should have a very controlled phase relationship in which data may safely cross the two BUFH clock domains. BUFHs can be used to gain access to MMCMs or PLLs in opposite regions to a clock input or GT. However, care must be taken in this approach to ensure that the MMCM or PLL is available.

Additional Clocking Considerations for SSI Devices

In general, all clocking considerations mentioned above also apply to SSI technology devices. However, there are additional considerations when targeting these devices due to their construction. When using a BUFMR, it cannot drive clocking resources across an SLR boundary. Accordingly, Xilinx recommends that you place the clocks driving BUFMRs into the bank or clocking region in the center clock region within an SLR. This gives access to all three clock regions on the left or right side of the SLR.

In terms of global clocking, for designs requiring sixteen or fewer global clocks (BUFGs), no additional considerations are necessary. The tools automatically assign BUFGs in a way to avoid any possible contention. When more than 16 (but fewer than 32) BUFGs are required, some consideration to pin selection and placement must be done to avoid any chance of contention of resources based on global clocking line contention and/or placement of clock loads.

As in all other Xilinx 7 series devices, Clock-Capable I/Os (CCIOs) and their associated Clock Management Tile (CMT) have restrictions on the BUFGs they can drive within the given SLR. CCIOs in the top or bottom half of the SLR can drive BUFGs only in the top or bottom half of the SLR (respectively). For this reason, pin and associated CMT selection should be done in a way in which no more than sixteen BUFGs are required in either the top or bottom half of all SLRs collectively. In doing so, the tools can automatically assign all BUFGs in a way to allow all clocks to be driven to all SLRs without contention.

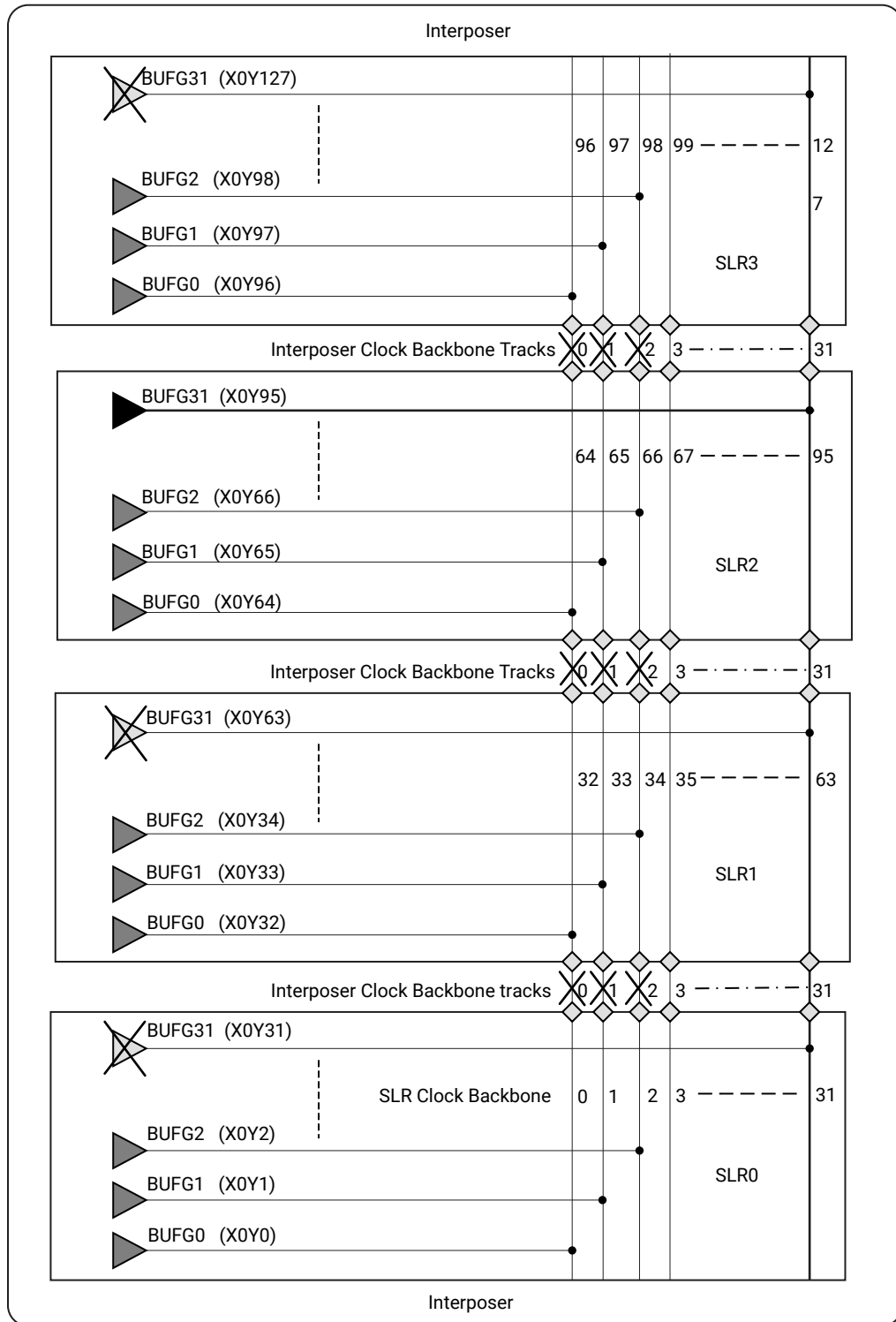
For designs that require more than 32 global clocks, Xilinx recommends that you explore using BUFMRs and BUFHs for smaller clock domains to reduce the number of needed global clock domains. BUFMRs with the use of a BUFMR to drive resources within three clock regions that encompasses one-half of an SLR (approximately 250,000 logic cells in a Virtex-7 class SLR). Horizontally adjacent clock regions may have both left and right BUFH buffers driven in a low-skew manner enabling a clocking domain of one-third of an SLR (approximately 167,000 logic cells).

Using these resources when possible not only leads to fewer considerations for clocking resource contention, but many times improves overall placement, resulting in improved performance and power.

If more than 32 global clocks are needed that must drive more than half of an SLR or to multiple SLRs, it is possible to segment the BUFG global clocking spines. Isolation buffers exist on the vertical global clock lines at the periphery of the SLRs that allow use of two BUFGs in different SLRs that occupy the same vertical global clocking track without contention. To make use of this feature, more user control and intervention is required. In the figure below, BUFG0 through BUFG2 in the three SLRs have been isolated, and hence have independent clocks within their respective SLRs. On the other hand, the BUFG31 line has not been isolated. Hence, the same BUFG31 (located in SLR2 in the figure) drives the clock lines in all the three SLRs - and BUFG31 located in other SLRs should be disabled.

Careful selection and manual placement (LOCs) must be used for the BUFGs. Additionally, all loads for each clock domain must be manually grouped and placed in the appropriate SLR to avoid clocking contention. If all global clocks are placed and all loads managed in a way to not create any clocking contention and allow the clock to reach all loads, this can allow greater use of the global clocking resources beyond 32.

Figure 79: Optional Isolation on Clock Lines for SSI Devices



X14051_122019

Clock Skew for Global Clocking Resources in SSI Technology Devices

Clock skew in any large device might represent a significant portion of the overall timing budget for a given path. Too much clock skew may not only represent issues with maximum clock speed, but may also manifest itself into stringent hold time requirements. Having multiple die in a device worsens the process portion of the PVT equation, but is managed by the Xilinx assembly process in which only die of similar speed are packaged together.

Even with that extra action, the Xilinx timing tools accounts for these differences as a part of the timing report. During path analysis, these aspects are analyzed as a part of the setup and hold calculations, and are reported as a part of the path delay against the specified requirements. No additional user calculations or consideration are necessary for SSI technology devices, because the timing analysis tools consider these factors in their calculations.

Skew can increase if using the top or bottom SLR as the delay-differential is higher among points farther away from each other. For this reason, Xilinx recommends for global clocks that must drive more than one SLR to be placed into the center SLR. This allows a more even distribution of the overall clocking network across the part resulting in less overall clock skew.

When targeting UltraScale devices, there is less repercussion to clock placement. However, it is still highly suggested to place the clock source as close as possible to the central point of the clock loads to reduce clock insertion delay and improve clock power.

Designing the Clock Structure

Now that you understand the major considerations for clocking decisions, let us see how you can achieve the desired clocking for your design.

Inference

Without user intervention, Vivado synthesis automatically specifies a global buffer (BUFG) for all clock structures up to the maximum allowed in an architecture (unless otherwise specified or controlled by the synthesis tool). As discussed above, the BUFG provides a well-controlled, low-skew network suitable for most clocking needs. Nothing additional is required unless your design clocking exceeds the number or capabilities of BUFGs in the part.

Applying additional control of the clocking structure, however, may prove to show better characteristics in terms of jitter, skew, placement, power, performance, or other characteristics.

Synthesis Constraints and Attributes

A simple way to control clocking resources is to use the `CLOCK_BUFFER_TYPE` synthesis constraint or attribute. Synthesis constraints may be used to:

- Prevent BUFG inference.
- Replace a BUFG with an alternative clocking structure.
- Specify a clock buffer where one would not exist otherwise.

Using synthesis constraints allows this type of control without requiring any modification to the code.

Attributes can be placed in either of the following locations:

- Directly in the HDL code, which allows them to persist in the code
- As constraints in the XDC file, which allows this control without any changes needed to the source HDL code

Use of IP

Certain IP assists in the creation of the clocking structures. Clocking Wizard and IO Wizard specifically can assist in the selection and creation of the clocking resources and structure, including:

- BUFG
- BUFGCE
- BUFGCE_DIV (UltraScale devices)
- BUFGCTRL
- BUFIO (7 series devices)
- BUFR (7 series devices)
- Clock modifying blocks such as:
 - Mixed Mode Clocking Manager (MMCM)
 - Phase-locked loop (PLL) components

More complex IP, such as PCIe or Transceivers Wizard IP, might also include clocking structures as part of the overall IP. This might provide additional clocking resources if properly taken into account. If not taken into account, it might limit some clocking options for the remainder of the design.

Xilinx highly recommends that, for any instantiated IP, the clocking requirements, capabilities, and resources are well understood and leveraged where possible in other portions of the design.

Related Information

[Working with Intellectual Property \(IP\)](#)

Instantiation

The most low-level and direct method of controlling clocking structures is to instantiate the desired clocking resources into the HDL design. This allows you to access all possible capabilities of the device and exercise absolute control over them. When using BUFGCE, BUFGMUX, BUFHCE, or other clocking structure that requires extra logic and control, instantiation is generally the only option. However, even for simple buffers, sometimes the quickest way to obtain a desired result is to be direct and instantiate it into your design.

An effective style to manage clocking resources (especially when instantiating) is to contain the clocking resources in a separate entity or module instantiated at the top or near the top of the code. By having it at the top-level of code, it may more easily be distributed to multiple modules in your design.

Be aware of where clocking resources can and should be shared. Creating redundant clocking resources is not only a waste of resources, but generally consume more power, create more potential conflicts and placement decisions resulting in longer overall implementation tool compile times and potentially more complex timing situations. This is another reason why having the clocking resources near the top module is important.



TIP: You can use Vivado HDL templates to instantiate specific clocking primitives.

Related Information

[Using Vivado Design Suite HDL Templates](#)

Controlling the Phase, Frequency, Duty-Cycle, and Jitter of the Clock

This section provides techniques for fine-tuning the clock characteristics.

Using Clock Modifying Blocks (MMCM and PLL)

You can use an MMCM or PLL to change the overall characteristics of an incoming clock. An MMCM is most commonly used to remove the insertion delay of the clock (phase align the clock to the incoming system synchronous data) or for conditioning and controlling the clock characteristics, such as:

- Creating tighter control of phase
- Filtering jitter in the clock
- Changing the clock frequency
- Correcting or changing the clock duty cycle

To use the MMCM or PLL, several attributes must be coordinated to ensure that the MMCM is operating within specifications and delivering the desired clocking characteristics on its output. For this reason, Xilinx highly recommends that you use the Clocking Wizard to properly configure this resource.

You can also directly instantiate the MMCM or PLL, which allows even greater control. However, be sure to use the proper settings to avoid causing the following issues:

- Increasing clock uncertainty due to increased jitter
- Building incorrect phase relationships
- Making timing closure more difficult



IMPORTANT! *When using the Clocking Wizard to configure the MMCM or PLL, the Clocking Wizard by default attempts to configure the MMCM for low output jitter using reasonable power characteristics.*

Depending on your goals, you can change the settings in the Clocking Wizard to further minimize jitter and thus, improve timing at the cost of higher power. Alternatively, you can reduce power but increase output jitter.

While using MMCM or PLL, be sure to do the following:

- Do not leave any inputs floating. Relying on synthesis or other optimization tools to tie off the floating values is not recommended, because the values might be different than expected.
- Connect RST to the user logic, so that it can be asserted by logic controlled by a reliable clocking source. Grounding of RST can cause problems if the clock is interrupted.
- Use LOCKED output in the implementation of reset. For example, hold the synchronous logic clocked from the PLL in reset until LOCKED is asserted. The LOCKED signal must be synchronized before it is used in a synchronous portion of the design. Xilinx recommends adding LOCKED to a processor map so it is visible when debugging.
- Confirm the connectivity between CLKFBIN and CLKFBOUT. The BUFG only needs to be included in the feedback path if the PLL/MMCM output clock needs to be phase aligned with the input reference clock, for example, when using ZHOLD compensation mode.
- To avoid the MMCM or PLL phase error timing penalty on synchronous clock domain crossing paths in UltraScale devices, use BUFGCE_DIVs instead of BUFGCE.



RECOMMENDED: *Explore the different settings within the Clocking Wizard to ensure that the most desirable configuration is created based on your overall design goals.*

Related Information

[Synchronous CDC](#)

Using IDELAYs on Clocks to Control Phase

For 7 series devices, if only minor phase adjustments are necessary, you can use IDELAY or ODELAY (instead of MMCM or PLL) to add additional delay. This increases the phase offset of the clock in relation to any associated data. When using UltraScale devices, you cannot use an IDELAY on an input clock source. Therefore, if phase manipulation is necessary, Xilinx recommends using an MMCM.

Using Gated Clocks

Xilinx devices include dedicated clock networks that can provide a large-fanout, low-skew clocking resource. Fine-grained clock gating techniques included in the HDL code can disrupt the functionality and prevent efficient use of the dedicated clocking resources. Therefore, when writing HDL to target a device, Xilinx does not recommend that you code clock gating constructs into the clock path. Instead, control clocking by using coding techniques to infer clock enables to stop portions of the design, either for functionality or power reasons.

Converting Clock Gating to Clock Enable

If the code already contains clock gating constructs, or if it is intended for a different technology that requires such coding styles, Xilinx recommends that you use a synthesis tool that can remap gates placed within the clock path to clock enables in the data path. Doing so allows for a better mapping to the clocking resources; and simplifies the timing analysis of the circuit for data entering and exiting the gated domain. For example, use the `-gated_clock_conversion auto` option with Vivado synthesis to attempt automatic conversion to register clock enable logic. For the complex gated clock structures, use the `GATED_CLOCK` attribute in the RTL code to guide Vivado synthesis.

Gating the Clock Buffer

When larger portions of the clock network can be shut down for periods of time, you can enable or disable the clock network using a BUFGCE or BUFGCTRL. In addition, when targeting UltraScale devices, you can gate the BUFGCE_DIV and BUFG_GT. For 7 series devices, you can also use the BUFHCE, BUFR, and BUFMRCE to gate the clock.

When a clock can be slowed down during periods of time, you can also use these buffers with additional logic to periodically enable the clock net. Alternatively, you can use a BUFGMUX or BUFGCTRL to switch the clock source from a faster clock signal to a slower clock.

Any of these techniques can effectively reduce dynamic power. However, depending on the requirements and clock topology, one technique may prove more effective than another. For example, in 7 series devices:

- A BUFR might work best if it is an externally generated clock (under 450 MHz) that is only needed to source up to three clock regions.

- For Virtex-7 devices, a BUFMRCE might also be needed to use this technique with more than one clock region (but only up to three vertically adjacent regions).
- A BUFHCE is better suited for higher-speed clocks that can be contained in a single clock region. Although a BUFGCE may span the device and is the most flexible approach, it might not be the best choice for the greatest power savings.

Controlling and Synchronizing Device Startup

After the device completes configuration, a sequence of events occurs in which the device completes the configuration state and enters into general operation. In most configuration sequences, one of the last steps is the deassertion of the Global Set Reset (GSR), followed by the deassertion of the Global Enable (GWE) signal. When this happens, the design is in a known initial state and is then released for operation.

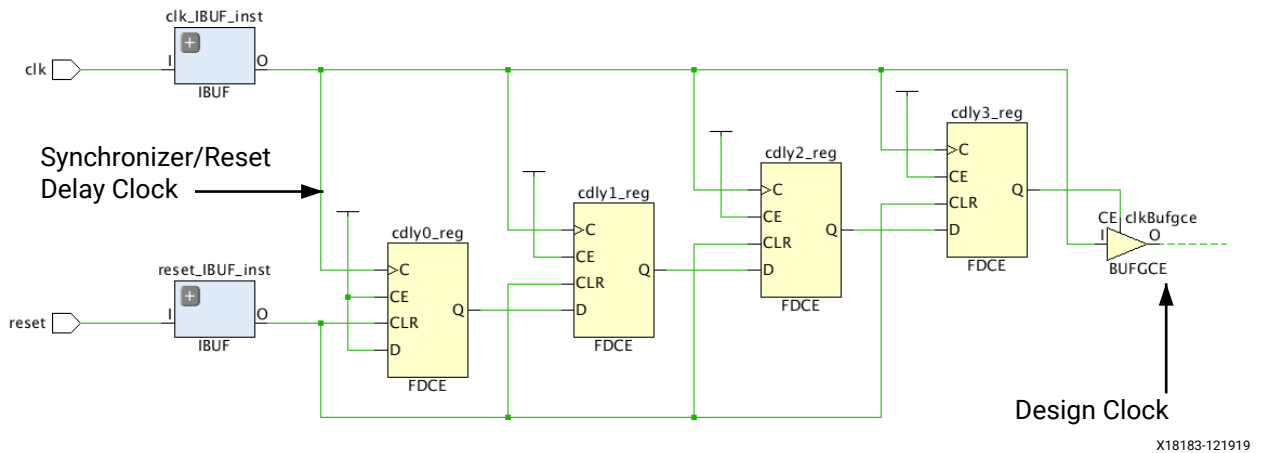
If this release point is not synchronized to the given clock domain or if the clock is operating at a faster time than the GWE can safely be released, portions of the design can go into an unknown state. For some designs, this does not matter. In other designs, this can cause the design to become unstable or to incorrectly process the initial data set.

If the design must start up in a known state, Xilinx recommends that you take action to control the start-up synchronization process using any of the following methods:

- Use clock enables, local reset (synchronized), or both, on critical parts of the design, such as a state machine, to ensure that the start-up of those portions of the design are controlled and known.
- Use instantiated clock buffer components with clock enable capability.

Delay the reset release by as many cycles as needed before enabling the design clock. The following example shows how to delay the first design clock edge after the reset is released in an UltraScale device. By setting `ASYNC_REG=TRUE` on the synchronizer registers, all registers are placed in a single SLICE and therefore, do not need to be driven by a global clock resource. To prevent clock buffer insertion on the synchronizer clock, use the `CLOCK_BUFFER_TYPE=NONE` property on the input clock port.

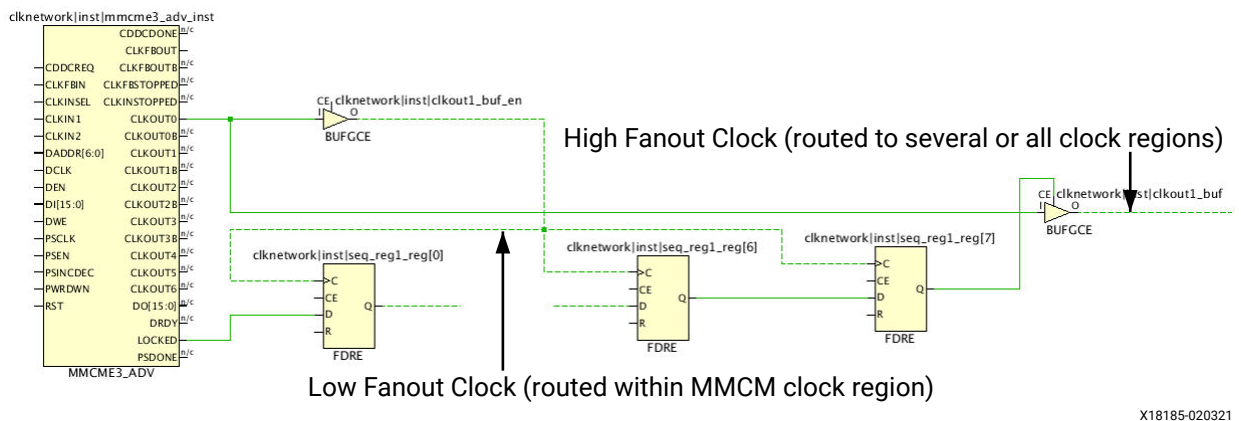
Figure 80: Reset Synchronization and Delay for Safe Clock Startup Example



- When using an MMCM, you can select the **Safe Clock Startup** option from the Clocking Wizard to ensure that design clocks are enabled only after they are stable and reliable.

The following example shows the synchronization stages of an UltraScale device MMCM LOCKED signal connected to the CE pin of the BUFCE, which drives the user logic. A second BUFCE is connected in parallel to the high fanout BUFCE (user clock) and is dedicated to the logic controlling the BUFCE/CE pin. This topology helps timing closure on the BUFCE/CE in UltraScale devices by minimizing the clock skew between the synchronizer and the BUFCE pin.

Figure 81: MMCM Safe Clock Startup Example



TIP: If the MMCM or PLL compensation mode is set to ZHOLD or BUF_IN, all clocks from CLKOUT0 are grouped with the feedback clock and use the same CLOCK_ROOT. If this introduces timing violations on BUFCE/CE, create a CLOCK_DELAY_GROUP constraint between the high fanout clock and the feedback clock only. Optionally, you can also set a USER_CLOCK_ROOT constraint on the low fanout clock net to constrain the loads to the same clock region as the MMCM. For 7 series devices, the second clock buffer is usually not needed for helping timing closure due to the different clocking architecture.

Avoiding Local Clocks

Local clocks are clock nets routed with regular fabric resources instead of dedicated global clocking resources. In most cases, the Vivado synthesis and Vivado logic optimization tools insert clock buffers where mandated by the architecture or for clock nets with more than 30 clock loads. Local clocks typically occur when:

- A global clock is divided by a counter implemented with fabric logic
- Clock gating conversion is not able to remove all LUTs from the clock path
- Too many clock buffers are used in 7 series devices

Note: UltraScale devices have more clock buffers than 7 series devices, and high utilization of low fanout clock buffers is usually not a concern.

In general, avoid using local clocks. Local clocks introduce several challenges to the implementation tools:

- Unpredictable clock skew, leading to difficult timing closure
- Increase of low to medium fanout nets that are processed with special care by the router, leading to potential routability problems



TIP: If local clocks introduce timing QoR problems, try floorplanning the clock driver and loads to a small area using a Pblock. Use `report_clock_utilization` to identify the location of the local clocks, review the clock placement, and decide on how to reduce their number or impact.

Creating an Output Clock

An effective way to forward a clock out of a device for clocking devices external to the device, is to use an ODDR component. By tying one of the inputs High and the other Low, you can easily create a well controlled clock in terms of phase relationship and duty cycle (for example, by holding D1 to 0 and the D2 pin to 1, you can achieve a 180 degree phase shift). By utilizing the set/reset and clock enable, you also have control over stopping the clock and holding it at a certain polarity for sustained amounts of time.

If further phase control is necessary for an external clock, an MMCM or PLL can be used with external feedback compensation and/or coarse or fine grained, fixed or variable phase compensation. This allows great control over clock phase and propagation times to other devices simplifying external timing requirements from the device.

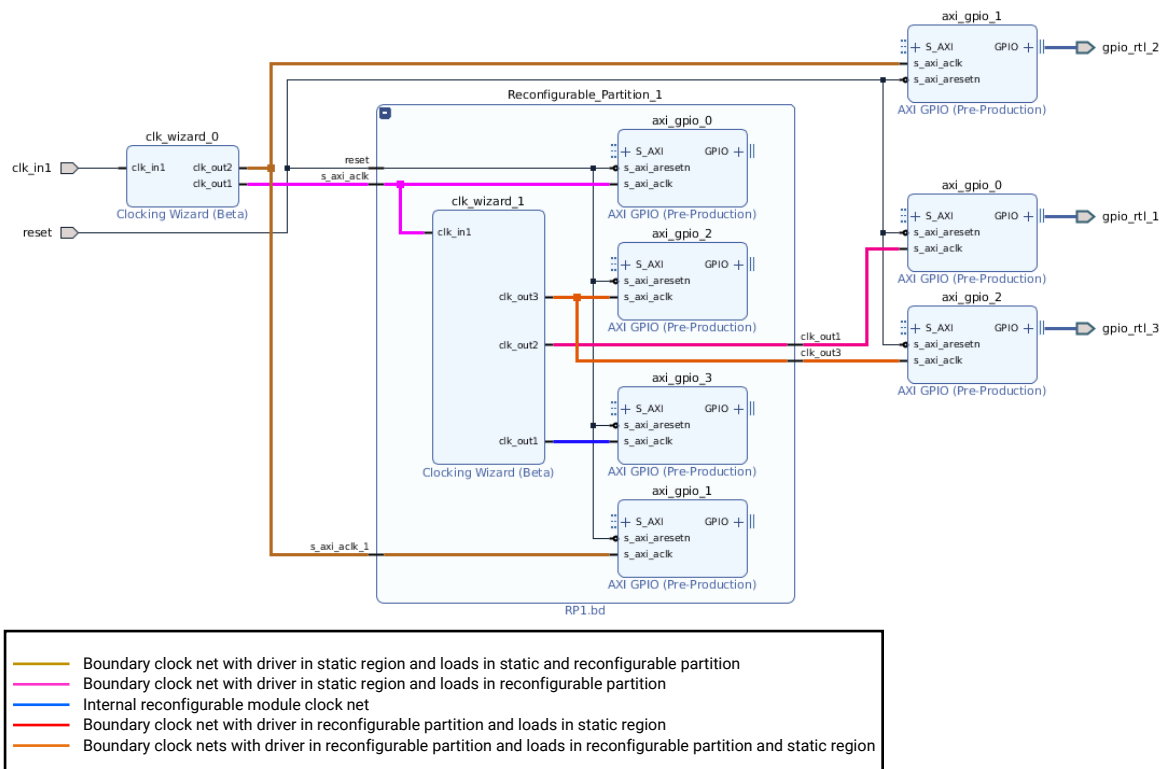
Clocking Recommendations for Platforms and Dynamic Function eXchange

This section covers the clocking guidelines for Dynamic Function eXchange (DFX) designs. In general, clocks in a DFX design are categorized as internal clocks and boundary clocks:

- **Reconfigurable module internal clocks:** Clocks with driver and loads inside the reconfigurable module (RM).
- **Boundary clocks:** Clocks with nets crossing the cell boundary of the reconfigurable module as follows:
 - Driver in the static region and loads in the RM
 - Driver in the RM and loads in the static region
 - Driver in the static region and loads distributed between RM and static region
 - Driver in the RM region and loads distributed between RM and static region

The following figure shows an example of the different boundary clocks.

Figure 82: DFX Clock Tile Sharing



X25409-062421

For more information on DFX, see the *Vivado Design Suite User Guide: Dynamic Function eXchange (UG909)*.

DFX Behavior for Clock Nets

Reconfigurable Module Internal Clock Nets

In a reconfigurable module (RM) internal clock net, the clock root is placed at the center of the loads inside the reconfigurable partition (RP) Pblock. This clock root placement offers more flexibility for placement and routing of the RM internal clock in subsequent implementations. Xilinx recommends this approach whenever possible to achieve better skew and optimal clock root placement.

Boundary Clock Nets

After the first implementation, boundary clock net tracks are locked. The partition pin location constraints (PPLOCs) on the boundary clock nets are distributed to all clock regions covered by the reconfigurable partition (RP) Pblock.

The clock root of the boundary clock net can be placed anywhere in the device, because the boundary clock net can drive both static and RP loads. Xilinx recommends using the `USER_CLOCK_ROOT` constraint on the boundary clock net to manually constrain the `CLOCK_ROOT` location due to the following:

- If the loads of the boundary clock are located mainly in the static region, the clock root might be placed in the static region.
- If the first implementation uses training logic in the RP Pblock, boundary clock nets might be locked down after the first implementation with an off-center clock root location.
- Because the boundary clock net is distributed to all clock regions covered by the RP Pblock, the clock insertion delay for the boundary clock is relatively high compared with the internal RM clock nets.

Clock Domain Crossing

The clock domain crossing (CDC) circuits in the design directly impact design reliability. You can design your own circuits, but the Vivado Design Suite must recognize the circuit and you must apply the `ASYNC_REG` attributes correctly. Xilinx provides XPMs to ensure correct circuit design, including:

- Driving specific features in `place_design` that reduce mean time between failures (MTBF) on synchronization circuits.
- Ensuring recognition by `report_synchronizer_mtbfs`.
- Avoiding `report_cdc` errors and warnings, which typically show up late in the design cycle when iterations are longer.



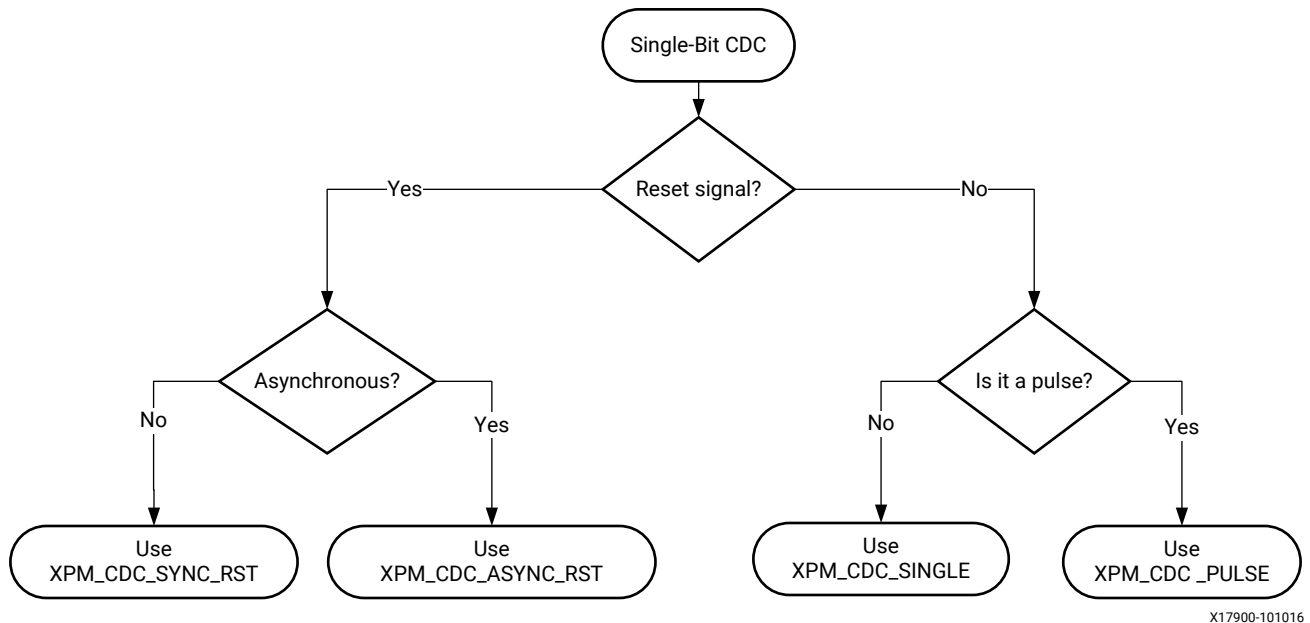
TIP: For CDC violations that can be safely ignored, you can use the waiver mechanism to waive the violations. For details, see this [link](#) in the Vivado Design Suite User Guide: Design Analysis and Closure Techniques (UG906).

A CDC circuit is required when crossing between two asynchronous clocks or when attempting to relax timing between two synchronous clocks by adding false path constraints. When using XPMs, you can select a single-bit or a multi-bit bus to cross between the domains.

Single-Bit CDC

The following figure shows the decisions required when using a single-bit crossing.

Figure 83: Single-Bit CDC Decision Tree

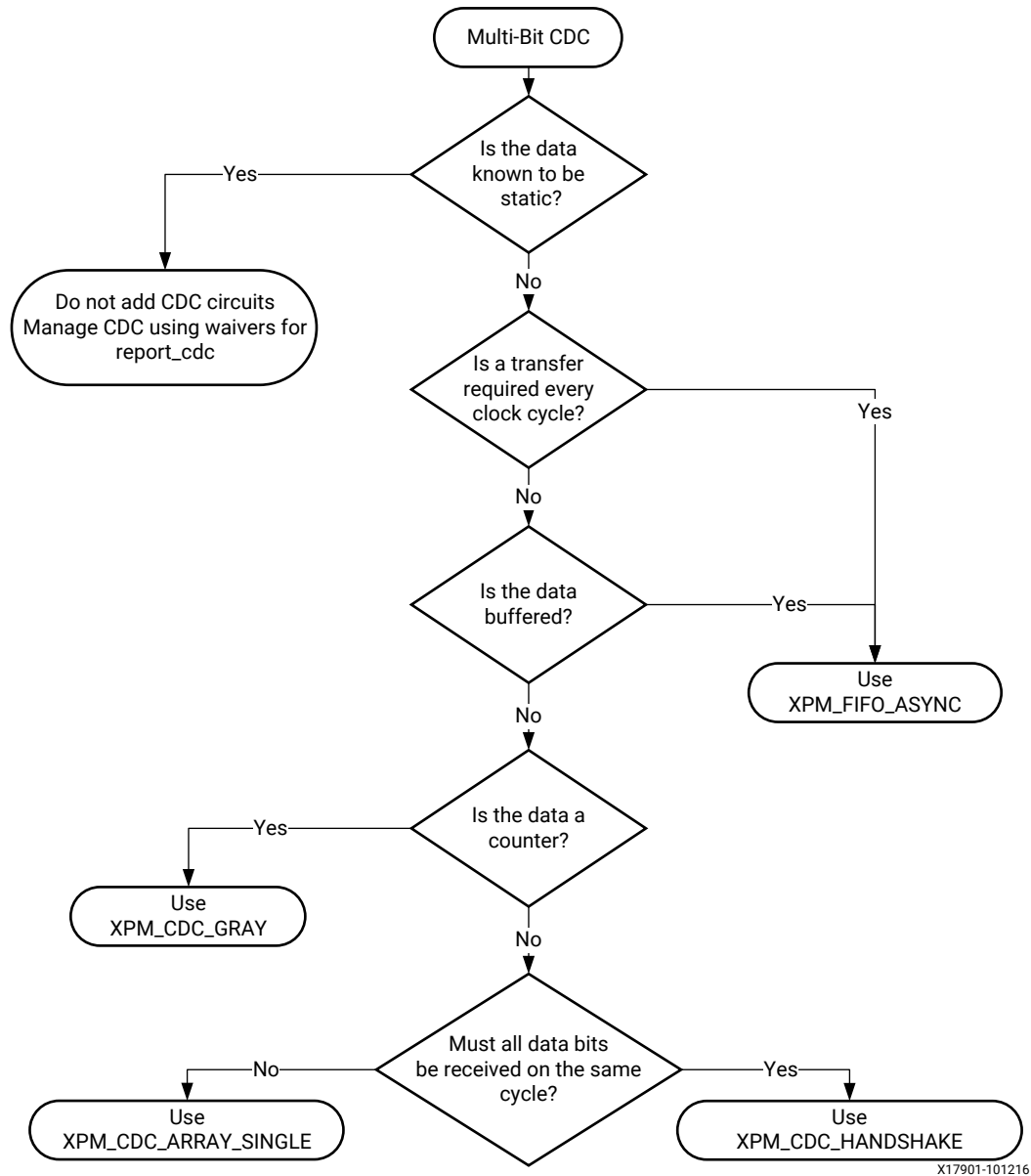


Note: For more information on the different single-bit synchronizers, see the Libraries Guide for your device.

Multi-Bit CDC

The following figure shows the decisions required when using a multi-bit crossing.

Figure 84: Multi-Bit CDC Decision Tree



Note: For more information on the different multi-bit synchronizers, see the Libraries Guide for your device.

Optimizing for MTBF

The total MTBF of a design is a function of:

- Synchronizer MTBF
- Device failure in time (FIT) rate due to single-event upsets (SEUs)

Note: The device FIT rate due to SEUs largely depends on process and device size.

The synchronizer MTBF is design dependent and varies with the following:

- Number of asynchronous CDC points
- Number of synchronizer stages at each crossing point
- Frequency of the destination FF
- Toggle rate of the source

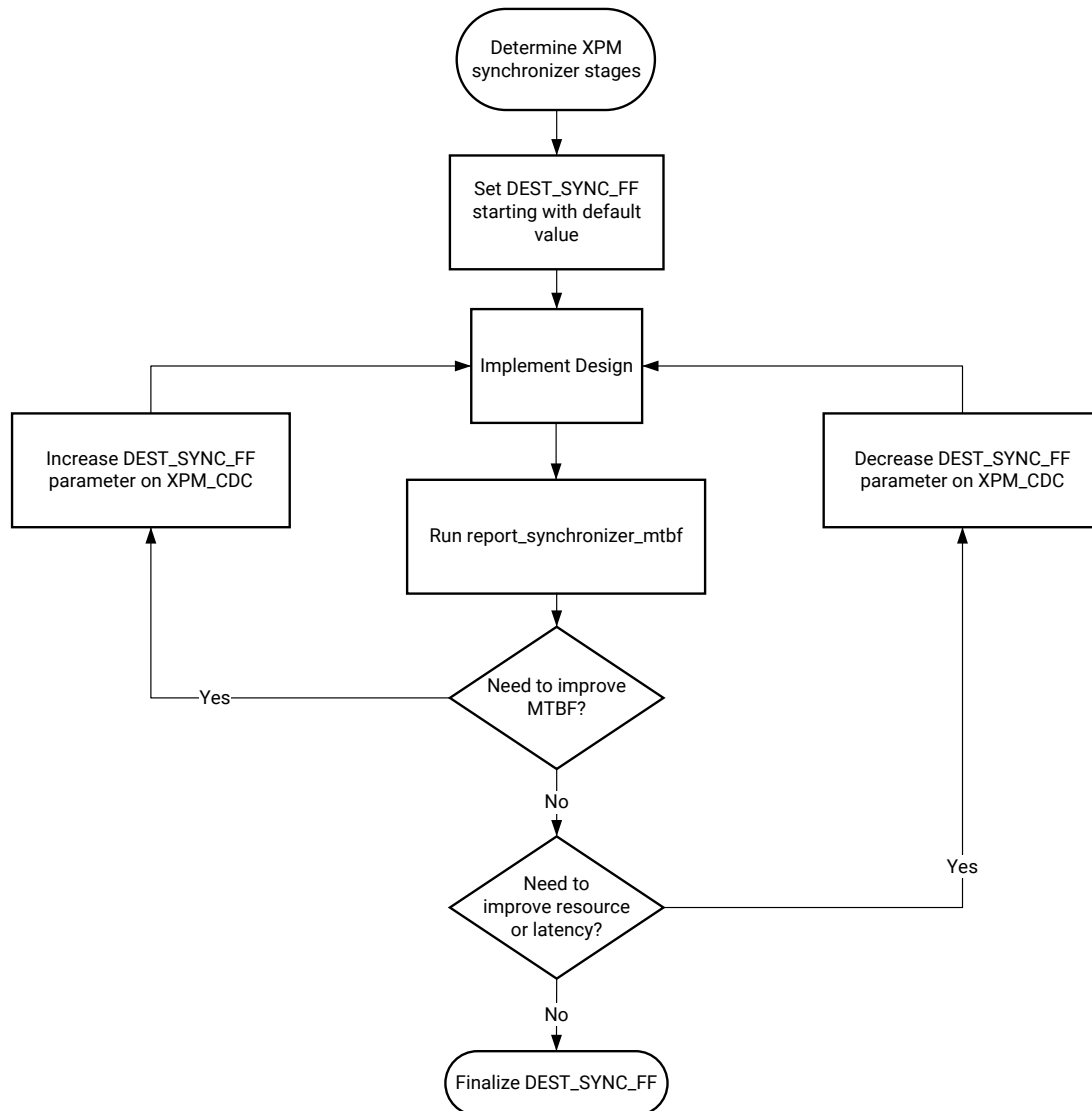
Selecting the Correct Value for the DEST_SYNC_FF Parameter

The `DEST_SYNC_FF` parameter sets the number of metastability protection registers when using an XPM CDC module. The value of this register influences MTBF, design size, and latency at the crossing point. Selecting the correct value of this register is an iterative process that requires the following:

1. Run the design through the Vivado Design Suite implementation flow.
2. Based on your targeted device, do one of the following:
 - For 7 series devices, select the default value for `DEST_SYNC_FF`. This is a conservative approach to meeting typical reliability requirements. For critical designs, conduct further analysis.
 - For UltraScale devices, run the `report_synchronizer_mtbfs` command, which reports the MTBF for the entire design. By iterating through the flow as shown in the following figure, you can find a suitable trade-off between MTBF, latency, and resources.

Note: You can also use this iterative process for a user CDC circuit in which the `ASYNC_REG` attribute is correctly applied to all the synchronization registers.

Figure 85: Synchronizer MTBF Optimization Flow for UltraScale Device



X17899-122019

Constraining the Design Correctly

XPM CDCs provide their own `set_max_delay -datapath_only` constraints. XPM CDCs are not compatible with the `set_clock_groups` constraint, which has a higher precedence and will overwrite the constraints in the XPM.

Related Information

[Defining Clock Groups and CDC Constraints](#)

Design Constraints

Design constraints define the requirements that must be met by the compilation flow for the design to be functional in hardware. For complex designs, constraints also define guidance for the tools to help with convergence and closure. Not all constraints are used by all steps in the compilation flow. For example, physical constraints are used only during the implementation steps: optimization, placement, and routing.

Because synthesis and implementation algorithms are timing-driven, creating proper timing constraints is essential. Over-constraining or under-constraining your design makes timing closure difficult. You must use reasonable constraints that correspond to your application requirements. For more information on constraints, see the following resources:

- *Vivado Design Suite User Guide: Design Analysis and Closure Techniques (UG906)*
- Applying Design Constraints video tutorials available from the [Vivado Design Suite Video Tutorials](#) page on the Xilinx® website

Note: Traditional and platform-based design flows use design constraints in a similar manner. However, platform-based designs require extra attention for signals crossing the boundary from the static region of the design to the dynamic region of the design. Constraining these signals properly ensures flexibility of the platform and minimizes platform revisions.

Organizing the Design Constraints

The constraints are usually organized by category, by design module, or both, in one or many files. Regardless of how you organize them, you must understand their overall dependencies and review their final sequence once loaded in memory. For example, because timing clocks must be defined before they can be used by any other constraints, you must make sure that their definition is located at the beginning of your constraint file, in the first set of constraint files loaded in memory, or both.

Recommended Constraint Files

There are many ways to organize your constraints depending on the size and complexity of your project. Following are a few suggestions.

Simple Design

For a simple design with a small team of designers:

- 1 file for all constraints
- 1 file for physical + 1 file for timing
- 1 file for physical + 1 file for timing (synthesis) + 1 file for timing (implementation)

Complex Design

For a complex design with IP cores or several designer teams:

- 1 file for top-level timing + 1 file for top-level physical + 1 file per IP/major block

Validating the Read Sequence

After you settle on the organization of your project constraint files, you must validate the read sequence of the files depending on the content of the files. In Project Mode, you can modify the constraint file sequence in the Vivado® IDE or by using the `reorder_files` Tcl command. In Non-Project Mode, the sequence is directly defined by the `read_xdc` (for XDC files) and `source` (for constraints generated by Tcl scripts) commands in your compilation flow Tcl script.

Recommended Constraints Sequence

The constraints language (XDC) is based on Tcl syntax and interpretation rules. Like Tcl, XDC is a sequential language:

- Variables must be defined before they can be used. Similarly, timing clocks must be defined before they can be used in other constraints.
- For equivalent constraints that cover the same paths and have the same precedence, the last one applies.
- When a path is covered by multiple timing exceptions, the constraint with the higher precedence applies.

When considering the priority rules above, the timing constraints should overall use the following sequence:

```
## Timing Assertions Section
# Primary clocks
# Virtual clocks
# Generated clocks
# Delay for external MMCM/PLL feedback loop
# Clock Uncertainty and Jitter
# Input and output delay constraints
# Clock Groups and Clock False Paths
## Timing Exceptions Section
```

```
# False Paths
# Max Delay / Min Delay
# Multicycle Paths
# Case Analysis
# Disable Timing
```

When multiple XDC files are used, you must pay particular attention to the clock definitions and validate that the dependencies are ordered correctly.

The physical constraints can be located anywhere in any constraint file.

Creating Synthesis Constraints

Synthesis takes the RTL description of the design and transforms it into an optimized technology mapped netlist by using timing-driven algorithms. The quality of the results is affected by the quality of the RTL code and the constraints provided. At this point of the compilation flow, the net delay modeling is approximate and does not reflect placement constraints or complex effects such as congestion. The main objective is to obtain a netlist which meets timing, or fails by a small amount, with realistic and simple constraints.

The synthesis engine accepts all XDC commands, but only some have a real effect:

- Timing constraints related to setup/recovery analysis influence the QoR:
 - `create_clock / create_generated_clock`
 - `set_input_delay / set_output_delay`
 - `set_clock_groups / set_false_path / set_max_delay / set_multicycle_path`
- Timing constraints related to hold and removal analysis are ignored during synthesis:
 - `set_min_delay / set_false_path -hold / set_multicycle_path -hold`
- RTL attributes forces decisions made by the mapping and optimization algorithms. Following are a few examples:
 - `DONT_TOUCH / KEEP / KEEP_HIERARCHY / MARK_DEBUG`
 - `MAX_FANOUT`
 - `RAM_STYLE / ROM_STYLE / USE_DSP / SHREG_EXTRACT`
 - `FULL_CASE / PARALLEL_CASE` (Verilog RTL only)

Note: The same attribute can also be set as a property from an XDC file. Using XDC-based constraints is convenient for influencing the synthesis results only in some cases without changing the RTL.
- Physical constraints are ignored (LOC, BEL, Pblocks)

Synthesis constraints must use names from the elaborated netlist, preferably ports and sequential cells. During elaboration, some RTL signals can disappear and it is not possible to attach XDC constraints to them. In addition, due to the various optimizations after elaboration, nets or logical cells are merged into the various technology primitives such as LUTs or DSP blocks. To know the elaborated names of your design objects, click **Open Elaborated Design** in the Flow Navigator and browse to the hierarchy of interest.

Some registers are absorbed into RAM blocks and some levels of the hierarchy can disappear to allow cross-boundary optimizations.

Any elaborated netlist object or level of hierarchy can be preserved by using a DONT_TOUCH, KEEP, KEEP_HIERARCHY, or MARK_DEBUG constraint, at the risk of degrading timing or area QoR.

Finally, some constraints can conflict and cannot be respected by synthesis. For example, if a MAX_FANOUT attribute is set on a net that crosses multiple levels of hierarchy, and some hierarchies are preserved with DONT_TOUCH, the fanout optimization will be limited or fully prevented.



IMPORTANT! *Unlike during implementation, RTL netlist objects that are used for defining timing constraints can be optimized away by synthesis to allow better area QoR. This is usually not a problem as long as the constraints are updated and validated for implementation. But if needed, you can preserve any object by using the KEEP constraint so that the constraints will apply during both synthesis and implementation.*

After synthesis is complete, Xilinx recommends that you review the timing and utilization reports to validate that the netlist quality meets the application requirements and can be used for implementation.

Creating Implementation Constraints

The implementation constraints must accurately reflect the requirements of the final application. Physical constraints such as I/O location and I/O standard are dictated by the board design, including the board trace delays, as well as the design internal requirements derived from the overall system requirements. Before you proceed to implementation, Xilinx highly recommends that you validate the correctness and accuracy of all your constraints. An improper constraint will likely contribute to degradation of the implementation QoR and can lower the confidence level in the timing signoff quality.

In many cases, the same constraints can be used during synthesis and implementation. However, because the design objects can disappear or have their name changed during synthesis, you must verify that all synthesis constraints still apply properly with the implementation netlist. If this is not the case, you must create an additional XDC file containing the constraints that are valid for implementation only.

Creating Block-Level Constraints

When working on a multi-team project, it is convenient to create individual constraint files for each major block of the top-level design. Each of these blocks is usually developed and validated separately before the final integration into one or many top-level designs.

The block-level constraints must be developed independently from the top-level constraints, and must be as generic as possible so that they can be used in various contexts. In addition, these constraints must not affect any logic that is beyond the block boundaries.

When implementing a sub-block it is desirable to have the full clocking network included in timing analysis to ensure accurate skew and clock domain crossing analysis. This might require an HDL wrapper containing the clocking components and an additional constraint file to replicate top level clocking constraints. It is used only in the timing validation of the sub-module.

For more information on constraints scoping as well as rules, guidelines, and mechanisms for loading the block-level constraints into the top-level design, see this [link](#) in the *Vivado Design Suite User Guide: Using Constraints (UG903)*.

Specifying Constraints for the Vitis Environment

In the Vitis™ environment, you can specify the hardware kernel as a C/C++ kernel or as an RTL kernel:

- When using a C/C++ kernel, you must specify additional user constraints for synthesis or implementation using Vitis HLS. The Vitis HLS output must then be packaged in the IP packager, and this packaged IP includes both the user and tool-generated constraints. For information, see the *Vitis HLS User Guide (UG1399)*.
- When using an RTL kernel, you must specify additional synthesis and implementation constraints during IP packaging. For information, see the *Vivado Design Suite User Guide: Creating and Packaging Custom IP (UG1118)*.

In the Vitis environment, all of the design constraints for synthesis and implementation must be packaged with the IP. If additional constraints are required for synthesis after the IP is packaged, you must repackage the IP to include the missing constraints.

However, after the IP is packaged, you can specify additional XDC constraints to be used only during implementation. Although the Vitis environment abstracts the underlying Vivado tools process for implementing the programmable logic region, the Vitis environment also provides advanced options to control the Vivado tools flow. With these advanced controls, you can specify certain Tcl scripts to be executed before (Pre) or after (Post) each implementation phase, including the following: `init_design`, `opt_design`, `place_design`, `phys_opt_design`, `route_design`, or `write_bitstream`. For more information on Tcl scripting, see the *Vivado Design Suite User Guide: Using Tcl Scripting (UG894)*. You can leverage the Pre and Post Tcl scripts to execute certain Vivado tools commands, such as to apply additional XDC constraints through the `read_xdc` or `source` Tcl commands.

You can specify the Pre and Post Tcl scripts either through the Vitis environment configuration file or directly on the v++ compiler command line.

To specify the Pre and Post Tcl scripts inside the Vitis environment configuration file, use the parameters `prop=run.impl_1.STEP.<PHASE>.TCL.<PRE|POST>` inside the `[vivado]` section.

Where:

- `<PHASE>` specifies the implementation phase: `INIT_DESIGN`, `OPT_DESIGN`, `PLACE_DESIGN`, `PHYS_OPT_DESIGN`, `ROUTE_DESIGN`, or `WRITE_BITSTREAM`.
- `PRE` executes the script before the specified implementation phase.
- `POST` executes the script after the specified implementation phase.

For example:

```
[vivado]
prop=run.impl_1.STEPS.OPT_DESIGN.TCL.PRE=<pathToTclScript>
prop=run.impl_1.STEPS.OPT_DESIGN.TCL.POST=<pathToTclScript>
prop=run.impl_1.STEPS.PLACE_DESIGN.TCL.PRE=<pathToTclScript>
prop=run.impl_1.STEPS.PLACE_DESIGN.TCL.POST=<pathToTclScript>
prop=run.impl_1.STEPS.PHYS_OPT_DESIGN.TCL.PRE=<pathToTclScript>
prop=run.impl_1.STEPS.PHYS_OPT_DESIGN.TCL.POST=<pathToTclScript>
prop=run.impl_1.STEPS.ROUTE_DESIGN.TCL.PRE=<pathToTclScript>
prop=run.impl_1.STEPS.ROUTE_DESIGN.TCL.POST=<pathToTclScript>
```

To specify the Pre and Post Tcl scripts as a v++ parameter, use the `--vivado.prop run.impl_1.STEP.<PHASE>.TCL.<PRE|POST>=<pathToTclScript>` command line option. For example, to specify a Tcl script to be executed before `opt_design`:

```
--vivado.prop run.impl_1.STEP.OPT_DESIGN.TCL.PRE=<pathToTclScript>
```

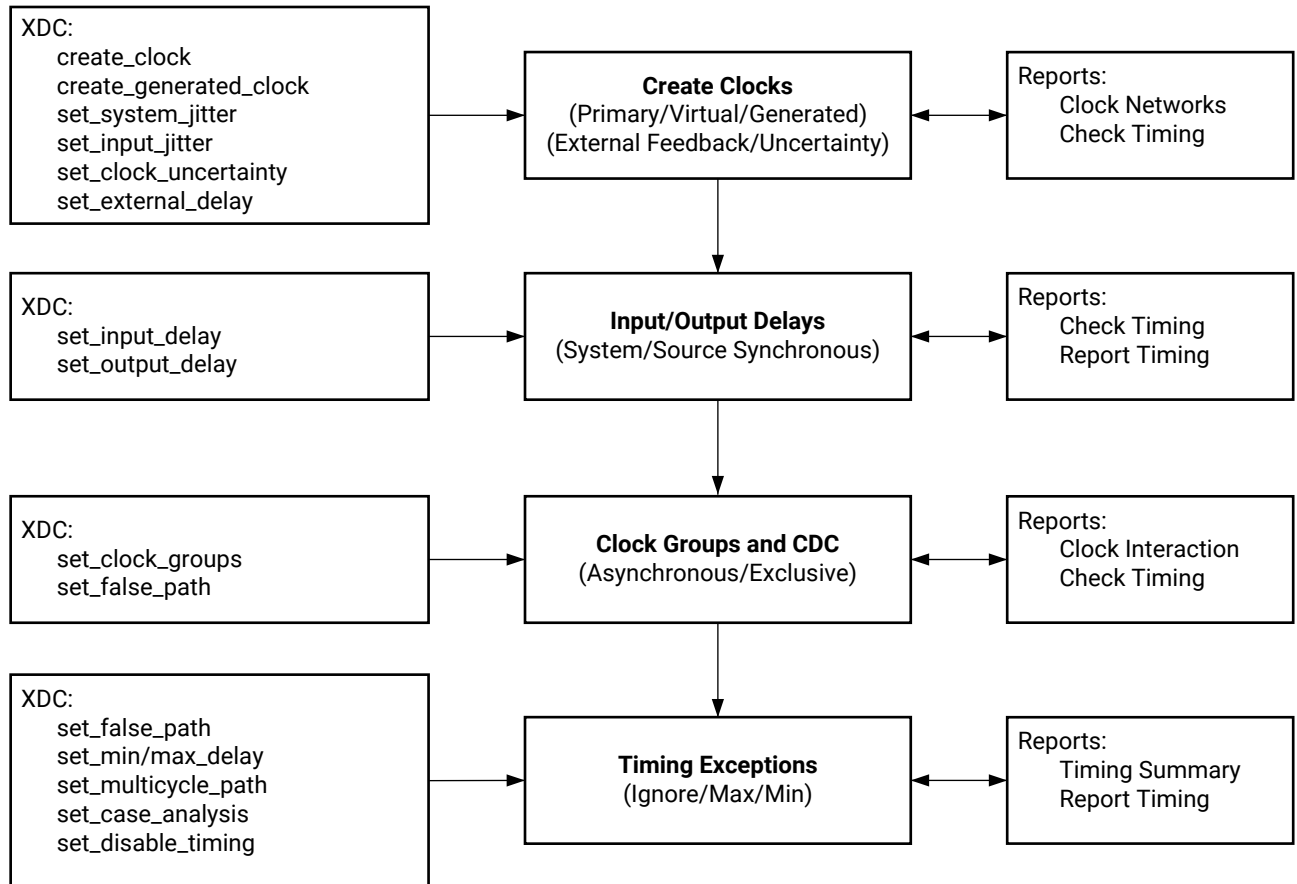
Where:

- `--vivado` is the v++ command line option to specify directives for the Vivado tools.
- `prop` indicates a property setting.
- `run.` indicates a run property.
- `impl_1.` indicates the name of the run.
- `STEP.OPT_DESIGN.TCL.PRE` indicates the run property you are specifying.
- `<pathToTclScript>` indicates the property value.

Defining Timing Constraints in Four Steps

The process of defining good constraints is broken into the four major steps shown in the following figure. The steps follow the timing constraints precedence and dependency rules, as well as the logical way of providing information to the timing engine to perform the analysis.

Figure 86: Steps for Developing Timing Constraints



X13445-122019

- The first two steps refer to the timing assertions where the default timing path requirements are derived from the clock waveforms and I/O delay constraints.
- During the third step, relationships between the asynchronous/exclusive clock domains that share at least one logical path are reviewed. Based on the nature of the relationships, clock groups or false path constraints are entered to ignore the timing analysis on these paths.
- The last step corresponds to the timing exceptions, where the designer can decide to alter the default timing path requirements by ignoring, relaxing or tightening them with specific constraints.

Constraints creation is associated with constraints identification and constraints validation tasks that are only possible with the various reports generated by the timing engine. The timing engine only works with a fully mapped netlist, for example, after synthesis. While it is possible to enter constraints with an elaborated netlist, it is recommended to create the first set of constraints with the post-synthesis netlist so that analysis and reports on the constraints can be performed interactively.

When creating timing constraints for a new design or completing existing constraints, Xilinx recommends using the Timing Constraints Wizard to quickly identify missing constraints for the first three steps in the previous figure. The Timing Constraints Wizard follows the methodology described in this section to ensure the design constraints are safe and reliable for proper timing closure. You can find more information on the Timing Constraints Wizard in *Vivado Design Suite User Guide: Using Constraints (UG903)*.

The following sections describe in detail the four steps described above:

- [Defining Clock Constraints](#)
- [Constraining Input and Output Ports](#)
- [Defining Clock Groups and CDC Constraints](#)
- [Specifying Timing Exceptions](#)

Refer to each section for a detailed methodology and use case when you are at the appropriate step in the constraint creation process.

Defining Clock Constraints

Clocks must be defined first so that they can be used by other constraints. The first step of the timing constraint creation flow is to identify where the clocks must be defined and whether they must be defined as a *primary clock* or a *generated clock*.



IMPORTANT! When defining a clock with a specific name (*-name* option), you must verify that the clock name is not already used by another clock constraint or an existing auto-generated clock. The Vivado Design Suite timing engine issues a message when a clock name is used in several clock constraints to warn you that the first clock definition is overridden. When the same clock name is used twice, the first clock definition is lost as well as all constraints referring to that name and entered between the two clock definitions. Xilinx recommends that you avoid overriding clock definitions unless no other constraints are impacted and all timing paths remain constrained.

Identifying Clock Sources

The unconstrained clock sources can be identified in the design by the Clock Networks report and the Check Timing report.

Clock Networks Report

Both constrained and unconstrained clock source points are listed in two separate categories. For each unconstrained source point, you must identify whether a primary clock or a generated clock must be defined.

```
% report_clock_networks
Unconstrained Clocks
Clock sysClk (endpoints: 15633 clock, 0 nonclock)
Port sysClk
Clock TXOUTCLK (endpoints: 148 clock, 0 nonclock)
GTXE2_CHANNEL/TXOUTCLK
(mgtEngine/ROCKETIO_WRAPPER_TILE_i/gt0_ROCKETIO_WRAPPER_TILE_i/gtxe2_i)
Clock Q (endpoints: 8 clock, 0 nonclock)
FDRE/Q (usbClkDiv2_reg)
```

Check Timing Report

The `no_clock` check reports the groups of active leaf clock pins with no clock definition. Each group is associated with a clock source point where a clock must be defined to clear the issue.

```
% check_timing -override_defaults no_clock
1. checking no_clock
-----
There are 15633 register/latch pins with no clock driven by root clock pin: sysClk
(HIGH)
There are 148 register/latch pins with no clock driven by root clock pin:
mgtEngine/ROCKETIO_WRAPPER_TILE_i/gt0_ROCKETIO_WRAPPER_TILE_i/gtxe2_i/TXOUTCLK
(HIGH)
There are 8 register/latch pins with no clock driven by root clock pin:
usbClkDiv2_reg/C (HIGH)
```

With `check_timing`, the same clock source pin or port can appear in several groups depending on the topology of the entire clock tree. In such case, creating a clock on the recommended source pin or port will resolve the missing clock definition for all the associated groups.

Related Information

[Checking That Your Design is Properly Constrained](#)

Creating Primary Clocks

A primary clock is a clock that defines a timing reference for your design and that is used by the timing engine to derive the timing path requirements and the phase relationship with other clocks. Their insertion delay is calculated from the clock source point (driver pin/port where the clock is defined) to the clock pins of the sequential cells to which it fans out.

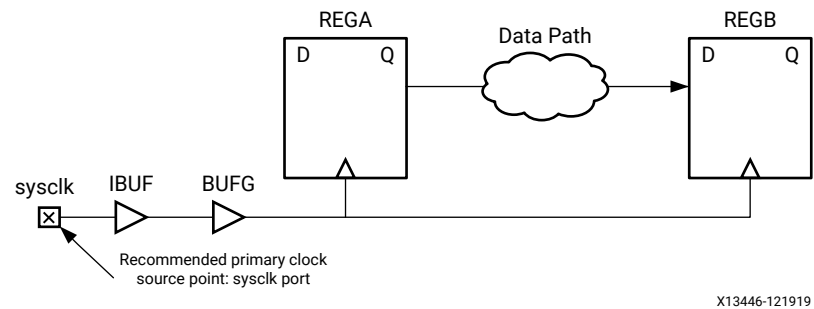
For this reason, it is important to define the primary clocks on objects that correspond to the boundary of the design, so that their delay, and indirectly their skew, can be accurately computed.

The following sections describe the typical primary clock roots.

Input Ports

You can use an input port as the primary clock root as shown in the following figure.

Figure 87: `create_clock` for Input Ports



Constraint example:

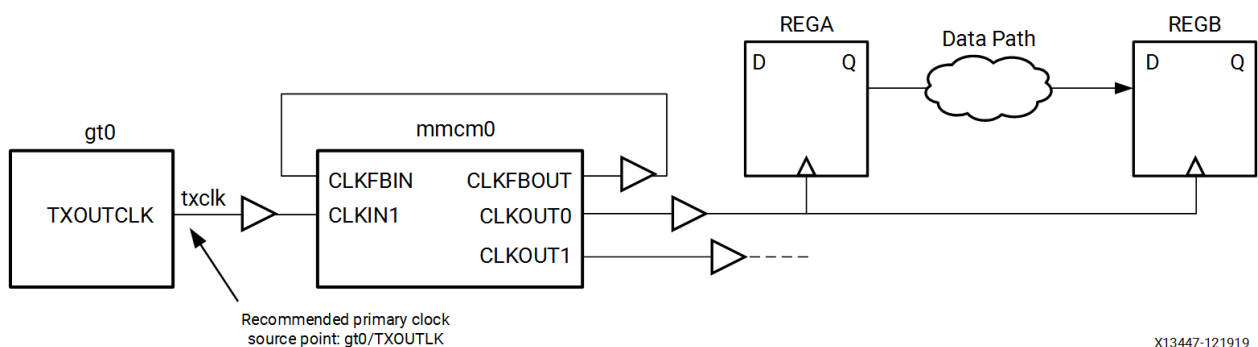
```
create_clock -name SysClk -period 10 -waveform {0 5} [get_ports sysclk]
```

In this example, the waveform is defined to have a 50% duty cycle. The `-waveform` argument is shown above to illustrate its usage and is only necessary to define a clock with a duty cycle other than 50%. For more information, see the `create_clock` Tcl command in the *Vivado Design Suite Tcl Command Reference Guide* (UG835). For a differential clock input buffer, the primary clock only needs to be defined on the P-side of the pair.

Gigabit Transceiver Output Pins in 7 Series Devices

You can use a gigabit transceiver output pin (e.g., a recovered clock) as the primary clock root as shown in the following figure.

Figure 88: `create_clock` on a Primitive Pin



Constraint example:

```
create_clock -name txclk -period 6.667 [get_pins gt0/TXOUTCLK]
```



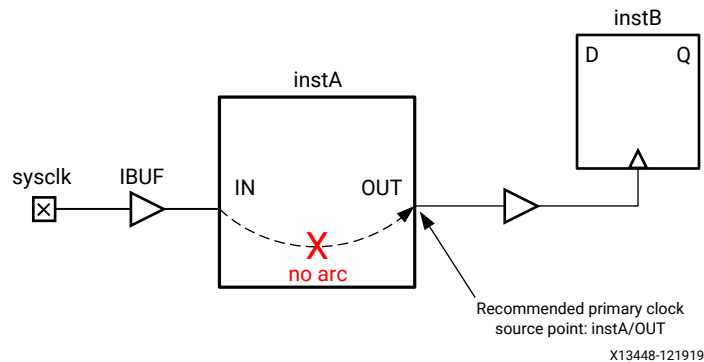
RECOMMENDED: For designs that target 7 series devices, Xilinx recommends also defining the GT incoming clocks, because the Vivado tools calculate the expected clocks on the GT output pins and compare these clocks with the user-created clocks. If the clocks differ or if the incoming clocks to the GT are missing, the tools issue a methodology check warning.

Note: For designs that target UltraScale™ and UltraScale+™ devices, Xilinx does not recommend defining a primary clock on the output of GTs, because GT clocks are automatically derived when the REFCLK input clocks are defined.

Certain Hardware Primitive Output Pins

You can use the output pin of certain hardware primitives as the primary clock root, such as the output pin shown in the following figure, which does not have a timing arc from an input pin of the same primitive.

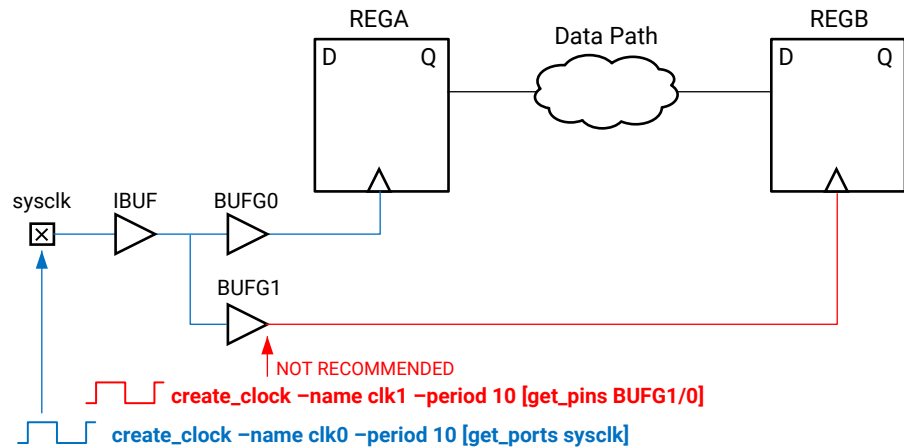
Figure 89: Clock Path Broken Due to a Missing Timing Arc



IMPORTANT! No primary clock should be defined in the transitive fanout of another primary clock because this situation does not correspond to any hardware reality. It will also prevent proper timing analysis by preventing the complete clock insertion delay calculation. Any time this situation occurs, the constraints must be revisited and corrected.

The following figure shows an example in which the clock `clk1` is defined in the transitive fanout of the clock `clk0`. The clock `clk1` overrides `clk0` starting at the output of `BUFG1`, where it is defined. Therefore, the timing analysis between `REGA` and `REGB` is not accurate because of the invalid skew computation between `clk0` and `clk1`.

Figure 90: create_clock in the Fanout of Another Clock is Not Recommended



X13449-121919

Creating Generated Clocks

A generated clock is a clock derived from another existing clock called the master clock. It usually describes a waveform transformation performed on the master clock by a logic block. Because the generated clock definition depends on the master clock characteristics, the master clock must be defined first. For explicitly defining a generated clock, the `create_generated_clock` command must be used.

Auto-Derived Clocks

Most generated clocks are automatically derived by the Vivado timing engine which recognizes the clock modifying blocks (CMB) and the transformation they perform on the master clocks.

In the Xilinx 7 series device family, the CMBs are:

- MMCM*/ PLL*
- BUFR
- PHASER*

In the Xilinx UltraScale device family, following are the CMBs:

- MMCM* / PLL*
- BUFG_GT / BUFGCE_DIV
- GT*_COMMON / GT*_CHANNEL / IBUFDS_GTE3
- BITSlice_CONTROL / RX*_BITSlice
- ISERDESE3

For any other combinatorial cell located on the clock tree, the timing clocks propagate through them and do not need to be redefined at their output, unless the waveform is transformed by the cell. In general, you must rely on the auto-derivation mechanism as much as possible as it provides the safest way to define the generated clocks that correspond to the actual hardware behavior.

If the auto-derived clock name chosen by the Vivado Design Suite timing engine does not seem appropriate, you can force your own name by using the `create_generated_clock` command without specifying the waveform transformation. This constraint should be located right after the constraint that defines the master clock in the constraint file. For example, if the default name of a clock generated by a MMCM instance is `net0`, you can add the following constraint to force your own name (`fftClk` in the given example):

```
create_generated_clock -name fftClk [get_pins mmcm_i/CLKOUT0]
```

To avoid any ambiguity, the constraint must be attached to the source pin of the clock. For more information, see *Vivado Design Suite User Guide: Using Constraints* (UG903).

User-Defined Generated Clocks

When all the primary clocks have been defined, you can use the Clock Networks or Check Timing (`no_clock`) reports to identify the clock tree portions that do not have a timing clock and define the generated clocks accordingly.

It is sometimes difficult to understand the transformation performed by a cone of logic on the master clock. In this case, you must adopt the most conservative constraint. For example, the source pin is a sequential cell output. The master clock is at least divided by two, so the proper constraint should be, for example:

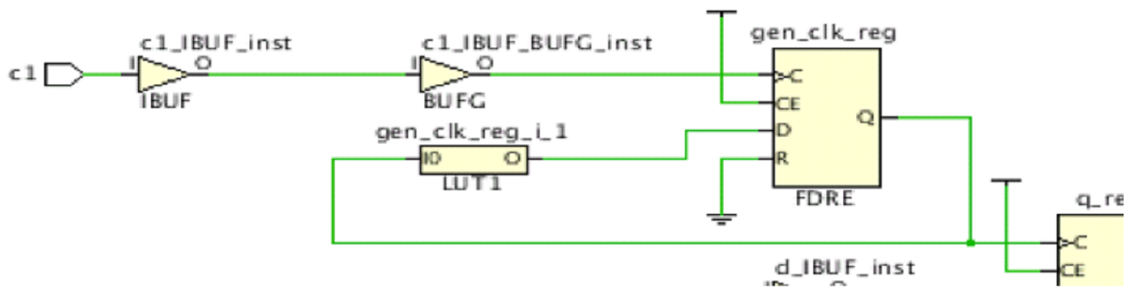
```
create_generated_clock -name clkDiv2 -divide_by 2 \
-source [get_pins fd/C] [get_pins fd/Q]
```

Finally, if the design contains latches, the latch gate pins also need to be reached by a timing clock and will be reported by Check Timing (`no_clock`) if the constraint is missing. You can follow the examples above to define these clocks.

Path Between Master and Generated Clocks

Unlike primary clocks, generated clocks must be defined in the transitive fanout of their master clock, so that the timing engine can accurately compute their insertion delay. Failure to follow this rule will result in improper timing analysis and most likely in invalid slack computation. For example, in the following figure `gen_clk_reg/Q` is being used as a clock for the next flop (`q_reg`), and it is also in the fanout cone of the primary clock `c1`. Hence `gen_clk_reg/Q` should have a `create_generated_clock` on it, rather than a `create_clock`.

Figure 91: Generated Clock in the Fanout Of Master Clock



```
create_generated_clock -name GC1 -source [get_pins gen_clk_reg/C] -divide_by 2
[get_pins gen_clk_reg/Q]
```

Verifying Clocks Definition and Coverage

When all design clocks are defined and applied in memory, you can verify the waveform of each clock, the relationship between master and generated clocks by using the `report_clocks` command:

```
Clock Period Waveform Attributes Sources
sysClk 10.00000 {0.00000 5.00000} P {sysClk}
clkfbout 10.00000 {0.00000 5.00000} P,G {clkgen/mmcm_adv_inst/CLKFBOUT}
cpuClk 20.00000 {0.00000 10.00000} P,G {clkgen/mmcm_adv_inst/CLKOUT0}
...
=====
Generated Clocks
=====
Generated Clock : cpuClk
Master Source : clkgen/mmcm_adv_inst/CLKIN1
Master Clock : sysClk
Edges : {1 2 3}
Edge Shifts : {0.000 5.000 10.000}
Generated Sources : {clkgen/mmcm_adv_inst/CLKOUT0}
```

You can also verify that all internal timing paths are covered by at least one clock. The Check Timing report provides two checks for that purpose:

- **no_clock:** Reports any active clock pin that is not reached by a defined clock.
- **unconstrained_internal_endpoint:** Reports all the data input pins of sequential cells that have a timing check relative to a clock but the clock has not been defined.

If both checks return zero, the timing analysis coverage will be high.

Alternatively, you can run the XDC and Timing Methodology checks to verify that all clocks are defined on recommended netlist objects without introducing any constraint conflict or inaccurate timing analysis scenario.

Use the following command to run these checks:

```
report_methodology -checks [get_methodology_checks {TIMING-* XDC*}]
```

Related Information

[Running Report Methodology](#)

Adjusting Clock Characteristics

After defining the clocks and their waveform, the next step is to enter any information related to noise or uncertainty modeling. The XDC language differentiates uncertainty related to jitter and phase error from the one related to skew and delay modeling.

Jitter

For jitter, it is best to use the default values used by the Vivado Design Suite. You can modify the default computation as follows:

- If a primary clock enters the device with a random jitter greater than zero, use the `set_input_jitter` command to specify the peak-to-peak jitter value in nanoseconds.
- To adjust the global jitter if the device power supply is noisy, use `set_system_jitter`. Xilinx does *not* recommend increasing the default system jitter value.

For generated clocks, the jitter is derived from the master clock and the characteristics of the clock modifying block. You do not need to adjust these numbers.

Additional Uncertainty

When you need to add extra margin on the timing paths of a clock or between two clocks, you must use the `set_clock_uncertainty` command. This is also the best and safest way to over-constrain a portion of a design without modifying the actual clock edges and the overall clocks relationships. The clock uncertainty defined by you is additive to the jitter computed by the Vivado tools, and can be specified separately for setup and hold analysis.

For example, the margin on all intra-clock paths of the design clock `clk0` needs to be tightened by 500 ps to make the design more robust to noise for both setup and hold:

```
set_clock_uncertainty -from clk0 -to clk0 0.500
```

Note: Tightening the hold margin on a design can lead to hold violations on dedicated intra-site and cascade paths that the router cannot fix by detouring the intra-site net.

If you specify additional uncertainty between two clocks, the constraint must be applied in both directions (assuming data flows in both directions). The example below shows how to increase the uncertainty by 250 ps between `clk0` and `clk1` for setup only:

```
set_clock_uncertainty -from clk0 -to clk1 0.250 -setup
set_clock_uncertainty -from clk1 -to clk0 0.250 -setup
```

Clock Latency at the Source

It is possible to model the latency of a clock at its source by using the `set_clock_latency` command with the `-source` option. This is useful in two cases:

- To specify the clock delay propagation outside the device independently from the input and output delay constraints.
- To model the internal propagation latency of a clock used by a block during out-of-context compilation. In such a compilation flow, the complete clock tree is not described, so the variation between min and max operating conditions outside the block cannot be automatically computed and must be manually modeled.


This constraint should only be used by advanced users as it is usually difficult to provide valid latency values.

MMCM or PLL External Feedback Loop Delay

When the MMCM or PLL feedback loop is connected for compensating a board delay instead of an internal clock insertion delay, you must specify the delay outside the device for both best and worst delay cases by using the `set_external_delay` command. Failure to specify this delay makes I/O timing analysis associated with the MMCM or PLL irrelevant and can potentially lead to an impossible timing closure situation. Also, when using external compensation, you must adjust the input and output delay constraint values accordingly instead of just considering the clock trace delay on the board like in normal cases.

Constraining Input and Output Ports

In addition to specifying the location and I/O standard for each port of the design, input and output delay constraints must be specified to describe the timing of external paths to/from the interface of the device. These delays are defined relative to a clock that is usually also generated on the board and enters the device. In some cases, the delays must be defined related to a virtual clock when the I/O path is related to a clock that has a waveform different from the board clock.

 **IMPORTANT!** I/O delays can only be constrained for interfaces using I/O logic, such as ISERDES/OSERDES/IDDR/ODDR/IOB registers or fabric. For guidance on component mode timing, see *Designing Using SelectIO Interface Component Primitives (XAPP1324)*. For high-speed I/O interfaces created using UltraScale device SelectIO native mode, see [Xilinx Answer Record 68618](#).

System Level Perspective

The I/O paths are modeled like register-to-register paths by the Vivado Design Suite timing engine, except that you must define a constraint to model the part of the path delay located outside the device. When analyzing internal paths, minimum and maximum delays are considered for both setup and hold analysis. This is also true for I/O paths. For this reason, it is important to describe both min and max delay conditions. The I/O timing paths are analyzed as single-cycle paths by default, which means the following:

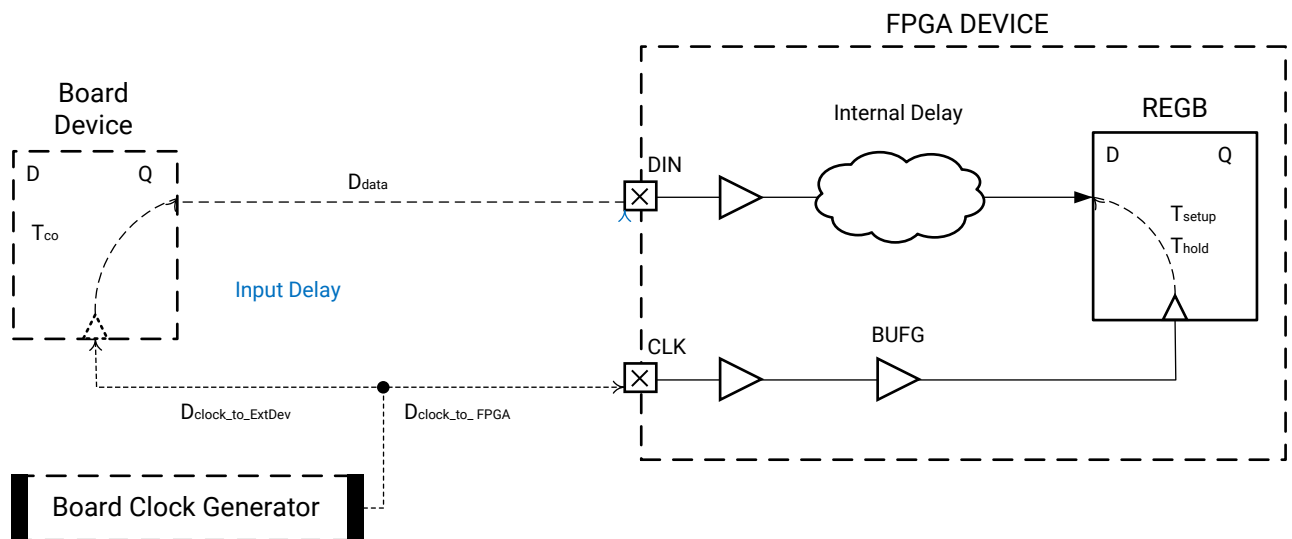
- For max delay analysis (setup), the data is captured one clock cycle after the launch edge for single data rate interface, and half clock cycle after the launch edge for a double data rate interface.
- For min delay analysis (hold), the data is launched and captured by the same clock edge.

If the relationship between the clock and I/O data must be timed differently, like for example in a source synchronous interface, different I/O delays and additional timing exceptions must be specified. This corresponds to an advanced I/O timing constraints scenario.

Defining Input Delays

The input delay is defined relative to a clock at the interface of the device. Unless `set_clock_latency` has been specified on the source pin of the reference clock, the input delay corresponds to the absolute time from the launch edge, through the clock trace, the external device and the data trace. If clock latency has already been specified separately, you can ignore the clock trace delay.

Figure 92: Input Delay Computation



X13450-121919

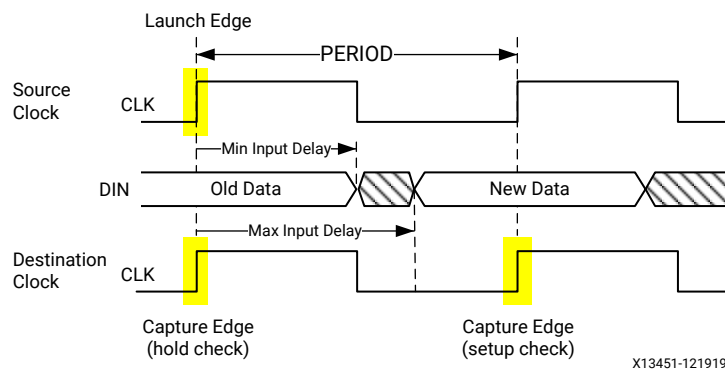
The input delay values for the both types of analysis are:

```
Input Delay(max) = Tco(max) + Ddata(max) + Dclock_to_ExtDev(max) - Dclock_to_FPGA(min)
Input Delay(min) = Tco(min) + Ddata(min) + Dclock_to_ExtDev(min) - Dclock_to_FPGA(max)
```

The following figure shows a simple example of input delay constraints for both setup (max) and hold (min) analysis, assuming the `sysClk` clock has already been defined on the `CLK` port:

```
set_input_delay -max -clock sysClk 5.4 [get_ports DIN]
set_input_delay -min -clock sysClk 2.1 [get_ports DIN]
```

Figure 93: Interpreting Min and Max Input Delays

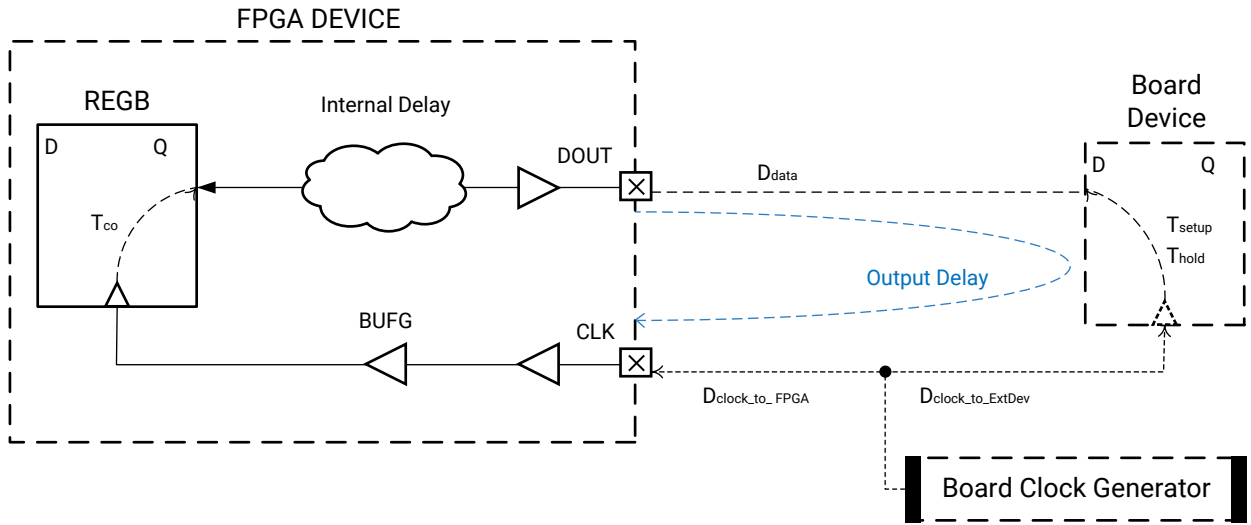


A negative input delay means that the data arrives at the interface of the device before the launch clock edge.

Defining Output Delays

Output delays are similar to input delays, except that they refer to the output path minimum and maximum time outside the device to be functional under all conditions.

Figure 94: Output Delay Computation



X23060-111419

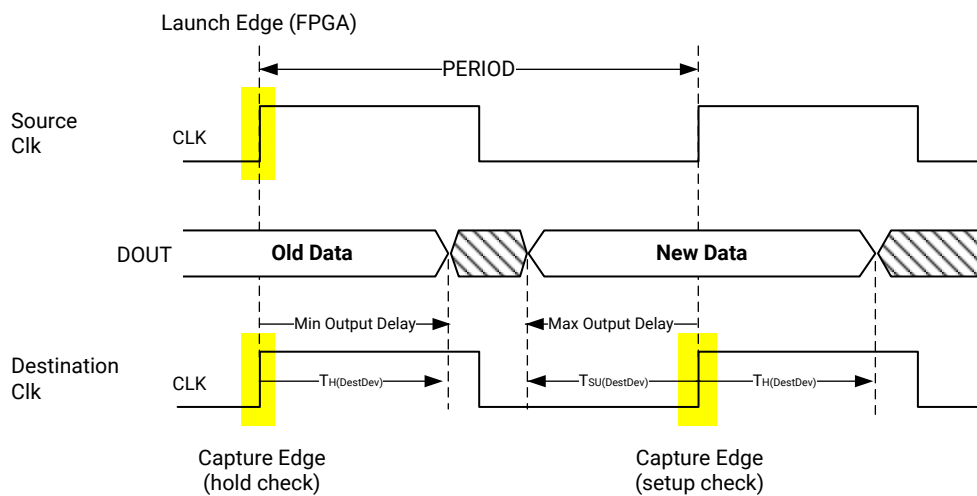
The output delay values for the both types of analysis are:

$$\begin{aligned} \text{Output Delay(max)} &= T_{\text{setup}} + D_{\text{data(max)}} + D_{\text{clock_to_FPGA(max)}} - D_{\text{clock_to_ExtDev(min)}} \\ \text{Output Delay(min)} &= D_{\text{data(min)}} - T_{\text{hold}} + D_{\text{clock_to_FPGA(min)}} - D_{\text{clock_to_ExtDev(max)}} \end{aligned}$$

The following figure shows a simple example of output delay constraints for both setup (max) and hold (min) analysis, assuming the sysClk clock has already been defined on the CLK port:

```
set_output_delay -max -clock sysClk 2.4 [get_ports DOUT]
set_output_delay -min -clock sysClk -1.1 [get_ports DOUT]
```

Figure 95: Interpreting Min and Max Output Delays



X13453-121919

The output delay corresponds to the delay on the board before the capture edge. For a regular system synchronous interface where the clock and data board traces are balanced, the setup time of the destination device defines the output delay value for max analysis. And the destination device hold time defines the output delay for min analysis. The specified min output delay indicates the minimum delay that the signal will incur after coming out of the design, before it will be used for hold analysis at the destination device interface. Thus, the delay inside the block can be that much smaller. A positive value for min output delay means that the signal can have negative delay inside the design. This is why min output delay is often negative. For example, the following code example indicates that the delay inside the design until DOUT has to be at least +0.5 ns to meet the hold time requirement.

```
set_output_delay -min -0.5 -clock CLK [get_ports DOUT]
```

Choosing the Reference Clock

Depending on the clock tree topology that controls the sequential cells related to input or output ports, you have to choose the most appropriate clock to define the input or output delay constraints. If the clock of the I/O path register is a generated clock, the delay constraint usually needs to be defined relative to the primary clock, which is defined upstream of the generated clocks. There are some exceptions to this rule that are explained in this section.

Identifying the Clocks Related to Each Port

Before defining the I/O delay constraint, you must identify which clocks are related to each port. You can identify the clocks using the methods described in the following sections.

Browse the Board Schematics

For a group of I/O ports connected to another device interface on the board, you can use the board clock that is connected to both the Xilinx device and to the external device interface as the reference clock for the input or output delay constraints. To control the timing of the related group of ports, you must verify in the external device data sheet that the board clock is internally transformed for timing the I/O ports, which ensures that the design generates the same clock inside the Xilinx device.

Browse the Design Schematics

For each port, you can expand the path schematics to the first level of sequential cells, and then trace the clock pins of those cells back to the clock source(s). This approach can be impractical for ports that are connected to high fanout nets.

Report Timing from or to the Port

Whether a port is already constrained or not, you can use the `report_timing` command to identify its related clocks in the design. Once all the timing clocks have been defined, you can report the worst path from or to the I/O port, create the I/O delay constraint relative to the clock reported, and rerun the same timing report from/to the other clocks of the design. If it appears that the port is related to more than one clock, create the corresponding constraint and repeat the process.

For example, the `din` input port is related to the clocks `clk1` and `clk2` inside the design:

```
report_timing -from [get_ports din] -sort_by group
```

The report shows that the `din` port is related to `clk1`. The input delay constraint is (for both min and max delay in this example):

```
set_input_delay -clock clk1 5 [get_ports din]
```

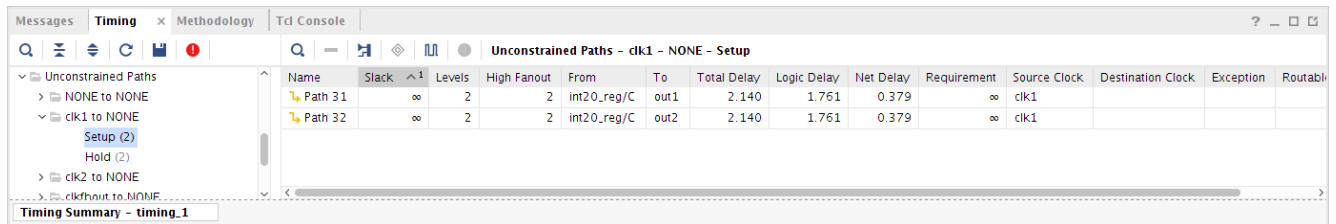
Rerun timing analysis with the same command as previously, and observe that `din` is also related to `clk2` due to the `-sort_by group` option, which reports N paths per endpoint clock. You can add the corresponding delay constraint and rerun the report to validate that the `din` port is not related to another clock.

You can also run the same analysis using the Timing Summary report with the `-report_unconstrained` option. With only clock constraints in your design, the Unconstrained Paths section appears as follows:

```
-----
| Unconstrained Path Table
|-----
Path Group      From Clock      To Clock
-----
(none)
(none)          clk1
(none)          clk2
(none)                  clk1
(none)                  clk2
```

The fields without a clock name (or <NONE> in the Vivado IDE) refer to a group of paths where the startpoints (From Clock) or the endpoints (To Clock) are not associated with a clock. The unconstrained I/O ports fall in this category. You can retrieve their name by browsing the rest of the report. For example in the Vivado IDE, by selecting the Setup paths for the `clk1` to NONE category, you can see the ports driven by `clk1` in the To column:

Figure 96: Getting a List of Unconstrained Output Ports



Name	Slack	Levels	High Fanout	From	To	Total Delay	Logic Delay	Net Delay	Requirement	Source Clock	Destination Clock	Exception	Routabl
Path 31	∞	2	2	int20_reg/C	out1	2.140	1.761	0.379	∞	clk1			
Path 32	∞	2	2	int20_reg/C	out2	2.140	1.761	0.379	∞	clk1			

After adding the new constraints and applying them in memory, you must rerun the report to determine which ports are still unconstrained. For most designs, you must increase the number of reported paths to make sure all the I/O paths are listed in the report.

Use Automatically Identified Sampling Clocks

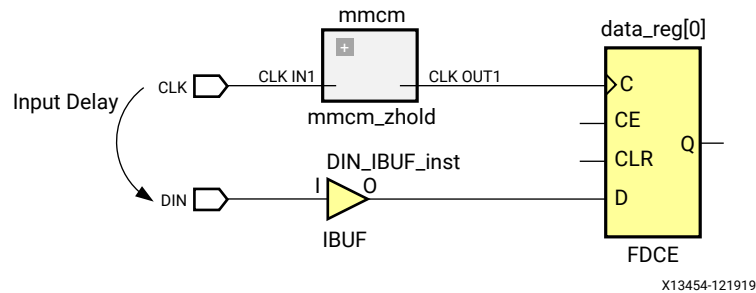
You can use the `set_input_delay` and `set_output_delay` constraints without specifying the related clock. The Vivado Design Suite timing engine will analyze the design and associate each port with all the sampling clocks automatically. Then by reporting timing on the I/O paths, you can see how the tool constrained each I/O port. This is convenient for quickly constraining a design, but this type of generic constraints can become a problem if they are too generic and do not model the hardware reality accurately.

Using a Primary Clock

A primary clock (that is, an incoming board clock) should be used when it directly controls the I/O path sequential cells, without traversing any clock modifying block. I/O delay lines are not considered as clock modifying blocks because they only affect the clock insertion delay and not the waveform. This case is illustrated by the two examples provided in [Defining Input Delays](#) and [Defining Output Delays](#). Most of the time, the external device also has its interface characteristics defined with respect to the same board clock.

When the primary clock is compensated by a PLL or MMCM inside the device with the zero hold violation (ZHOLD) mode, the I/O paths sequential cells are connected to an internal copy (for example, a generated clock) of the primary clock. Because the waveforms of both clocks are identical, Xilinx recommends using the primary clock as the reference clock for the input/output delay constraints.

Figure 97: Input Delay in the Presence of a ZHOLD MMCM in Clock Path



The constraints are identical to the example provided in [Defining Input Delays](#) because the ZHOLD MMCM acts like a clock buffer with a negative insertion delay, which corresponds to the amount of compensation.

Using a Virtual Clock

When the board clock traverses a clock modifying block which transforms the waveform in addition to compensating the overall insertion delay, it is recommended to use a virtual clock as a reference clock for the input and output delay instead of the board clock. There are three main cases for using a virtual clock:

- The internal clock and the board clock have different period: The virtual clock must be defined with the same period and waveform as the internal clock. This results in a regular single-cycle path requirement on the I/O paths.
- For input paths, the internal clock has a positive shifted waveform compared to the board clock: the virtual clock is defined like the board clock, and a multicycle path constraint of two cycles for setup is defined from the virtual clock to the internal clock. These constraints force the setup timing analysis to be performed with a requirement of one clock cycle + amount of phase shift.
- For output paths, the internal clock has a negative shifted waveform compared to the board clock: the virtual clock is defined like the board clock and a multicycle path constraint of two cycles for setup is defined from the internal clock to the virtual clock. These constraints force the setup timing analysis to be performed with a requirement of one clock cycle + amount of phase shift.

To summarize, the use of a virtual clock adjusts the default timing analysis to avoid treating I/O paths as clock domain crossing paths with a tight and unrealistic requirement.

★ IMPORTANT! You only need to use the multicycle path for I/O paths with phase-shifted clocks when the phase-shift results in modification of the clock waveform. When the phase shift is added to the insertion delay of the clock modifying block and the clock waveform is preserved, you do not need to use a multicycle path. For more information, see this [link](#) in the Vivado Design Suite User Guide: Design Analysis and Closure Techniques (UG906).

For example, consider the `sysClk` board clock that runs at 100 MHz and gets multiplied by an MMCM to generate `clk266` that runs at 266 MHz. An output that is generated by `clk266` should use `clk266` as the reference clock. If you try to use `sysClk` as the reference clock (for the `set_output_delay` specification), it will appear as asynchronous clocks, and the path can no longer be timed as a single cycle.

Using a Generated Clock

For an output source synchronous interface, the design generates a copy of the internal clock and forwards it to the board along with the data. This clock is usually used as the reference clock for the output data delay constraints whenever the intent is to control and report on the phase relationship (skew) between the forwarded clock and the data. The forwarded clock can also be used in input and output delay constraints for a system synchronous interface.

Rising and Falling Reference Clock Edges

The clock edges used in the I/O constraint must reflect the data sheet of the external device connected to the device. By default, the `set_input_delay` and `set_output_delay` commands define a delay constraint relative to the rising reference clock edge. You must use the `clock_fall` option to specify a delay relative the falling clock edge. You can also specify separate constraints for delays related to both rising and falling clock edges by using the `add_delay` option with the second constraint on a port.

In most cases, the I/O reference clock edges correspond to the clock edges used to latch or launch the I/O data inside the device. By analyzing the I/O timing paths, you can review which clock edges are used and verify that they correspond to the actual hardware behavior. If by mistake a rising clock edge is used as a reference clock for an I/O path that is only related to the falling clock edge internally, the path requirement is $\frac{1}{2}$ -period, which makes timing closure more difficult.

Verifying Delay Constraints

Once the I/O timing constraints have been entered, it is important to review how timing is analyzed on the I/O paths and the amount of slack violation for both setup and hold checks. By using the timing reports from/to all ports for both setup and hold analysis (that is, `delay type = min_max`), you can verify that:

- The correct clocks and clock edges are used as reference for the delay constraints.
- The expected clocks are launching and capturing the I/O data inside the device.
- The violations can reasonably be fixed by placement or by setting the proper delay line tap configuration. If this is not the case, you must review the I/O delay values entered in the constraints and evaluate whether they are realistic, and whether you must modify the design to meet timing.

I/O Path Report Command Lines Example

```
report_timing -from [all_inputs] -nworst 1000 -sort_by group \  
-delay_type min_max
```

```
report_timing -to [all_outputs] -nworst 1000 -sort_by group \  
-delay_type min_max
```

Improper I/O delay constraints can lead to impossible timing closure. The implementation tools are timing driven and work on optimizing the placement and routing to meet timing. If the I/O path requirements cannot be met and I/O paths have the worst violations in the design, the overall design QoR will be impacted.

Input to Output Feed-through Path

There are several equivalent ways to constrain a combinatorial path from an input port to an output port.

Example One

Use a virtual clock with a period greater or equal to the target maximum delay for the feed-through path, and apply max input and output delay constraints as follows:

```
create_clock -name vclk -period 10  
set_input_delay -clock vclk <input_delay_val> [get_ports din] -max  
set_output_delay -clock vclk <output_delay_val> [get_ports dout] -max
```

where

```
input_delay_val(max) + feedthrough path delay (max) + output_delay_val(max)  
<= vclk period.
```

In this example, only the maximum delay is constrained.

Example Two

Use a combination of min and max delay constraints between the feedthrough ports. Example:

```
set_max_delay -from [get_ports din] -to [get_ports dout] 10  
set_min_delay -from [get_ports din] -to [get_ports dout] 2
```

This is a simple way to constrain both minimum and maximum delays on the path. Any existing input and output delay constraints on the same ports are also used during the timing analysis. For this reason, this style is not very popular.

The max delay is usually optimized and reported against the Slow timing corner, while the min delay is in the Fast timing corner. It is best to run a few iterations on the feedthrough path delay constraints to validate that they are reasonable and can be met by the implementation tools, especially if the ports are placed far from one another.

Using XDC Templates - Source Synchronous Interfaces

Xilinx recommends using I/O constraint templates for the source synchronous interfaces. The source synchronous constraints can be written in several ways. The templates provided by the Vivado Design Suite are based on the default timing analysis path requirement. The syntax is simpler, but the delay values must be adjusted to account for the fact that the setup analysis is performed with different launch and capture edges (1-cycle or 1/2-cycle) instead of same edge (0-cycle). The timing reports can be more difficult to read as the clock edges do not directly correspond to the active ones in hardware. You can navigate to these templates in the Vivado IDE through **Tools** → **Language Templates** → **XDC** → **Timing Constraints** → **Input Delay Constraints** → **Source Synchronous**.

Defining Clock Groups and CDC Constraints

The Vivado IDE times the paths between all the clocks in your design by default. You can use the following constraints to modify this default behavior:

- `set_clock_groups`: Disables timing analysis between groups of clocks that you identify but not between the clocks within a same group.
- `set_false_path`: Disables timing analysis between the clocks only in the direction specified by the `-from` and `-to` options.

In some cases, you might want to use the following constraints on one or more paths of the clock domain crossing (CDC) to limit latency or bus skew:

- `set_max_delay -datapath_only`: Sets the maximum delay constraints on asynchronous CDC paths to limit the latency.

Note: If clock groups or false path constraints already exist between the clocks or on the same CDC paths, the maximum delay constraints will be ignored. Therefore, it is important to thoroughly review every path between all clock pairs before choosing one CDC timing constraint over another to avoid constraints collision.



RECOMMENDED: Xilinx also recommends running `report_methodology` to identify when a `set_max_delay -datapath_only` constraint is overridden by a `set_clock_groups` or `set_false_path` constraint.

- `set_bus_skew`: Constrains a set of signals between asynchronous CDC paths by bus skew instead of latency.



TIP: You can also set a bus skew constraint from the Vivado IDE. In the Timing Constraints window, expand **Assertions**, and double-click **Set Bus Skew**.

Related Information

[Running Report Methodology](#)

Reviewing Clock Interactions

Clocks that have a logical path between them are timed. The possible clock relationships are synchronous, asynchronous, and exclusive.

Synchronous

Clock relationships are synchronous when two clocks have a fixed phase relationship. This is the case when two clocks share the following:

- Common circuitry (common node)
- Primary clock (same initial phase)

Asynchronous

Clock relationships are asynchronous when the clocks do not have a fixed phase relationship. This is the case when one of the following is true for the clocks:

- Do not share any common circuitry in the design and do not have a common primary clock.
- Do not have a common period within 1000 cycles (unexpandable) and the timing engine cannot properly time them together.
- Have a common clock but do not share a common node.
- Are part of a topology that does not ensure a known phase relationship through the clocks auto-derivation process.

If two clocks are synchronous but their common period is very small, the setup paths requirement is too tight for timing to be met. Xilinx recommends that you treat the two clocks as asynchronous and implement safe asynchronous CDC circuitry.

Exclusive

Clock relationships are exclusive when they propagate on a same clock tree and reach the same sequential cell clock pins but cannot physically be active at the same time.

Categorizing Clock Pairs

The clock pairs can be categorized by using the Clock Interaction and Check Timing reports.

Clock Interaction Report

The Clock Interaction report provides a high-level summary of how two clocks are timed together:

- Do the two clocks have a common primary clock? When clocks are properly defined, all clocks that originate from the same source in the design share the same primary clock.
- Do the two clocks have a common period? This shows in the setup or hold path requirement column (unexpandable), when the timing engine cannot determine the most pessimistic setup or hold relationship.
- Are the paths between the two clocks partially or completely covered by clock groups or timing exception constraints?
- Is the setup path requirement between the two clocks very tight? This can happen, when two clocks are synchronous, but their period is not specified as an exact multiple (for example, due to rounding off). Over multiple clock cycles, the edges could drift apart, causing the worst case timing requirement to be very tight.

Check Timing Report

The Check Timing report (`multiple_clock`) identifies the clock pins that are reached by more than one clock and a `set_clock_groups` or `set_false_path` constraint has not already been defined between these clocks.

Constraining Exclusive Clock Groups

You can use the regular timing or clock network reports to review the clock paths and identify the situations where two clocks propagate on a same clock tree and are used at the same time in a timing path where the startpoint and endpoint clock pins are connected to the same clock tree. This analysis can be a time consuming task. Instead, you can review the `multiple_clock` section of the Check Timing report. This section returns a list of clock pins and their associated timing clocks.

Based on the clock tree topology, you must apply different constraints as described in the following sections.

Overlapping Clocks Defined on the Same Clock Source

This occurs when two clocks are defined on the same netlist object with the `create_clock -add` command and represent the multiple modes of an application. In this case, it is safe to apply a clock groups constraint between the clocks. For example:

```
create_clock -name clk_mode0 -period 10 [get_ports clk_in]
create_clock -name clk_mode1 -period 13.334 -add [get_ports clk_in]
set_clock_groups -physically_exclusive -group clk_mode0 -group clk_mode1
```

If the `clk_mode0` and `clk_mode1` clocks generate other clocks, the same constraint needs to be applied to their generated clocks as well, which can be done as follows:

```
set_clock_groups -physically_exclusive \
  -group [get_clocks -include_generated_clock clk_mode0] \
  -group [get_clocks -include_generated_clock clk_mode1]
```

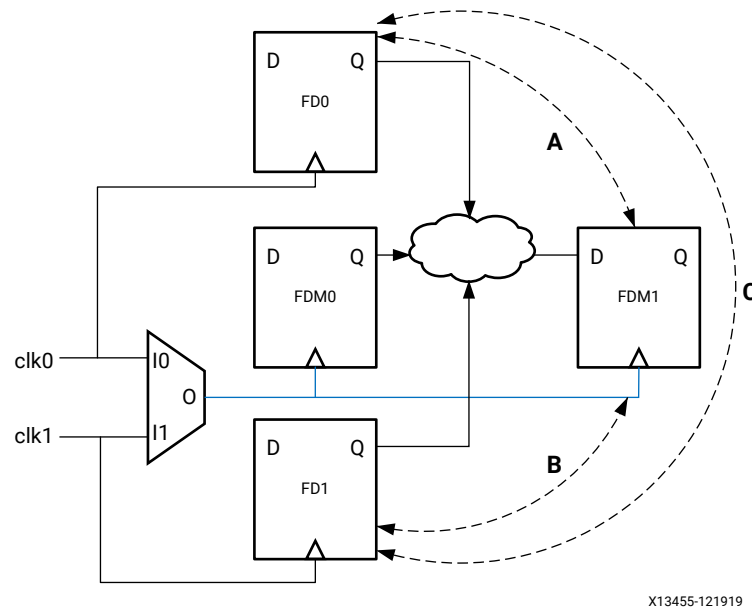
Overlapping Clocks Driven by a Clock Multiplexer

When two or more clocks drive into a multiplexer (or more generally a combinatorial cell), they all propagate through and become overlapped on the fanout of the cell. Realistically, only one clock can propagate at a time, but timing analysis allows reporting several timing modes at the same time.

For this reason, you must review the CDC paths and add new constraints to ignore some of the clock relationships. The correct constraints are dictated by how and where the clocks interact in the design.

The following figure shows an example of two clocks driving into a multiplexer and the possible interactions between them before and after the multiplexer.

Figure 98: Multiplexed Clocks



- Case in which the paths A, B, and C do not exist

`clk0` and `clk1` only interact in the fanout of the multiplexer (FDM0 and FDM1). It is safe to apply the clock groups constraint to `clk0` and `clk1` directly.

```
set_clock_groups -logically_exclusive -group clk0 -group clk1
```

- Case in which only the paths A or B or C exist

`clk0` and/or `clk1` directly interact with the multiplexed clock. To keep timing paths A, B, and C, the constraint cannot be applied to `clk0` and `clk1` directly. Instead, it must be applied to the portion of the clocks in the fanout of the multiplexer, which requires additional clock definitions.

```
create_generated_clock -name clk0mux -divide_by 1 \
-source [get_pins mux/I0] [get_pins mux/O]

create_generated_clock -name clk1mux -divide_by 1 \
-add -master_clock clk1 \
-source [get_pins mux/I1] [get_pins mux/O]

set_clock_groups -physically_exclusive -group clk0mux -group clk1mux
```

Constraining Asynchronous Clock Groups and Clock Domain Crossings

The asynchronous relationship can be quickly identified in the Clock Interaction report: clock pairs with no common primary clock or no common period (unexpanded). Even if clock periods are the same, the clocks will still be asynchronous, if they are being generated from different sources. The asynchronous Clock Domain Crossing (CDC) paths must be reviewed carefully to ensure that they use proper synchronization circuitry that does not rely on timing correctness and that minimizes the chance for metastability to occur. Asynchronous CDC paths usually have high skew and/or unrealistic path requirements. They should not be timed with the default timing analysis, which cannot prove they will be functional in hardware.

Report CDC

The Report CDC (`report_cdc`) command performs a structural analysis of the clock domain crossings in your design. You can use this information to identify potentially unsafe CDCs that might cause metastability or data coherency issues. Report CDC is similar to the Clock Interaction report, but Report CDC focuses on structures and related timing constraints. Report CDC does not provide timing information because timing slack does not make sense on paths that cross asynchronous clock domains.

Report CDC identifies the most common CDC topologies as follows:

- Single bit synchronizers
- Multi-bit synchronizers for buses
- Asynchronous reset synchronizers
- MUX and CE controlled circuitry
- Combinatorial logic before synchronizer
- Multi-clock fanin to synchronizer
- Fanout to destination clock domain

For more information on the `report_cdc` command, see this [link](#) in the *Vivado Design Suite User Guide: Design Analysis and Closure Techniques (UG906)*. Also, see `report_cdc` in the *Vivado Design Suite Tcl Command Reference Guide (UG835)*.

Specific constraints should be applied to prevent default timing analysis on asynchronous clock domain crossings.

Related Information

[Global Constraints Between Clocks in Both Directions](#)
[Constraints on Individual CDC Paths](#)

Global Constraints Between Clocks in Both Directions

When there is no need to limit the maximum latency, the clock groups can be used. Following is an example to ignore paths between `clkA` and `clkB`:

```
set_clock_groups -asynchronous -group clkA -group clkB
```

When two master clocks and their respective generated clocks form two asynchronous domains between which all the paths are properly synchronized, the clock groups constraint can be applied to several clocks at once:

```
set_clock_groups -asynchronous \  
-group {clkA clkA_gen0 clkA_gen1 } \  
-group {clkB clkB_gen0 clkB_gen1 }
```

Or simply:

```
set_clock_groups -asynchronous \  
-group [get_clocks -include_generated_clock clkA] \  
-group [get_clocks -include_generated_clock clkB]
```

Constraints on Individual CDC Paths

If a CDC bus uses gray-coding (for example, FIFO) or if latency needs to be limited between the two asynchronous clocks on one or more signals, you must use the `set_max_delay` constraint with the option `-datapath_only` to ignore clock skew and jitter on these paths, plus override the default path requirement by the latency requirement. It is usually sufficient to use the source clock period for the max delay value, just to ensure that no more than one data is present on the CDC path at any given time.

When the ratio between clock periods is high, choosing the minimum of the source and destination clock periods is also a good option to reduce the transfer latency. A clean asynchronous CDC path should not have any logic between the source and destination sequential cells, so the Max Delay Datapath Only constraint is normally easy to meet for the implementation tools.

Some asynchronous CDC paths require a skew control between the bits of the bus instead of a constraint on the bus latency. Using a bus skew constraint prevents the receiving clock domain from latching multiple states of the bus on the same clock edge. You can set the bus skew constraint on the bus with `set_bus_skew` command. For example, you can apply `set_bus_skew` to a CDC bus that uses gray-coding instead of using the Max Delay Datapath Only constraint. For more information, see this [link](#) in the *Vivado Design Suite User Guide: Using Constraints (UG903)*.

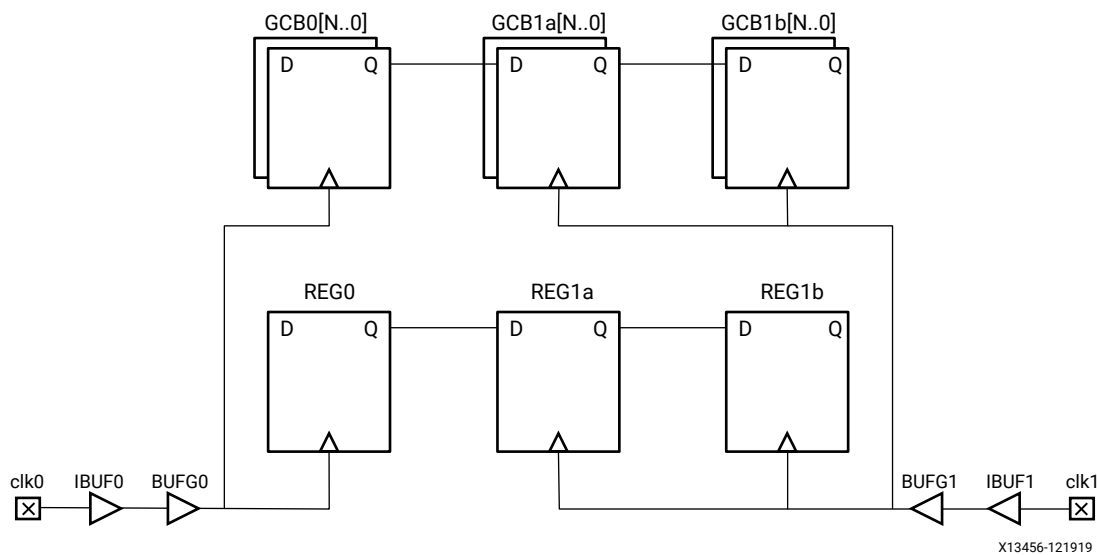
For the paths that do not need latency control, you can define a point-to-point false path constraint.

Clock Exceptions Precedence Over `set_max_delay`

When writing the CDC constraints, verify that the precedence is respected. If you use `set_max_delay -datapath_only` on at least one path between two clocks, the `set_clock_groups` constraint cannot be used between the same clocks, and the `set_false_path` constraint can only be used on the other paths between the two clocks.

In the following figure, the clock `clk0` has a period of 5 ns and is asynchronous to `clk1`. There are two paths from the `clk0` domain to the `clk1` domain. The first path is a 1-bit data synchronization. The second path is a multi-bit gray-coded bus transfer.

Figure 99: Multiple Interactions Between Two Asynchronous Clocks



The designer decides that the gray-coded bus transfer requires a Max Delay Datapath Only to limit the delay variation among the bits, so it becomes impossible to use a Clock Groups or False Path constraint between the clocks directly. Instead, two constraints must be defined:

```
set_max_delay -from [get_cells GCB0[*]] -to [get_cells [GCB1a[*]]] \
-datapath_only 5
set_false_path -from [get_cells REG0] -to [get_cells REG1a]
```

There is no need to set a false path from `clk1` to `clk0` because there is no path in this example.

Specifying Timing Exceptions

Timing exceptions are used to modify how timing analysis is done on specific paths. By default, the timing engine assumes that all paths should be timed with a single cycle requirement for setup analysis to cover the most pessimistic clocking scenario. For certain paths, this is not true. Following are a few examples:

- Asynchronous CDC paths cannot be safely timed due to the lack of fixed phase relationship between the clocks. They should be ignored (Clock Groups, False Path), or simply have datapath delay constraint (Max Delay Datapath Only)
- The sequential cells launch and capture edges are not active at every clock cycle, so the path requirement can be relaxed accordingly (Multicycle Path)
- The path delay requirement needs to be tightened to increase the design margin in hardware (Max Delay)
- A path through a combinatorial cell is static and does not need to be timed (False Path, Case Analysis)
- The analysis should be done with only a particular clock driven by a multiplexer (Case Analysis).

In any case, timing exceptions must be used carefully and must not be added to hide real timing problems.

Timing Exceptions Guidelines

Use a limited number of timing exceptions and keep them simple whenever possible. Otherwise, you will face the following challenges:

- The implementation compile time significantly increases when many exceptions are used, especially when they are attached to a large number of netlist objects.
- Constraints debugging becomes extremely complicated when several exceptions cover the same paths.
- Presence of constraints on a signal can hamper the optimization of that signal. Therefore, including unnecessary exceptions or unnecessary points in exception commands can hamper optimization.

Following is an example of timing exceptions that can negatively impact the run time:

```
set_false_path -from [get_ports din] -to [all_registers]
```

- If the `din` port does not have an input delay, it is not constrained. So there is no need to add a false path.

- If the `din` port feeds only to sequential elements, there is no need to specify the false path to the sequential cells explicitly. This constraint can be written more efficiently:

```
set_false_path -from [get_ports din]
```

- If the false path is needed, but only a few paths exist from the `din` port to any sequential cell in the design, then it can be more specific (`all_registers` can potentially return thousands of cells, depending upon the number of registers used in the design):

```
set_false_path -from [get_ports din] -to [get_cells blockA/config_reg[*]]
```

Timing Exceptions Precedence and Priority Rules

Timing exceptions are subject to strict precedence and priority rules. The most important rules are as follows:

- The more specific the constraint, the higher the priority. For example:

```
set_max_delay -from [get_clocks clkA] -to [get_pins inst0/D] 12
set_max_delay -from [get_clocks clkA] -to [get_clocks clkB] 10
```

The first `set_max_delay` constraint has a higher priority because the `-to` option uses a pin, which is more specific than a clock.

- The exceptions priority is as follows:
 1. `set_false_path`
 2. `set_max_delay` or `set_min_delay`
 3. `set_multicycle_path`

The `set_clock_groups` command is not considered a timing exception even though it is equivalent to two `set_false_path` commands between two clocks. It has higher precedence than the timing exceptions.

The `set_case_analysis` and `set_disable_timing` commands disable timing analysis on specific portions of the design. They have higher precedence than the timing exceptions.

For details on XDC precedence and priorities, see this [link](#) in the *Vivado Design Suite User Guide: Using Constraints (UG903)*.

Adding False Path Constraints

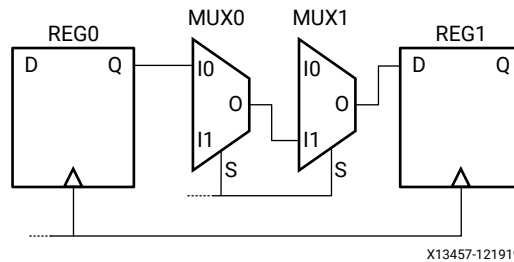
False path exceptions can be added to timing paths to ignore slack computation on these paths. It is usually difficult to prove that a path does not need timing to be functional, even with simulation tools. Xilinx does not usually recommend using a false path unless the risk associated with it has been properly assessed and appear to be acceptable.

Use Cases

The typical cases for using the false path constraint are:

- Ignoring timing on a path that is never active. For example, a path that crosses two multiplexers that can never let the data propagate in a same clock cycle because of the select pins connectivity.

Figure 100: Path Cannot be Sensitized



```
set_false_path -through [get_pins MUX0/I0] -through [get_pins MUX1/I1]
```

- Ignoring timing on an asynchronous CDC path.
- Ignoring static paths in the design. Some registers take a value once during the initialization phase of the application and never toggle again. When they appear to be on the critical path of the design, they can be ignored for timing to relax the constraints on the implementation tools and help with timing closure. It is sufficient to define a false path constraint from the static register only, without explicitly specifying the paths endpoints. For example, the paths from a 32-bit configuration register `config_reg[31..0]` to the rest of the design can be ignored by adding the following false path constraint:

```
set_false_path -from [get_cells config_reg[*]]
```

Impact on Synthesis

The false path constraint is supported by synthesis and will only impact max delay (setup/recovery) path optimization. It is usually not needed to use false path exceptions during synthesis except for ignoring CDC paths.

Impact on Implementation

All the implementation steps are sensitive to the false path timing exception.

Adding Min and Max Delay Constraints

The min and max delay exceptions are used to override the default path requirement respectively for hold/removal and setup/recovery checks by replacing the launch and capture edge times with the delay value from the constraint.

Use Cases

Common reasons for using the min or max delay constraints are as follows:

- Overconstraining a few paths of the design by tightening the setup/recovery path requirement.

This is useful for forcing the logic optimization or placement tools to work harder on some critical path cells, which can provide more flexibility to the router to meet timing later on (after removing the max delay constraint).

- Replacing a multicycle constraint.

This is a valid but not recommended way to relax the setup requirement on a path that has active launch and capture edges every N clock cycles. Although it is the only option to overconstrain a multicycle path by a fraction of a clock period to help with timing closure during the routing step. For example, a path with a multicycle constraint of 3 appears to be the worst violating path after route and fails timing by a few hundred ps.

The original multicycle path constraint can be replaced by the following constraint during optimization and placement, where 14.5 corresponds to 3 clock periods (of 5 ns each), minus 500 ps that correspond to amount of extra margin desired:

```
set_max_delay -from [get_pins <startpointCell>/C] \  
-to [get_pins <endpointCell>/D] 14.5
```

- Constraining the maximum datapath delay on asynchronous CDC paths

This technique is described in [Defining Clock Groups and CDC Constraints](#).

It is not common or recommended to force extra delay insertion on a path by using the `set_min_delay` constraint. The default min delay requirement for hold or removal is usually sufficient to ensure proper hardware functionality when the slack is positive.

Impact on Synthesis

The `set_max_delay` constraint is supported by synthesis, including the `-datapath_only` option. The `set_min_delay` constraint is ignored.

Impact on Implementation

The `set_max_delay` constraint replaces the setup path requirement and influences the entire implementation flow. The `set_min_delay` constraint replaces the hold path requirement and only affects the router behavior whenever it introduces the need to fix hold.

Avoiding Path Segmentation

Path segmentation is introduced when specifying invalid startpoint or endpoint for the `-from` or `-to` options of the `set_max_delay` and `set_min_delay` commands only. When a `set_max_delay` introduces path segmentation on a path, the default hold analysis no longer takes place. You must constrain the same path with `set_min_delay` if you desire to constrain the hold analysis as well. The same rule applies with the `set_min_delay` command relative to the setup analysis.

Path segmentation must only be used by experts as it alters the fundamentals of timing analysis:

- Path segmentation breaks clock skew computation on the segmented path.
- Path segmentation can break more paths than the one constrained by the segmenting `set_max_delay` or `set_min_delay` command.

Valid Startpoints and Endpoints

Path segmentation is reported by the tools in the log file when the constraints are applied. You must avoid it by using valid startpoints and endpoints:

- **Startpoints:** Clock, clock pin, sequential cell (implies valid startpoint pins of the cell), input or inout port.
- **Endpoints:** Clock, input data pin of sequential cell, sequential cell (implies valid endpoint pins of the cell), output or inout port.

For details on path segmentation, see this [link](#) in the *Vivado Design Suite User Guide: Using Constraints (UG903)*.

Adding Multicycle Path Constraints

Multicycle path exceptions must reflect the design functionality and must be applied on paths that do not have an active clock edge at every cycle, on either the source clock, the destination clock or both clocks. The path multiplier is expressed in terms of clock cycles, either based on the source clock when the `-start` option is used, or the destination clock when the `-end` option is used. This is particularly convenient for modifying the setup and hold relationships between the startpoint and endpoint independently of the clock period value.

The hold relationships are always tied to the setup ones. Consequently, in most cases, the hold relationship also needs to be separately adjusted after the setup one has been modified. This is why a second constraint with the `-hold` option is needed. The main exception to this rule is for synchronous CDC paths between phase-shifted clocks: only setup needs to be modified.

Relaxing the Setup Requirement While Keeping Hold Unchanged

This occurs when the source and destination sequential cells are controlled by a clock enable signal that activates the clock every N cycles. The following example has a clock enable active every three cycles, with the same clock for both startpoint and endpoint:

Figure 101: Enabled Flops with Same Clock Signal

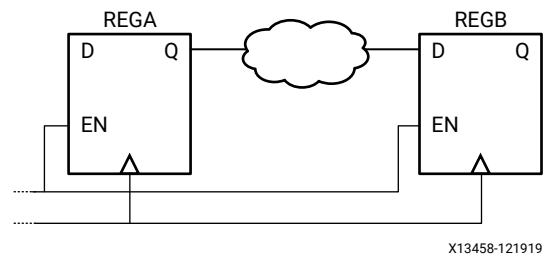
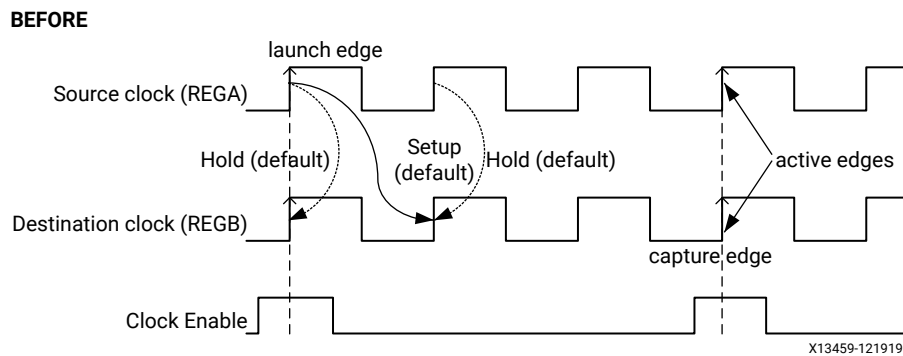


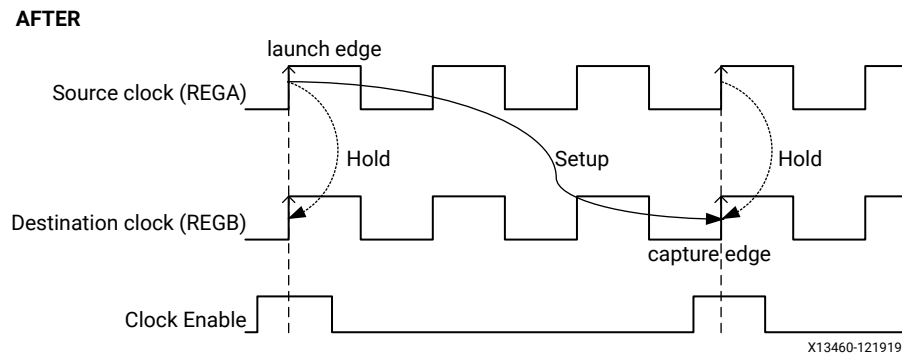
Figure 102: Timing Diagram for Setup/Hold Check



Constraints:

```
set_multicycle_path -from [get_pins REGA/C] -to [get_pins REGB/D] -setup 3
set_multicycle_path -from [get_pins REGA/C] -to [get_pins REGB/D] -hold 2
```

Figure 103: Setup/Hold Checks Modified After Multicycle Specification



Note: With the first command, as the setup capture edge moved to the third edge (that is, by two cycles from its default position), the hold edge also moved by two cycles. The second command is for bringing the hold edge back to its original location by moving it again by two cycles (in the reverse direction).

For more information on other common multicycle path scenarios, such as phase shift and multicycle paths between synchronous clocks, see this [link](#) in the *Vivado Design Suite User Guide: Using Constraints (UG903)*.



IMPORTANT! When the clock phase shift does not modify the clock waveform but is instead included in the insertion delay of the clock modifying block, you do not need to add a setup-only multicycle path to properly time the path from or to the clock. For more information, see this [link](#) in the *Vivado Design Suite User Guide: Design Analysis and Closure Techniques (UG906)*.

Impact on Synthesis and Implementation

The `set_multicycle_path` constraint is supported by synthesis and can greatly improve the timing QoR (for setup only) by relaxing long paths that are functionally not active at every clock cycle.

As for synthesis, multicycle path exceptions help the implementation timing-driven algorithms to focus on the real critical paths. The hold requirements are important only during routing. If a setup relationship was adjusted with a `set_multicycle_path` constraint but not its corresponding hold relationships, the worst hold requirement may become too hard to meet if it is over 2 or 3 ns. This situation can have a negative impact on setup slack because of the additional delay inserted by the router while fixing hold violations.

Common Mistakes

Following are common mistakes that you must avoid:

- Relaxing setup without adjusting hold back to same launch and capture edges in the case of a multicycle path not functionally active at every clock cycle.

The hold requirement can become very high (at least one clock period in most cases) and impossible to meet.

- Setting a multicycle path exception between incorrect points in the design.

This occurs when you assume that there is only one path from a startpoint cell to an endpoint cell. In some cases, this is not true. The endpoint cell can have multiple data input pins, including clock enable and reset pins, which are active on at least two consecutive clock edges.

For this reason, Xilinx recommends that you specify the endpoint pin instead of just the cell (or clock). For example, the endpoint cell REGB has three input pins: C, EN and D. Only the REGB/D pin should be constrained by the multicycle path exception, not the EN pin because it can change at every clock cycle. If the constraint is attached to a cell instead of a pin, all the valid endpoint pins are considered for the constraints, including the EN (clock enable) pin.

To be safe, Xilinx recommends that you always use the following syntax:

```
set_multicycle_path -from [get_pins REGA/C] \  
-to [get_pins REGB/D] -setup 3  
set_multicycle_path -from [get_pins REGA/C] \  
-to [get_pins REGB/D] -hold 2
```

Other Advanced Timing Constraints

A few other timing constraints can be set to ignore and modify the default timing analysis as described in the following sections.

Case Analysis

The case analysis command is commonly used to describe a functional mode in the design by setting constants in the logic like what configuration registers do. It can be applied to input ports, nets, hierarchical pins, or leaf cell input pins. The constant value propagates through the logic and turns off the analysis on any path that can never be active. The effect is similar to how the false path exception works.

The most common example is to set a multiplexer select pin to 0 or 1 to only allow one of the two multiplexer inputs to propagate through. The following example turns off the analysis on the paths through the `mux/S` and `mux/I1` pins:

```
set_case_analysis 0 [get_pins mux/S]
```


Disable Timing

The `disable_timing` command turns off a timing arc in the timing database, which completely prevents any analysis through that arc. The disabled timing arcs can be reported by the `report_disable_timing` command.



CAUTION! Use the `disable_timing` command carefully. It can break more paths than desired!

Data Check

The `set_data_check` command sets the equivalent of a setup or hold timing check between two pins in a design. For example, this constraint can be used to report timing on asynchronous interfaces. This command is ignored by the implementation tools and must only be used for timing reporting purposes, typically by expert users.

Max Time Borrow

The `set_max_time_borrow` command sets the maximum amount of time a latch can borrow from the next stage (logic after the latch), and give it the previous stage (logic before the latch). Latches are not recommended in general as they are difficult to test and validate in hardware. This command should be used by expert users.

Defining Power and Thermal Constraints

When developing your design in the Vivado tools or Vitis environment, you must ensure that your design is within the constraints of your power delivery and thermal solution, which is typically based on early power estimation as determined by Xilinx Power Estimator (XPE). It is extremely important to ensure your design is properly constrained, because changes to your power delivery and thermal solution can be costly.

At a minimum, Xilinx recommends applying the total power budget, maximum process, and worst-case junction temperature to create a worst-case power analysis, using the following XDC constraints:

```
set_operating_conditions -design_power_budget <Power in Watts>
set_operating_conditions -process maximum
set_operating_conditions -junction_temp <Max Tj based on Temp Grade>
```



POWER TIP: For a worst-case power estimation and until the Θ_{Ja} of the thermal solution is known, Xilinx recommends setting the T_j to the maximum allowed for the targeted temperature range. Θ_{Ja} can be calculated as follows based on the thermal simulation result: $\Theta_{Ja} = (T_j - T_a) / P_d$. Units are Celsius per watt ($^{\circ}\text{C}/\text{W}$).

The most accurate power estimation can be achieved after the Theta Ja of the thermal design is known. You can apply the Theta Ja and the maximum supported ambient temperature (Ta) of the application to `report_power` using the following constraints to replace the junction temperature setting. Using these constraints allows `report_power` to estimate the junction temperature more accurately and therefore, give a more accurate static power estimation.

```
set_operating_conditions -design_power_budget <Power in Watts>
set_operating_conditions -process maximum
set_operating_conditions -ambient_temp <Max Supported by Application>
set_operating_conditions -thetaja <Increase in Tj for every W dissipated C/W>
```

In addition, you can specify the power delivery design using XDC constraints. Using this approach allows `report_power` to report the margin on the total power, check the power estimation on the power rails, and report the margin based on the specified estimation and power rail consolidation. For more information on these constraints, see the *Vivado Design Suite User Guide: Power Analysis and Optimization* ([UG907](#)).

```
create_power_rail <power rail name> -power_sources {supply1, supply2, ..}
add_to_power_rail <power rail name> -power_sources {supply1, supply2, ..}
set_operating_conditions -supply_current_budget {<supply rail name>
<current budget in Amp>} -voltage {<supply rail name> <voltage>}
```



POWER TIP: Ensure the text power report is used for the most detailed power rail constraints reporting.

Defining Physical Constraints

Physical constraints are used to control floorplan, specific placement, I/O assignments, routers and similar functions. Make sure that each pin has an I/O location and standard specified. Physical constraints are covered in the following user guides:

- For locking placement and routing, including relative placement of macros, see the *Vivado Design Suite User Guide: Using Constraints* ([UG903](#)).
- For floorplanning, see the *Vivado Design Suite User Guide: Design Analysis and Closure Techniques* ([UG906](#)).
- For configuration, see the *Vivado Design Suite User Guide: Programming and Debugging* ([UG908](#)).

Floorplanning Constraints for Dynamic Function eXchange

Optimal floorplanning is one of the most critical aspects in DFX designs to ensure timing closure and avoid routability issues. This section provides best practices for achieving the maximum solution space for the router and quicker timing closure. For more information on DFX, see the *Vivado Design Suite User Guide: Dynamic Function eXchange* ([UG909](#)).

Reduce the Number of Partition Pins

In a DFX design, signals between the reconfigurable module (RM) and static region are called *boundary signals*. All RM pins must have a partition pin location (PPLOC) deposited on the boundary signal by the placer. The only exceptions are dedicated paths between hard primitives. The partition pin is the physical interface on fabric that separates the static and reconfigurable portions of a boundary signal. For more information on PPLOCs, see this [link](#) in the *Vivado Design Suite User Guide: Dynamic Function eXchange* ([UG909](#)).

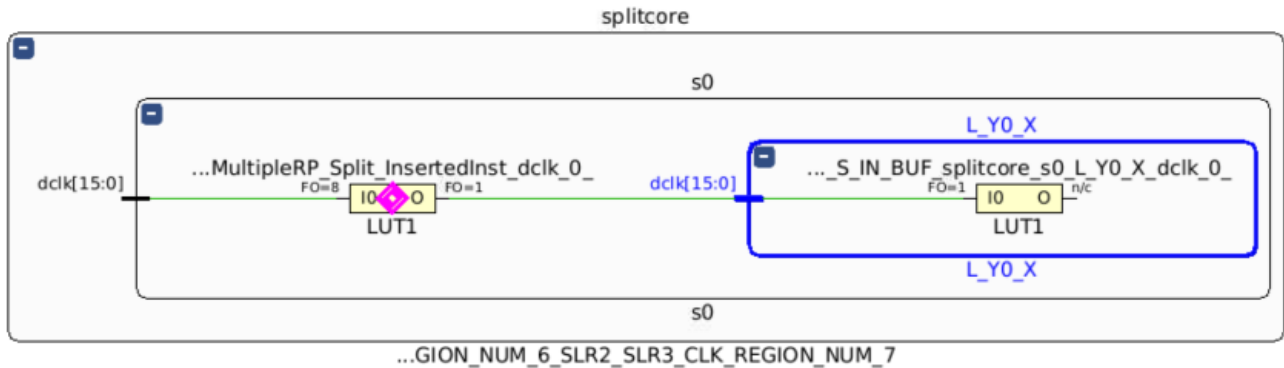
The presence of partition pins reduces the solution space for the router, because the corresponding boundary net is always forced to route through the partition pin. To alleviate this issue, the DFX flow includes expanded routing. Expanded routing is the additional routing footprint for a reconfigurable partition (RP) that can include routing tiles from the static region. For more information on expanded routing, see this [link](#) in the *Vivado Design Suite User Guide: Dynamic Function eXchange* ([UG909](#)).

The boundary nets of an RM have fanout in the static region as well as within the RM. In a DFX design, the loads of a boundary net in the static region are static boundary leaf cells. When a static boundary leaf cell is placed in the expanded routing footprint of an RP, the PPLOC is not needed and the router will have more flexibility routing to the static cell in subsequent RM implementations. Xilinx recommends expanded routing to reduce the dependency of the router on PPLOCs.

Note: PPLOC reduction occurs only on single fanout boundary signals. Therefore, Xilinx recommends avoiding multiple fanouts for boundary signals in a DFX design. For more information, see [Avoid Multiple RPs Driving Same Static Leaf Cell](#) and [Replicate Static Register Driving Multiple RPs](#).

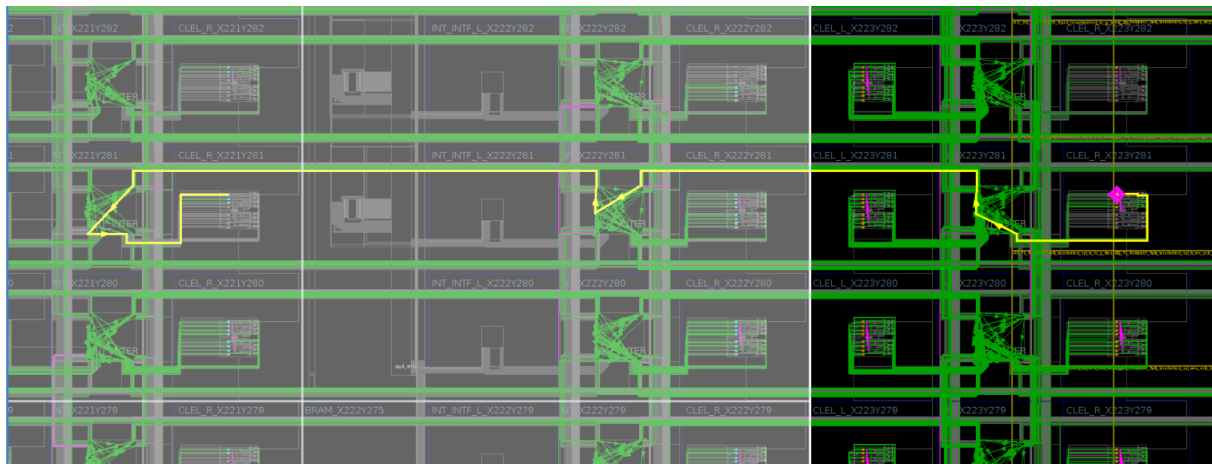
If PPLOC reduction is triggered for a boundary signal when a blackbox is created for the RM after initial implementation, the boundary net is removed up to the SITE or BEL pin of the static boundary leaf pin. Subsequent RM implementations route the boundary signal from the static boundary leaf pin to the RM logic. The following figure shows the PPLOC reduction in a boundary signal. LUT1 is the boundary static leaf cell and L_Y0_X is the RM.

Figure 104: Schematic of a Boundary Signal



The following figure shows the device view for the boundary net after `route_design`. The boundary signal is shown in yellow, and the boundary static leaf cell is shown in magenta. In this example, the boundary static leaf cell is placed in the expanded routing footprint of an RM. After initial implementation is complete and a blackbox is created for the RM, the boundary net is removed up to the static boundary leaf pin.

Figure 105: Routed Device View for Boundary Net



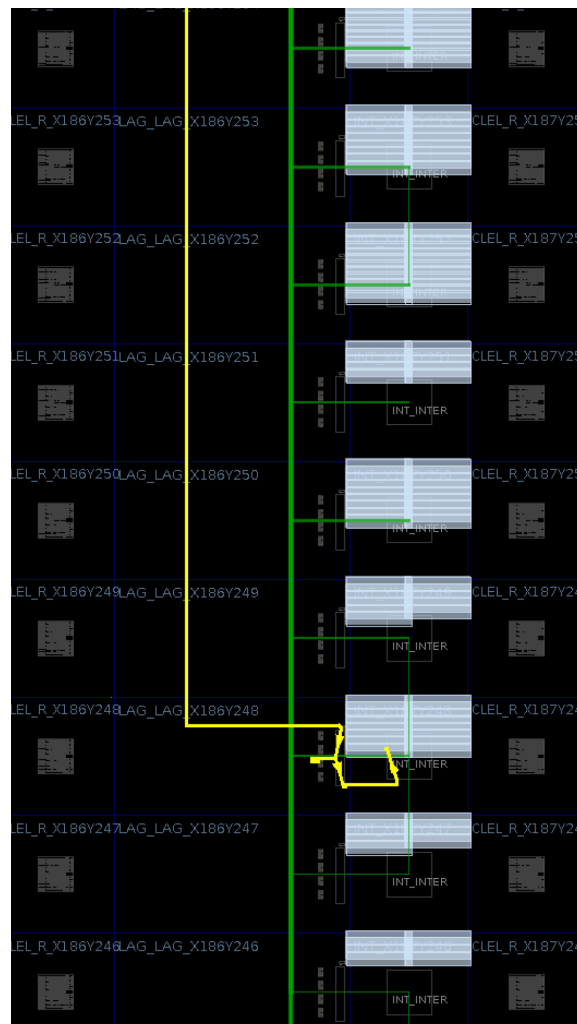
If PPLOC reduction is *not* triggered for a boundary signal when the blackbox is created for the RM after initial implementation, the static boundary net segment is preserved from the static leaf pin to the PPLOC. During subsequent RM implementations, boundary nets are routed only from the PPLOC to the RM logic. This reduces the solution space for the router due to the following:

- The static segment of the boundary net (from the static boundary leaf pin to the PPLOC) is locked down during all subsequent implementations. The router must obey the `IS_FIXED_ROUTE` constraint on the static segment of the boundary signal and cannot reroute during subsequent RM implementations.

- The presence of locked static nets (IS_FIXED_ROUTE TRUE) at the boundary of the reconfigurable Pblock causes the tool to exclude some sites from placement, because access to these sites might be blocked by the locked static nets.
- The PPLOC deposit occurs only on single or double interconnect nodes. This approach has lower connectivity than using site pins and is equivalent to having no PPLOCs.

The following figure shows the device view for the static boundary net segment that terminates at the PPLOC when the static boundary leaf cell is not placed in the expanded routing footprint.

Figure 106: Device View for Static Boundary Net Segment

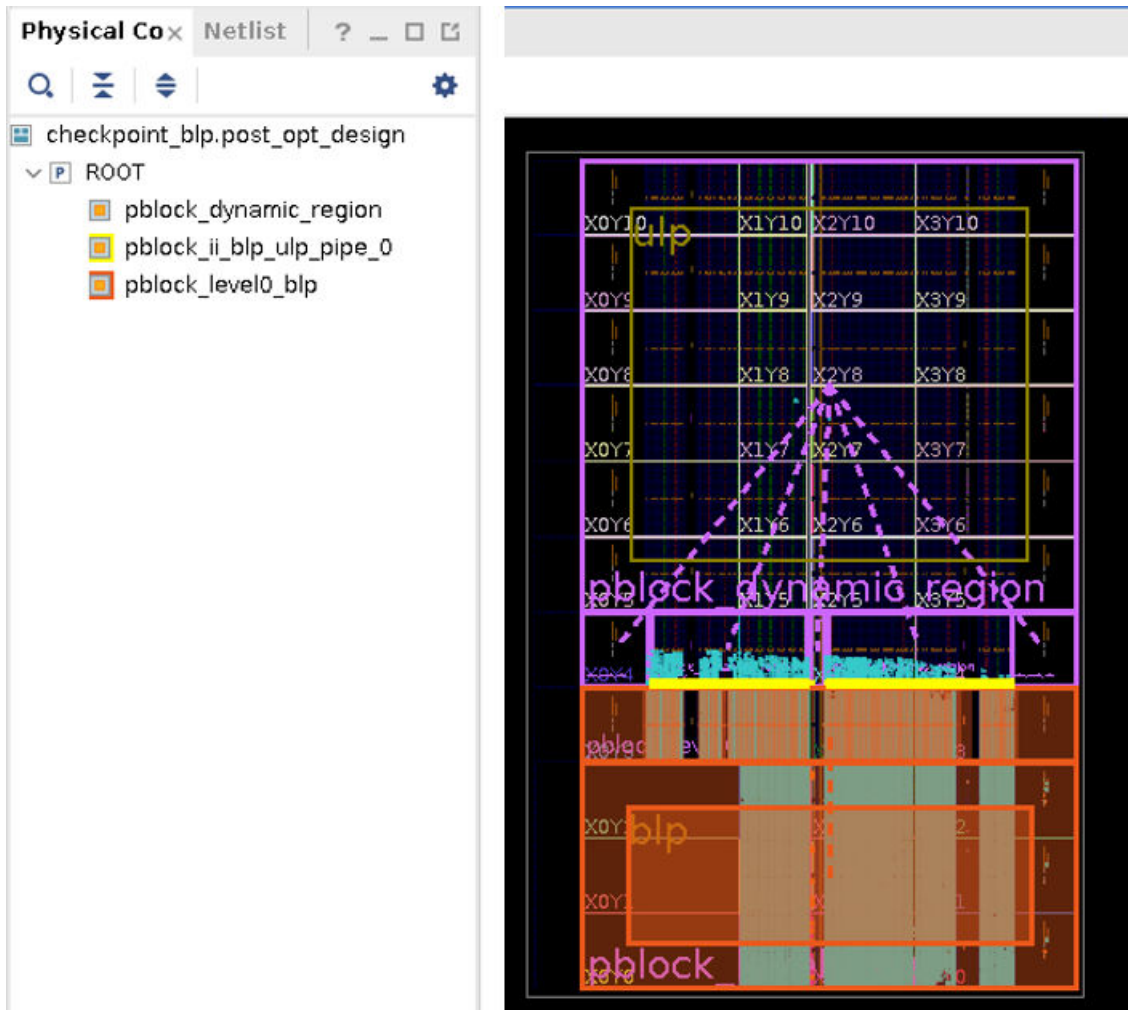


The routing footprint of the reconfigurable Pblock (pblock_dynamic_region) is the same as the reconfigurable Pblock size (CLOCKREGION_X0Y4: CLOCKREGION_X4Y10). All of the static region logic is assigned to the static Pblock region (pblock_static_region), which is outside the routing footprint of the reconfigurable Pblock. Therefore, PPLOC reduction is not triggered, and the reconfigurable Pblock contains a large number of PPLOCs after `route_design`.

In the following example, static boundary leaf cells to the reconfigurable Pblock (pblock_dynamic_region) are assigned to a thin static Pblock (pblock_ii_blp_ulp_pipe_0), which is defined in the expanded routing footprint of the RP Pblock. There are no PPLOCs remaining after `route_design`.

The following figure shows the static boundary leaf cells assigned to a thin static Pblock defined in the expanded routing footprint of the RP.

Figure 107: Static Boundary Leaf Cells in the Expanded Routing Footprint of the RP



To achieve maximum PPLOC reduction, Xilinx recommends that you guide the placer to keep static boundary leaf cells in the expanded routing footprint of the reconfigurable Pblock. One way to achieve this is to use thin static Pblocks defined in the expanded routing footprint of the reconfigurable Pblock.



TIP: To highlight the tiles in the routing footprint of a reconfigurable Pblock, source the `<pblock_name>_Routing_AllTiles.tcl` Tcl script generated by the placer and located in the `hd_visual` folder in the implementation directory.

Recommended Netlist Structure at the DFX Boundary for Maximum PPLOC Reduction

Avoid RP to RP Direct Paths

Xilinx recommends avoiding direct timing paths between multiple reconfigurable partitions (RPs) for the following reasons:

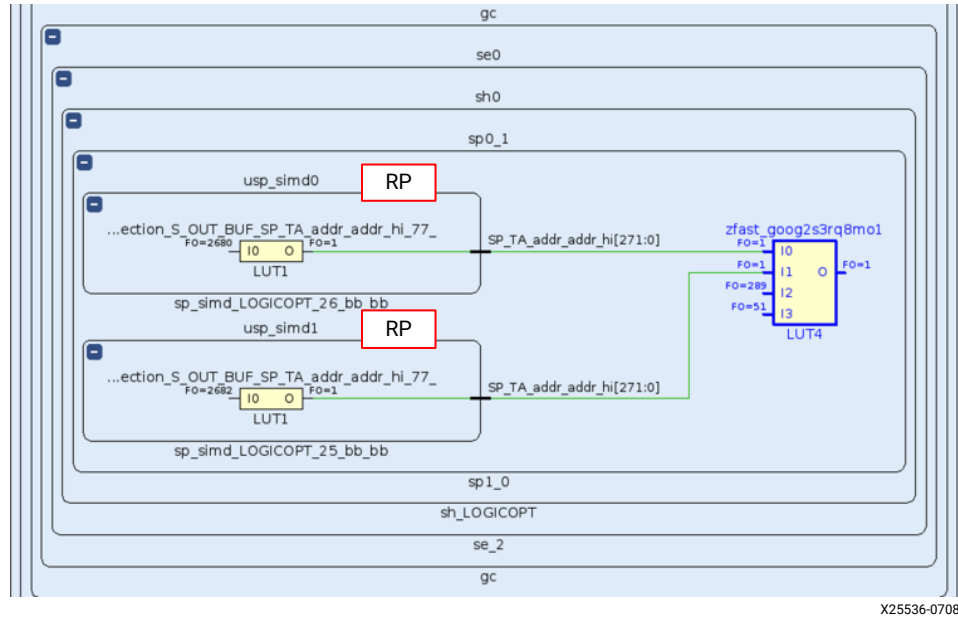
- If the boundary signal between the RPs does not have a static boundary leaf cell, the DFX flow must deposit a PPLOC on both RPs. As a result, tool capabilities like expanded routing with PPLOC reduction cannot be used. The presence of PPLOCs also causes routability challenges in subsequent reconfigurable module (RM) implementations, also known as child implementations in Vivado Project Mode.
- If the timing paths across an RP do not have a static boundary leaf cell, there might be a combination of RMs in two RPs that do not meet timing. The omission of a synchronous timing point in the static portion of the design can also lead to timing and hardware failures depending on the RMs that are currently loaded. The HDPR-34 and HDPR-35 DRCs flag this issue.

Avoid Multiple RPs Driving Same Static Leaf Cell

Ensure that a static boundary leaf cell is connected to only one reconfigurable partition (RP). PPLOC reduction is triggered for a reconfigurable module (RM) pin only if the static boundary leaf cell is placed in the expanded routing footprint of the RP. A leaf cell with individual leaf pins connected to multiple RPs can be placed in the expanded routing footprint of only one RP. Therefore, PPLOC reduction cannot occur on two RPs at the same time.

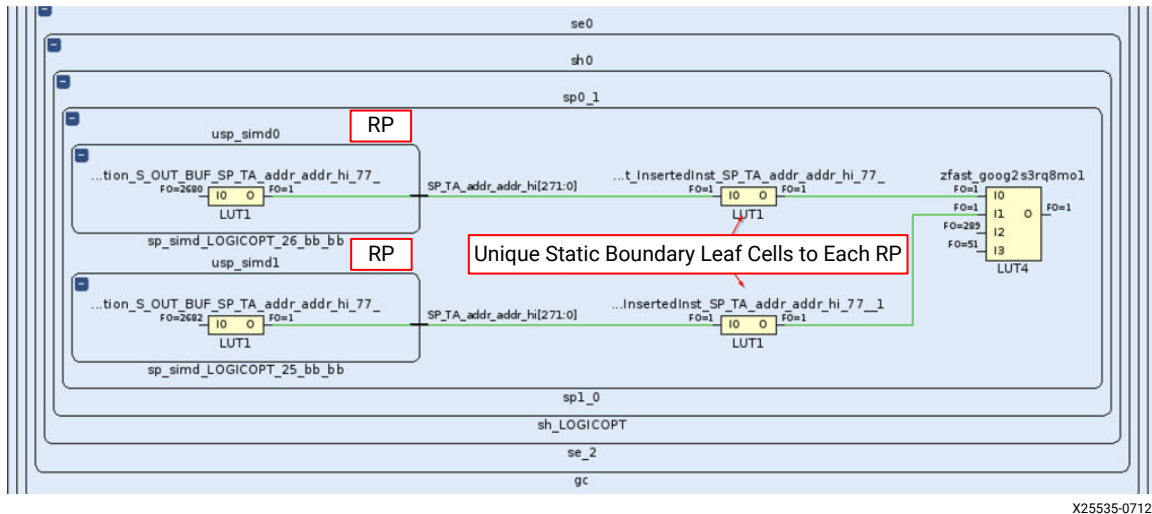
In the following example, two RPs are connected to the same static boundary leaf cell. This approach is *not* recommended, because it negatively impacts routability.

Figure 108: Two RPs Driving the Same Static Boundary Leaf Cell



In the following example, two RPs each drive different static boundary leaf cells. LUT1 is inserted to separate the static boundary leaf cells. This approach is recommended for improved routability.

Figure 109: Two RPs Driving Unique Static Boundary Leaf Cells

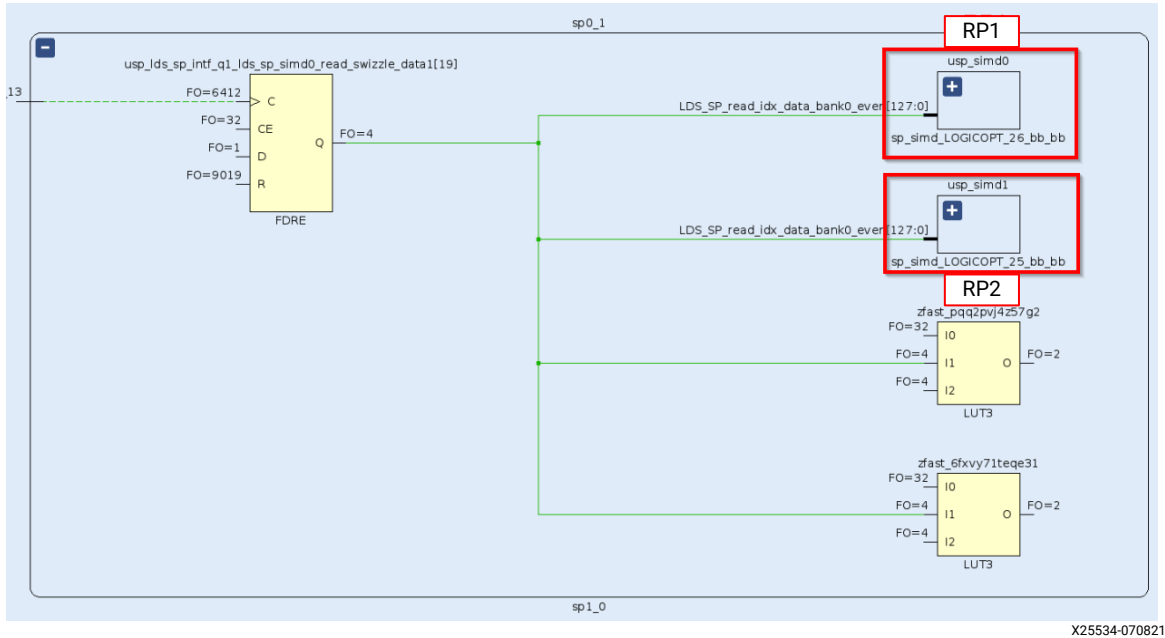


Replicate Static Register Driving Multiple RPs

To ensure that a static boundary leaf cell is not shared by multiple reconfigurable partitions (RPs), avoid single registers that drive multiple RPs. The DFX-1 methodology check (`report_methodology`) flags this violation.

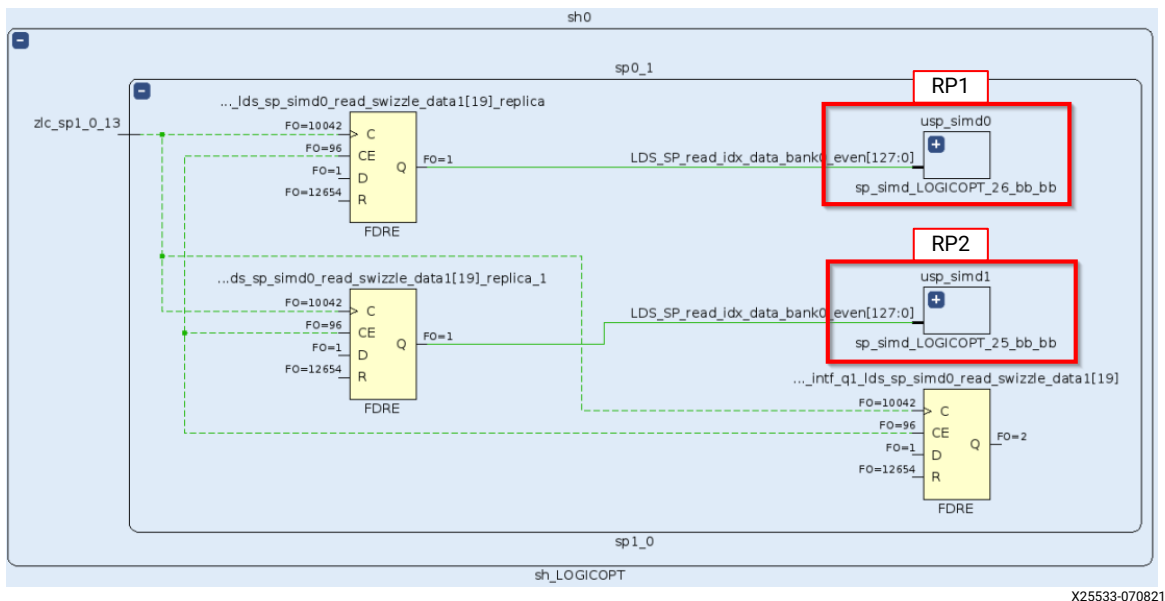
In the following example, a single static register drives multiple RPs as well as static logic. This approach is *not* recommended in the DFX flow.

Figure 110: One Static Register Driving Multiple RPs and Static Logic



In the following example, two separate but equivalent registers drive two RPs. This is the recommended approach when using the DFX flow.

Figure 111: Two Registers Driving Two RPs



Register Inputs and Outputs of RMs

Xilinx recommends registering inputs and outputs of reconfigurable modules (RMs) in a DFX design for multiple reasons.

In the parent implementation, the RM used for the partition does not need to be the actual design. Instead, the RM can be training logic used as a placeholder while you define the platform. If the training logic is sub-optimal and there is combinatorial logic in the static portion of the boundary timing paths, it is likely that the static portion of the path consumes a significant amount of timing budget of the path. During child implementation, this can cause timing closure issues for the signals in the RM connected to this boundary signal.

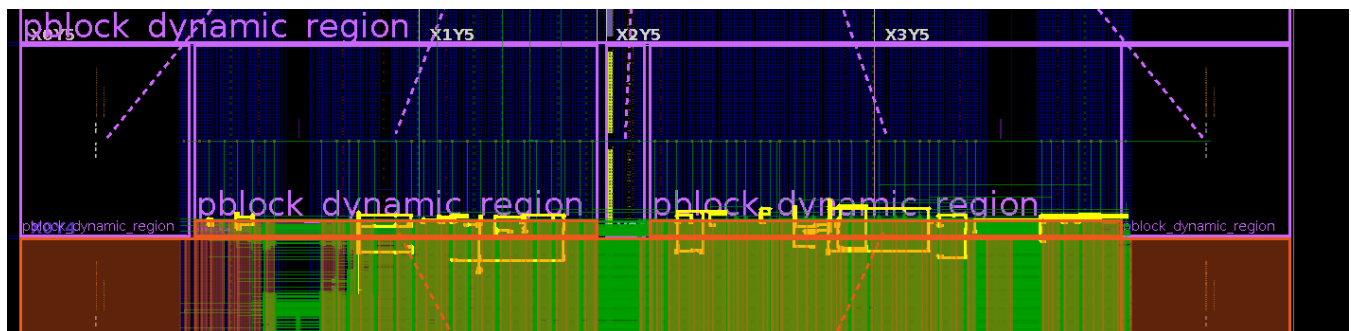
In addition, creating an abstract shell of a reconfigurable partition (RP) prunes most of the static region and keeps only the logic up to the first sequential cell in the static region. Registering the input and output pins of the RMs enables maximum abstraction, thereby reducing the size of the abstract shell. For more information on abstract shells, see this [link](#) in the *Vivado Design Suite User Guide: Dynamic Function eXchange (UG909)*.

Reduce Bleed Over of Static Nets to the Reconfigurable Pblocks

By default, nets in the static region of a DFX design can use any routing resources in the device. However, this might cause the static nets to bleed over to the dynamic region. Although allowed from a functional perspective, this approach reduces the solution space of the router for reconfigurable modules (RMs) inside the reconfigurable partition (RP) Pblock. After the first implementation is complete and static routes and placement are locked using the `lock_design` command, the static nets are locked with some of the nets in the RP Pblock. During subsequent child implementations, the DFX flow identifies these locked static nets during RM implementation and attempts to perform place and route in a reduced solution space to avoid unroutability. To avoid the bleed over of static nets to RP Pblocks, Xilinx recommends containing the static nets inside a static Pblock by setting the `CONTAIN_ROUTING TRUE` constraint.

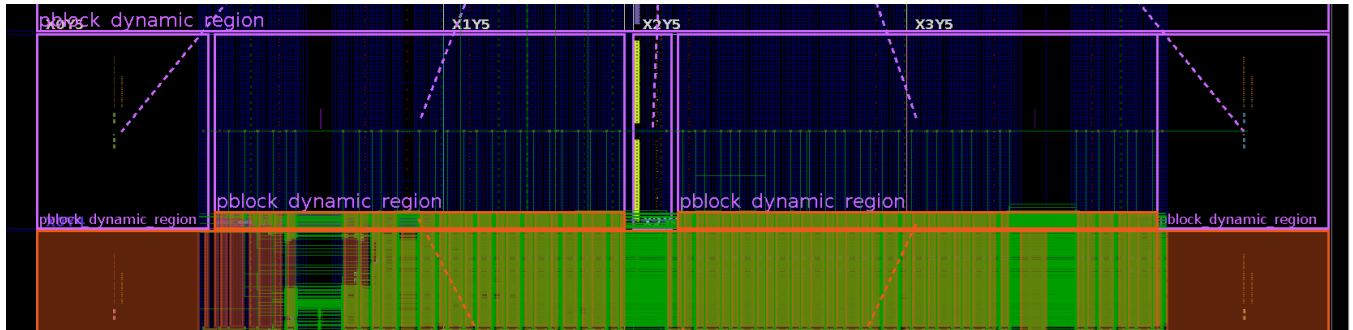
In the following example, the static region Pblock does not have the `CONTAIN_ROUTING` constraint enabled. The bleed over from the static nets to the RP Pblock is highlighted in yellow. This approach is *not* recommended, because it negatively impacts routability during RM compile.

Figure 112: Static Region Pblock without `CONTAIN_ROUTING` Constraint



In the following example, the CONTAIN_ROUTING constraint is enabled on the static Pblock, and there is no bleed over of static nets to the RP Pblock. This is the recommended approach for improved routability during RM compile.

Figure 113: Static Region Pblock with CONTAIN_ROUTING Constraint



Make Pblocks as Rectangular as Possible to Avoid Unroutability at the Edges

For Pblocks with CONTAIN_ROUTING enabled, the router has less solution space at the corners of the Pblock. To avoid unroutable situations, Xilinx recommends making Pblocks as rectangular as possible. Using another shape results in more corners for the Pblock, which can introduce congestion at those corners.

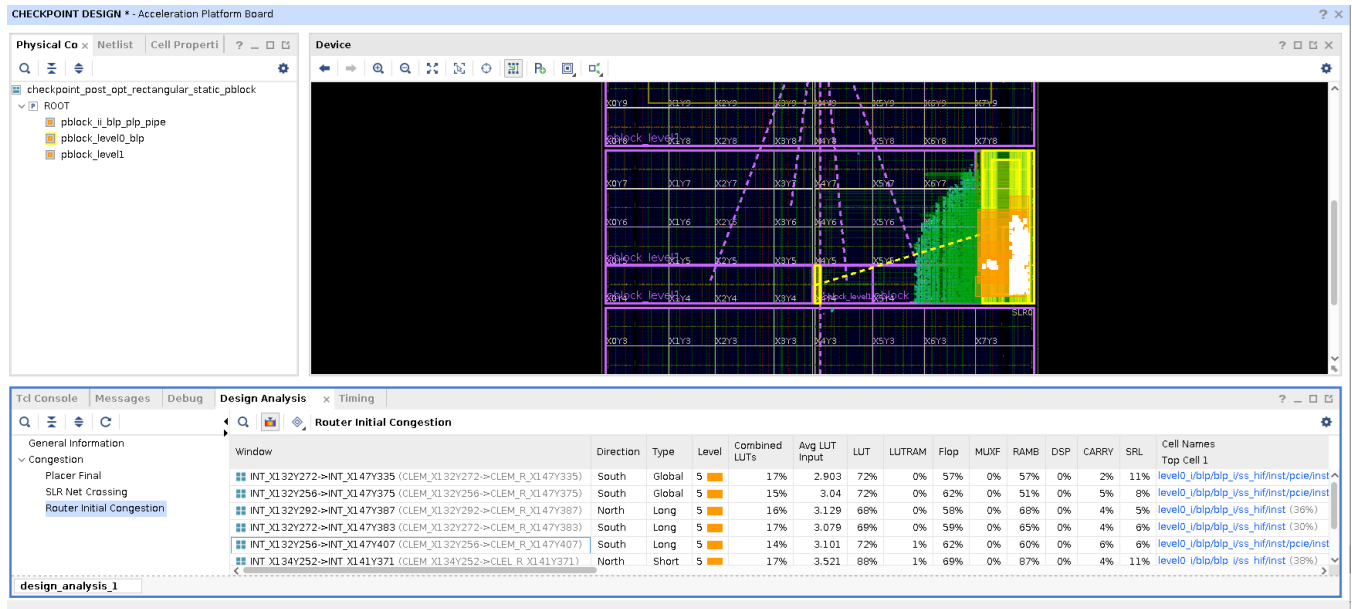
The following figure shows a non-rectangular static Pblock (shown in yellow) with an overlapping congestion area (shown in white). In this example, report_design_analysis reports a congestion of level 6 at the edges of the Pblock. This approach is *not* recommended.

Figure 114: Non-Rectangular Static Pblock with Congestion at the Corners

Window	Direction	Type	Level	Combined LUTs	Avg LUT Input	LUT	LUTRAM	Flop	MUXF	RAMB	DSP	CARRY	SRL	Cell Names
INT_X1_32Y272->INT_X147Y415 (CLEM_X1_32Y272->CLEM_R_X1_47Y415)	South	Global	6	11%	2.911	61%	2%	55%	0%	74%	0%	5%	6%	level0_i/blk/blk_vss_hif/inst (24%)
INT_X1_32Y275->INT_X147Y370 (CLEM_X1_32Y275->CLEM_R_X1_47Y370)	North	Long	5	9%	2.579	55%	0%	51%	0%	65%	0%	2%	8%	level0_i/blk/blk_vss_hif/inst/pcie/inst
INT_X1_32Y264->INT_X147Y423 (CLEM_X1_32Y264->CLEM_R_X1_47Y423)	South	Long	6	12%	2.935	61%	2%	55%	0%	71%	0%	5%	5%	level0_i/blk/blk_vss_hif/inst (24%)
INT_X1_36Y253->INT_X143Y324 (CLEM_X1_36Y253->CLEM_R_X1_43Y324)	South	Short	5	19%	2.958	71%	0%	74%	0%	57%	0%	0%	16%	level0_i/blk/blk_vss_hif/inst/pcie/inst
INT_X1_31Y268->INT_X146Y299 (CLEM_X1_31Y268->CLEM_R_X1_46Y299)	East	Short	5	13%	2.242	52%	0%	52%	0%	50%	0%	1%	11%	level0_i/blk/blk_vss_hif/inst/pcie/inst
INT_X1_33Y355->INT_X140Y426 (CLEM_X1_33Y355->CLEM_R_X1_40Y426)	West	Short	5	21%	4.113	81%	5%	67%	0%	93%	0%	5%	1%	level0_i/blk/blk_vss_hif/inst (66%)

The following figure shows the same design but with a rectangular Pblock (shown in yellow) and a reduced congestion area (shown in white). With this approach, `report_design_analysis` reports a reduced congestion level of 5 with no congestion reported at the edge of the rectangular Pblock. This is the recommended approach.

Figure 115: Rectangular Static Pblock with Reduced Congestion



Considerations for Static Pblocks with CONTAIN_ROUTING Enabled

Keep Routability as a Factor in Utilization

When defining Pblocks, you might look at utilization from a placement perspective only. However, for static Pblocks with the `CONTAIN_ROUTING` constraint enabled, routability is also an important factor. Unlike Pblocks without a containment requirement, the router must find a solution using the available routing tiles inside the static Pblock. Therefore, it is important to stay within the recommended utilization value. This approach provides more solution space for the router and allows you to converge to a solution more quickly. See the utilization recommendations for flat designs using `report_qor_assessment - full_assessment_details`.

Reduce the Number of Unique Control Sets

CLB packing restrictions caused by unique control sets can introduce sub-optimal placement and higher net delay. On a Pblock that already has a `CONTAIN_ROUTING` requirement, the additional restriction of unique control sets puts more constraints on the router, which might lead to an unroutable situation. Therefore, it is very important to reduce the unique control sets on static logic that is assigned to a Pblock, especially if the Pblock has `CONTAIN_ROUTING` enabled.

Related Information

[Reducing Control Sets](#)

Reduce the Detour Due to Hold Violations

The router gives priority to fixing hold violations by detouring through longer paths. However, this adds more restrictions if the violations are inside the CONTAIN_ROUTING static Pblocks. There might not be enough solution space for detouring inside the static Pblock that includes a CONTAIN_ROUTING requirement. Therefore, Xilinx strongly recommends having a good post-place timing summary for such logic.

Related Information

[Reducing Control Sets](#)

[Fixing Large Hold Violations Prior to Routing](#)

Exclude Containment Requirement for Nets

If you have routability challenges inside a static Pblock that has CONTAIN_ROUTING enabled, try the following options:

- Increase the Pblock size and reduce the overall utilization.
- Allow bleed over of certain static nets to the reconfigurable partition (RP) by setting the following property on those nets:

```
set_property HD.NO_ROUTE_CONTAINMENT 1 [get_nets <net_name>]
```

Design Implementation

After selecting your device, choosing and configuring the IP, and writing the RTL and the constraints, the next step is implementation. Implementation compiles the design through synthesis and place and route, and then generates the file that is used to program the device. The implementation process might have some iterative loops. This chapter describes the various implementation steps, highlights points for special attention, and gives tips and tricks to identify and eliminate specific bottlenecks.



IMPORTANT! You must regularly validate that synthesis and implementation occur without errors and with minimal timing violations before adding new blocks or generating a platform for the Vitis™ tools.

Note: The implementation steps are run automatically as part of the Vitis environment flow. You can improve performance by applying the techniques described in this chapter using the Vitis command line options and configuration file. For more information, see the [Vitis Unified Software Platform Documentation](#).

Running Synthesis

Synthesis takes in RTL and timing constraints and generates an optimized netlist that is functionally equivalent to the RTL. In general, the synthesis tool can take any legal RTL and create the logic for it. Synthesis requires realistic timing constraints.

For additional information about synthesis, refer to the following resources:

- [Vivado Design Suite User Guide: Synthesis \(UG901\)](#)
- [Vivado Design Suite QuickTake Video: Design Flows Overview](#)

Related Information

[Design Constraints](#)

[Baselining the Design](#)

Synthesis Flows

In the Vivado® Design Suite, you can run the synthesis flows described in the following sections, which each have different advantages and trade-offs.

Global Synthesis

In the global synthesis flow, the full design is synthesized in one run, which offers the following advantages:

- Allows the synthesis tool to perform the maximum optimization. Because the synthesis tool is aware of the full design, the tool can optimize between hierarchies that other flows might not.
- Enables easy analysis after the synthesis run.

The disadvantage of this flow is longer compile time. Every time synthesis is run, the full design is rerun. However, this disadvantage can be mitigated by using incremental synthesis.

Note: If your design includes XDC constraints, you must reference the objects to the top-level design.

Related Information

[Incremental Synthesis](#)

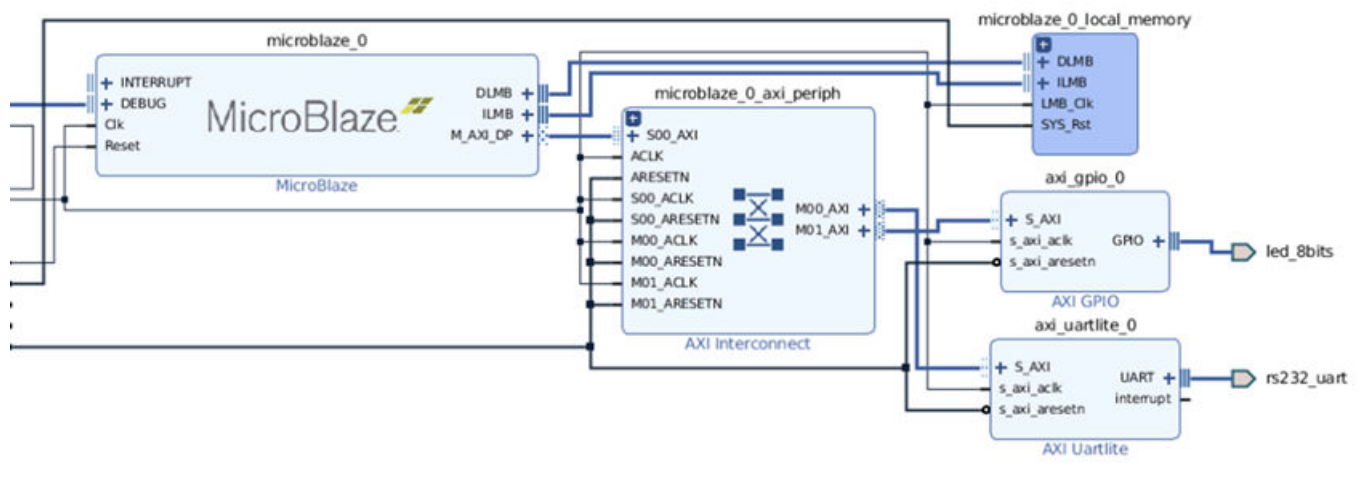
Block Design Synthesis

The block design synthesis flow allows you to create complex systems using custom and Xilinx IP. In this flow, a block design (BD) file is created using the Vivado IP integrator. Xilinx or custom IP is added to the .bd file and connected as a system. This flow offers the following advantages:

- Encapsulates a great deal of functionality into a small design.
- Allows focus on the system rather than individual parts of the system.
- Enables easier and faster setup and synthesis of the design.

The following figure shows an example of a block design.

Figure 116: Block Design Example



When creating the block design, you can run synthesis using either out-of-context (OOC) synthesis mode or global synthesis mode. If you use out-of-context synthesis mode, the block design is synthesized separately from the rest of the design. This allows for faster resynthesis when hierarchies outside the BD file are modified. If you use global synthesis mode, the full design is compiled and synthesized each time. Global synthesis mode is easier to set up because constraints are set on a global level. However, using this mode results in a higher run time on resynthesis. You can improve run time using incremental synthesis.

Out-of-Context Synthesis

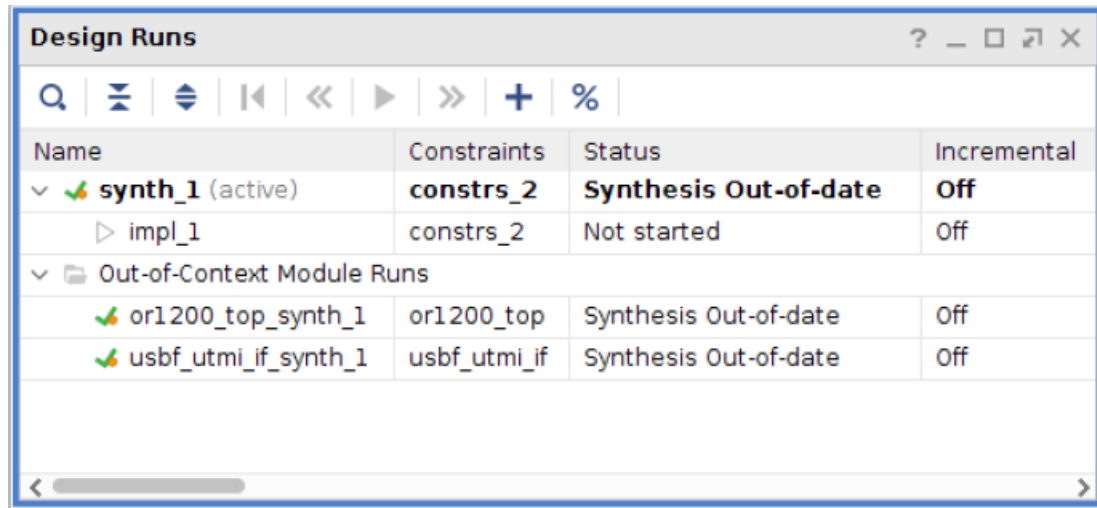
In the out-of-context (OOC) synthesis flow, certain levels of hierarchy are synthesized separately from the top-level. The out-of-context hierarchy are synthesized first. Then, the top-level synthesis is run, and each of the out-of-context runs are treated as a black box. Xilinx IP are often run in out-of-context synthesis mode. After all of the out-of-context synthesis runs and top-level synthesis runs are complete, the Vivado tools assemble the design from all of the synthesis runs when you open the top-level synthesized design. This flow offers the following advantages:




- Reduces compile time for subsequent synthesis runs. Only the runs you specify are resynthesized, leaving the other runs as is.
- Ensures stability when design changes are made. Only the runs that include changes are resynthesized.

The disadvantage of this flow is that it requires additional setup. You must be careful in selecting which modules to set as out-of-context synthesis modules. Any additional XDC constraints must be defined separately and must only be used for the out-of-context synthesis runs.

The following figure shows a design that has a top-level synthesis run (synth_1) and two lower-level out-of-context synthesis runs.

Figure 117: Top-Level Synthesis with Two Out-of-Context Synthesis Runs



Name	Constraints	Status	Incremental
▼  synth_1 (active)	constrs_2	Synthesis Out-of-date	Off
▶ impl_1	constrs_2	Not started	Off
▼ Out-of-Context Module Runs			
 or1200_top_synth_1	or1200_top	Synthesis Out-of-date	Off
 usbif_utmi_if_synth_1	usbif_utmi_if	Synthesis Out-of-date	Off

For more information on setting up out-of-context synthesis runs, see this [link](#) in the *Vivado Design Suite User Guide: Synthesis (UG901)*.

Incremental Synthesis

You can use incremental synthesis to reuse existing synthesis results. This approach offers the following advantages:

- Reduces typical synthesis compile times by 50%.
- When used with the incremental implementation flow, improves overall compile time and timing closure consistency.

When a design is synthesized, it is broken into RTL partitions. Incremental synthesis reuses RTL partitions from a previous synthesis run. RTL partitions are typically created along logical hierarchies. Incremental synthesis only runs if the design is large enough that synthesis creates at least 4 RTL partitions, each containing at least 25000 instances. Instances include both logical hierarchy and RTL primitives.

Following are the different modes available when using the `synth_design -incremental_mode <value>` command:

- `off`: Incremental synthesis is not run.
- `quick`: Fastest results but no cross boundary optimizations. This mode limits logic performance.
- `default`: Most logic optimizations enabled, including cross-boundary optimization. Compile time is significantly reduced from non-incremental synthesis.

- **aggressive:** All optimizations are enabled. Compile time is significantly reduced from non-incremental synthesis.

The `quick` mode is typically recommended for low-performance designs only. Without cross boundary optimizations, typical designs have reduced performance. However, if a design is well constructed with registered hierarchical boundaries, performance might not be affected. Due to the limits on cross-boundary optimization, resynthesis in a given area is caused by RTL changes only in that area. Changes in another synthesis partition do not trigger changes beyond that partition. This leads to more reuse and a faster synthesis result. For other modes, this is not the case, and RTL changes might trigger the resynthesis of more partitions beyond just the partition the cell is in. When more than 50% of partitions are modified, a full resynthesis is triggered.

For high-performance designs, the `default`, `aggressive`, and `off` modes are recommended. More optimizations are enabled in `aggressive` and `off` modes, which might lead to more resynthesis but higher QoR.

To more aggressively address compile time concerns, you can compare the incremental synthesis against OOC synthesis. OOC synthesis is typically used by IP, and setup is automatic. Global synthesis with incremental synthesis offers the advantage of cross-boundary optimizations as well as compile time benefits. Following are areas to consider:

- Compile time

OOC synthesis is faster, because it reduces the amount of code that is elaborated each time. RTL is only elaborated if the RTL is modified within the OOC module.

- Performance

Incremental synthesis has a performance advantage over OOC synthesis, because optimization is not possible across OOC boundaries.

- Setup

For non-IP flows, when you create an OOC synthesis run, you must create a wrapper when generics/parameters are passed from higher modules. In addition, you must create a separate timing constraint file to target the OOC-level ports. Incremental synthesis does not have these requirements.

For more information, see the *Vivado Design Suite User Guide: Synthesis* ([UG901](#)).

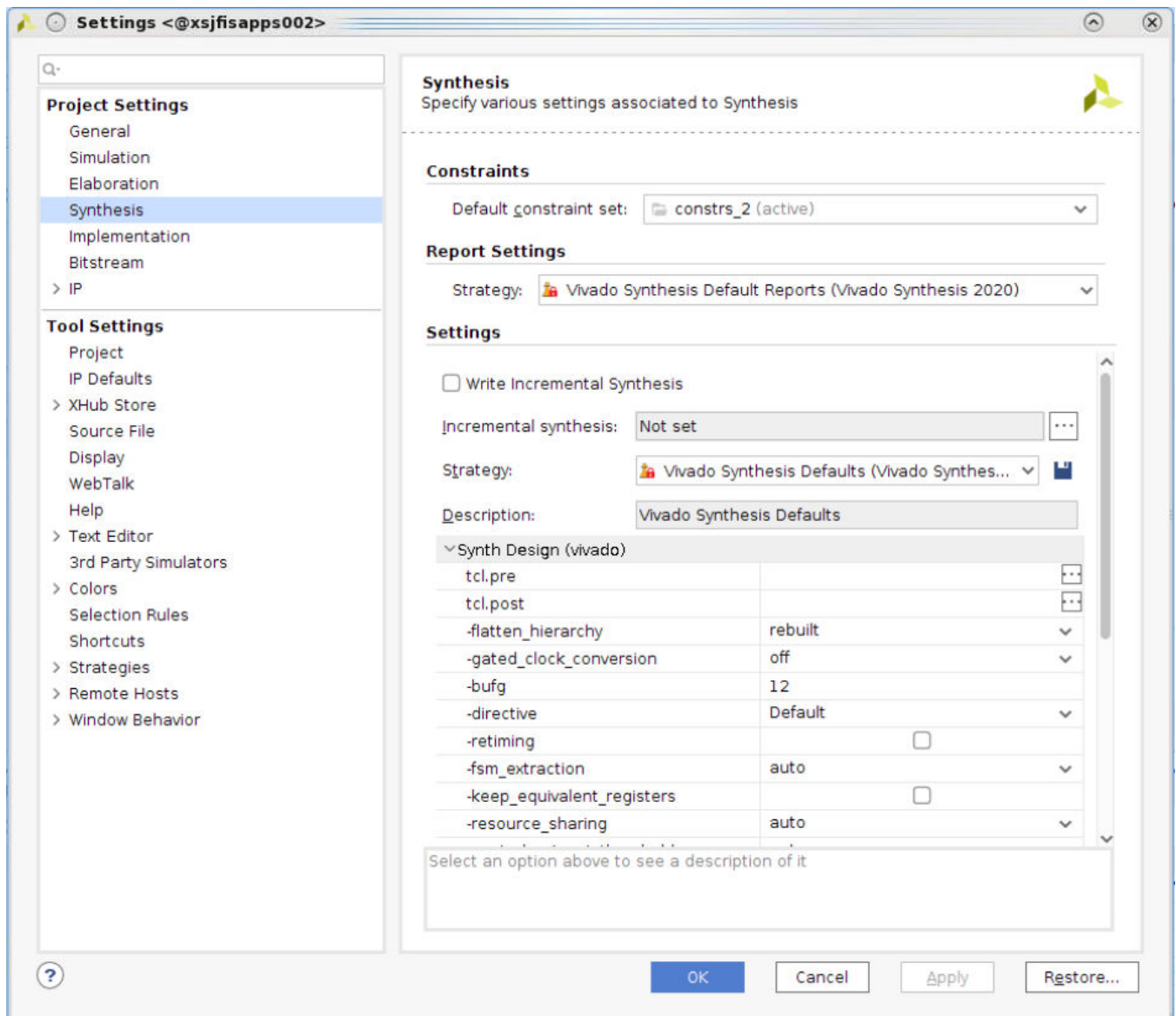
Synthesis Optimizations

By default, Vivado synthesis applies optimizations that yield the best results for the largest number of designs. However, you can adjust the default synthesis optimizations as described in the following sections.

Synthesis Settings

You can set several global settings that affect the entire design using the Vivado Design Suite Synthesis Settings. These settings optimize how logic is inferred and how incremental synthesis is run. Xilinx recommends using default options when you start your design and changing the options based on the specific needs of your design. For more information, see this [link](#) in the *Vivado Design Suite User Guide: Synthesis* (UG901).

Figure 118: Vivado Synthesis Settings



Synthesis Attributes

Synthesis attributes allow you to control the logic inference in a specific way. Although synthesis algorithms are set to give the best results for the largest number of designs, there are often designs with differing requirements. In this case, you can use attributes to alter the design to improve QoR. For information on the attributes supported by synthesis, see the *Vivado Design Suite User Guide: Synthesis (UG901)*.



POWER TIP: Evaluate synthesis settings carefully, because these settings can have a considerable impact on the power consumption of a design. For example, a low setting for the control set threshold increases the usage of register clock enables at the expense of less dense packing. Run the `report_power` command after synthesis to evaluate the impact of synthesis settings on power.

Note: Before retargeting your design to a new device, Xilinx recommends reviewing any synthesis attributes from previous design runs that target older devices.

When using the KEEP, DONT_TOUCH, and MAX_FANOUT attributes, be aware of the special considerations described in the following sections.

KEEP and DONT_TOUCH

KEEP and DONT_TOUCH are valuable attributes for debugging a design. They direct the tool to not optimize the objects on which they are placed.

- KEEP is used by the synthesis tool and is not passed along as a property in the netlist. KEEP can be used to retain a specific signal, for example, to turn off specific optimizations on the signal during synthesis.
- DONT_TOUCH is used by the synthesis tool and then passed along to the place and route tools so the object is never optimized.

Take care when using these attributes:

- A KEEP attribute on a register that receives RAM output prevents that register from being merged into the RAM, thereby preventing a block RAM from being inferred.
- Do not use these attributes on a level of hierarchy that is driving a 3-state output or bidirectional signal in the level above. If the driving signal and the 3-state condition are in this level of hierarchy, the IOBUF is not inferred, because the tool must change the hierarchy to create the IOBUF.
- Attributes that disable optimization often result in larger, higher power-consuming circuits. Xilinx recommends using these controls sparingly and removing them when no longer needed.

Also, be aware that there is a difference between putting DONT_TOUCH on a signal or on a level of hierarchy:

- If the attribute is placed on a signal, that signal is kept.

- If the attribute is placed on a level of hierarchy, the tool does not touch the boundaries of that hierarchy, and no constant propagation occurs through the hierarchy. However, optimizations inside that level of hierarchy are retained.

MAX_FANOUT

MAX_FANOUT forces the synthesis to replicate logic to meet a fanout limit. The tool is able to replicate logic, but not inputs or black boxes. Accordingly, if a MAX_FANOUT attribute is placed on a signal that is driven by a direct input to the design, the tool is unable to handle the constraint.

Take care to analyze the signals on which a MAX_FANOUT is placed. If a MAX_FANOUT is placed on a signal that is driven by a register with a DONT_TOUCH or drives signals that are in a different level of hierarchy when the DONT_TOUCH attribute is on that hierarchy, the MAX_FANOUT attribute will not be honored.

Synthesis appends the replicated cells with `_rep` for the first replication and subsequent replications are `_rep__0`, `_rep__1` and so on. These cells can be seen in the post synthesized netlist by selecting **Edit → Find** on cells.



IMPORTANT! Use MAX_FANOUT sparingly during synthesis. The `place_design` and `phys_opt_design` commands in the Vivado® tools perform placement-based replication, which is more effective than logical replication in synthesis. If a specific fanout is desired, it is often worth the time and effort to manually code the extra registers.

Block-Level Synthesis Strategy

With Vivado synthesis, you can use various strategies and global settings to customize how the design is synthesized. In most cases, these options are global and affect the entire design. You can use the block-level synthesis strategy to synthesize different levels of hierarchy with different global options in a top-down flow. This flow is faster and easier to perform than a bottom-up compile. You can set constraints for the full design rather than setting constraints for a lower level and then resetting for the top level.

Set the block-level synthesis strategy in the XDC file using the following syntax:

```
set_property BLOCK_SYNTH.<option_name> <value> [get_cells <instance_name>]
```

Where:

- `<option_name>` is the option to be set.
- `<value>` is the value to be assigned to the option.
- `<instance_name>` is the hierarchical instance on which to set the option.

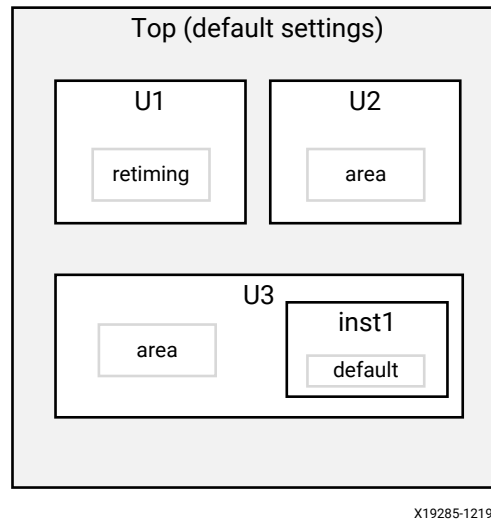
Note: These properties are always set on hierarchical instances. This allows modules or entities that are instantiated more than once to be synthesized with different options.

For example, you can set the following strategies in an XDC file:

```
set_property BLOCK_SYNTH.RETIMING 1 [get_cells U1]
set_property BLOCK_SYNTH.STRATEGY {AREA_OPTIMIZED} [get_cells U2]
set_property BLOCK_SYNTH.STRATEGY {AREA_OPTIMIZED} [get_cells U3]
set_property BLOCK_SYNTH.STRATEGY {DEFAULT} [get_cells U3/inst1]
```

Vivado synthesis is performed as shown in the following figure.

Figure 119: Block-Level Synthesis Strategy Example



You can set multiple BLOCK_SYNTH properties on the same instance to experiment with different options. For example:

```
set_property BLOCK_SYNTH.STRATEGY {ALTERNATE_ROUTABILITY} [get_cells inst]
set_property BLOCK_SYNTH.FSM_EXTRACTION {OFF} [get_cells inst]
```

When working with IP, you can use the block-level synthesis strategy as follows:

- If the IP is compiled globally, you can use this strategy on the top level of the IP.
- If the IP is out-of-context, you cannot use the strategy, because the IP appears as a black box. Instead, use global settings when compiling the IP.

Note: For more information on this feature and the supported strategies and options, see the *Vivado Design Suite User Guide: Synthesis* (UG901).

Moving Past Synthesis

Be sure that the netlist you obtained during synthesis is of good quality so that it does not create problems downstream. The following sections cover important items to check before proceeding with the rest of the implementation flow.

Reviewing and Cleaning DRCs

The `report_drc` command runs design rule checks (DRCs) to look for common design issues and errors. There are multiple rule decks. The default rule deck checks the following:

- Post-synthesis netlist
- I/O, BUFG, and other placement specific requirements
- Attributes and wiring on MGTs, IODELAYS, MMCMs, PLLs and other primitives



RECOMMENDED: Review and correct DRC violations as early as possible in the design process to avoid timing or logic related issues later in the implementation flow.



TIP: For DRC violations that can be safely ignored, you can use the waiver mechanism to waive the violations. For details, see this [link](#) in the Vivado Design Suite User Guide: Design Analysis and Closure Techniques (UG906).

Running Report Methodology

The Vivado tools provide a Methodology Report that specifically checks for compliance with methodology. The tools run different checks depending on the stage of the design process:

- RTL design: RTL lint-style checks
- Synthesized and implemented designs: Netlist, constraints, and timing checks

In Project Mode, the tools automatically run Report Methodology during implementation (`opt_design` or `route_design`) by default. To run these checks manually, use either of the following methods:

- At the Tcl prompt, open the design to be validated, and enter following Tcl command:
`report_methodology`
- To run these checks from the Vivado IDE, open the design to be validated, and select **Reports** → **Report Methodology**.



RECOMMENDED: To identify common design issues, run this report the first time you synthesize the design. Run this report again after significant module additions, constraint changes, or clocking circuit changes.

Note: For Xilinx®-supplied IP cores, the violations are already reviewed and checked.

Any violations are listed in the Methodology window, as shown in the following figure. If a specific methodology violation does not need to be fixed, make sure that you understand the violation and its implication clearly and why the violation does not negatively impact your design.



IMPORTANT! You must resolve all Critical Warnings and most Warnings to ensure good QoR, timing analysis accuracy, and reliable hardware stability are met. For methodology check violations that can be safely ignored, you can use the waiver mechanism to waive the violations. For details, see this [link](#) in the Vivado Design Suite User Guide: Design Analysis and Closure Techniques (UG906).

Note: Methodology checks related to RAMB and DSP primitive optional pipelining (SYNTH-6, SYNTH-11, SYNTH-12 and SYNTH-13) are not reported when setup timing is greater than 1 ns on all of the input or output paths for the primitives.

Figure 120: Methodology Window



For more information on running Report Methodology, see the Vivado Design Suite User Guide: System-Level Design Entry (UG895). Also, see this [link](#) in the Vivado Design Suite User Guide: Design Analysis and Closure Techniques (UG906).

Related Information

[Fixing Issues Flagged by report_methodology](#)

Reviewing the Synthesis Log

You must review the synthesis log files and confirm that all messages given by the tool match your expectations in terms of the design intent. Pay special attention to Critical Warnings and Warnings. In most cases, Critical Warnings need to be fixed for a reliable synthesis result.



CAUTION! If a message appears more than 100 times, the tool writes only the first 100 occurrences to the synthesis log file. You can change the limit of 100 through the Tcl command `set_param messaging.defaultLimit`.

Reviewing Timing Constraints

You must provide clean timing constraints, along with timing exceptions, where applicable. Bad constraints result in long compile time, performance issues, and hardware failures.



RECOMMENDED: Review all Critical Warnings and Warnings related to timing constraints which indicate that constraints have not been loaded or properly applied.

Related Information

[Organizing the Design Constraints](#)

Assessing Post-Synthesis Quality of Results

Report QoR Assessment combines logic level checks, utilization checks, and the most common clocking topology checks into one summary report that gives you an overall assessment of the design. This report helps you understand the severity of the timing closure issues. Xilinx recommends that you run this report on the synthesized design after any significant netlist update on a design with correct timing constraints.

Report QoR Assessment provides a score between 1 and 5 that indicates how likely the design is to close timing. The following table shows the definition of each score. Scores of 1 and 2 have no chance of meeting timing closure and a score of 3 is unlikely to close. Therefore, low scores mean more work in closing timing.

Table 4: Report QoR Assessment Scoring

Score	Meaning
1	Design is unlikely to complete the implementation flow
2	Design will complete the implementation flow but will not meet timing
3	Design is unlikely to meet timing
4	Design may meet timing
5	Design will easily meet timing

In the report, the detailed table provides information on the basis for the score. The thresholds in the detailed table are not absolute limits for the device. Instead, the thresholds indicate when timing closure might become increasingly difficult to achieve. After you exceed the threshold of any of these items, the difficulty in closing timing increases exponentially.

Plan to correct any items that are marked for review in the Report QoR assessment. Many of the items might be resolved automatically using Report QoR Suggestions.

Following Guidelines to Address Remaining Violations



IMPORTANT! Analyze timing post-synthesis to identify the major design issues that must be resolved before you move forward in the flow.

HDL changes tend to have the biggest impact on QoR. You are therefore better off solving problems before implementation to achieve faster timing convergence. When analyzing timing paths, pay special attention to the following:

- Most frequent offenders (that is, the cells or nets that show up the most in the top worst failing timing paths)
- Paths sourced by unregistered block RAMs
- Paths sourced by SRL
- Paths containing unregistered, cascaded DSP blocks
- Paths with large number of logic levels
- Paths with large fanout

Related Information

[Timing Closure](#)

Dealing with High Levels of Logic

Identifying long logic paths is useful to diagnose difficult QoR challenges. Estimated net delays post-synthesis are close to the best possible placement. To evaluate if a path with high logic-level delay is meeting timing, you can generate timing reports with no net delay. Timing closure cannot be achieved on paths that are still violating timing with no net delays.

Related Information

[Timing Closure](#)

Reviewing Utilization

It is important to review utilization for LUT, FF, block RAM, and DSP components independently. A design with low LUT/FF utilization might still experience placement difficulties if block RAM utilization is high. The `report_utilization` command generates a comprehensive utilization report with separate sections for all design objects.

Note: After synthesis, utilization numbers might change due to optimization later in the design flow.

Reviewing Clock Trees

This section discusses reviewing clock trees, including clock buffer utilization and clock tree topology.

Clock Buffer Utilization

The `report_clock_utilization` command provides details on clock primitive utilization. Observe the architecture clocking rules to avoid downstream placement issues. Invalid placement constraints or very high fanout for regional clock buffers might cause issues in the placer. For designs with very high clock buffer utilization, it might be necessary to lock the clock generators and some regional clock buffers to aid placement.

For some interfaces needing very tight timing relationship, it is sometimes better to lock specific resources for these signals which need very tight timing relationship, for example, source synchronous interfaces. In general, as a starting point for your design, lock only the I/Os unless there are specific reasons not to follow this approach as cited above.

Related Information

[Timing Closure](#)

Clock Tree Topology

When working with clock trees, follow these recommendations:

- Run the `report_clock_networks` command to show the clock network in detail tree view.
- Utilize clock trees in a way to minimize skew.
- For the outputs of PLLs and MMCMs, use the same clock buffer type to minimize skew.
- Look for unintended cascaded BUFG elements that can introduce additional delay, skew, or both.

Implementing the Design

Vivado Design Suite implementation includes all steps necessary to place and route the netlist onto the device resources, while meeting the design's logical, physical, and timing constraints. For additional information about implementation, refer to the following resources:

- *Vivado Design Suite User Guide: Implementation* ([UG904](#))
- [Vivado Design Suite QuickTake Video: Design Flows Overview](#)

Using Project Mode vs. Non-Project Mode

You can run implementation in Project Mode or Non-Project Mode. Project Mode provides the project infrastructure such as runs management, file sets management, reports generation, and cross probing. Non-Project Mode provides easy integration and is driven by a Tcl script which must explicitly call all the desired reports along the flow. For additional information about these modes, see this [link](#) in the *Vivado Design Suite User Guide: Design Flows Overview* ([UG892](#)).

Strategies

Strategies are used by the Vivado Design Suite to control both the tool options and the reports that are generated by synthesis and implementation runs in Project Mode. You can use the strategies to adjust the implementation goals and to control the reports that are generated. For more information on strategies, see the *Vivado Design Suite User Guide: Implementation* ([UG904](#)).



RECOMMENDED: Try the default strategy Vivado Implementation Defaults first. It provides a good trade-off between compile time and design performance.

Note: Strategies are tool and version specific. In some cases, strategies might require a longer compile time.

Directives

Directives provide different modes of behavior for the following implementation commands:

- `opt_design`
- `place_design`
- `phys_opt_design`
- `route_design`

Use the default directive initially. Use other directives when the design nears completion to explore the solution space for a design. You can specify only one directive at a time. For more information on directives, see the *Vivado Design Suite User Guide: Implementation* ([UG904](#)).

Iterative Flows

In Non-Project Mode, you can iterate between various optimization commands with different options. For example, you can run `phys_opt_design -directive AggressiveFanoutOpt` followed by `phys_opt_design -directive AlternateFlowWithRetiming` to run different physical synthesis optimizations on a placed design that does not meet timing.

Running `phys_opt_design` iteratively can provide timing improvement. The `phys_opt_design` command attempts to optimize the top timing problem paths. By running `phys_opt_design` iteratively, more critical paths can benefit from the optimization. Running `phys_opt_design` at the post-route stage reroutes any nets that might have been unrouted. Therefore, after running `phys_opt_design` at post-route, you do not need to explicitly run `route_design`.

Analyzing a Design at Different Stages Using Checkpoints

The Vivado Design Suite uses a physical design database to store placement and routing information. Design checkpoint files (`.dcp`) allow you to save (`write_checkpoint` command) and restore (`read_checkpoint` command) this physical database at key points in the design flow. Checkpoints are a snapshot of the design at a specific point in the flow. In Project Mode, the Vivado tools automatically generate design checkpoint files and store them in the implementation runs directory. These can be opened in a separate instance of the Vivado tools.

This design checkpoint file includes the following:

- Current netlist, including any optimizations made during implementation
- Design constraints
- Implementation results

Checkpoint designs can be run through the remainder of the design flow using Tcl commands. They cannot be modified with new design sources.

A few common examples for the use of checkpoints are:

- Saving results so you can go back and do further analysis on that part of the flow.
- Trying `place_design` using multiple directives and saving the checkpoints for each. This would allow you to select the `place_design` checkpoint with the best timing results for the subsequent implementation steps.

For more information on checkpoints, see the *Vivado Design Suite User Guide: Implementation (UG904)*.

Using Interactive Report Files

After opening a checkpoint, you can read in and immediately analyze generated reports in the Vivado IDE. To generate the reports, use the following reporting commands and append the `-rpx <filename.rpx>` option:

```
report_timing_summary
report_timing
report_power
report_methodology
report_drc
```

After the checkpoint is open, you can open the interactive report file using **Reports → Open Interactive Report**.

Note: In Project Mode, the interactive reports are generated and opened automatically.



RECOMMENDED: When a report is generated, there is a size limit on the RPX file. Therefore, Xilinx recommends using the `catch` command to prevent errors that might stop the flow. For example: `catch {report_timing_summary -rpx timing_summary.rpx -file timing_summary.rpt}`.

Using Incremental Implementation Flows

In the Vivado Design Suite, you can use incremental implementation to reuse existing placement and routing data, which reduces implementation compile time and produces more predictable results. When working with designs that have 95% or higher reuse, incremental place and route typically achieves at least a twofold improvement over normal place and route compile times while maintaining the WNS of the reference run. For more information, see this [link](#) in the *Vivado Design Suite User Guide: Implementation (UG904)*.



RECOMMENDED: *Incremental implementation is most useful during critical stages of the design cycle when changes to the flow scripts are difficult to make. Ensure that your flow scripts include incremental implementation early in the design cycle so you can enable incremental implementation during critical periods.*

Note: For further improvement in compile times and QoR, you can also use incremental synthesis.

Related Information

[Incremental Synthesis](#)

Incremental Implementation Flow Modes

Automatic Incremental Implementation Mode

You can use automatic incremental implementation mode to activate the incremental implementation flow but allow the Vivado tools to defer running incremental implementation until more is known about the reference checkpoint and the current design. When the `read_checkpoint` command is issued, the Vivado tools determine whether to run the implementation flow with the default flow algorithms or with the incremental flow algorithms. Automatic mode provides push-button ease of use, because the tools manage the reference design data for incremental implementation.

Note: The automatic incremental implementation mode is less aggressive than running the default incremental implementation flow and enables better maintenance of QoR when running the incremental implementation flow.

Project Mode

In Project Mode, the Vivado tools manage updating of the checkpoint as well as which algorithms to use. To enable the automatic incremental implementation mode in Project Mode, right-click an implementation run in the Design Runs window, select **Set incremental Compile → Automatically use the checkpoint from the previous run**.

The equivalent Tcl command is:

```
set_property AUTO_INCREMENTAL_CHECKPOINT 1 [get_runs <runName>]
```

Non-Project Mode

In Non-Project Mode, the Vivado tools manage which algorithms to use, but you must decide whether to update the checkpoint. To enable the automatic incremental implementation mode in Non-Project Mode, use the `-auto_incremental` option. Following is an example command:

```
read_checkpoint -incremental -auto_incremental <reference>.dcp
```

When updating the checkpoint, ensure that WNS did not degrade beyond acceptable limits by using the following command at the end of the implementation flow:

```
if {[get_property SLACK [get_timing_path -setup]] > -0.250} {
    file copy -force <postroute>.dcp <reference>.dcp
}
```

High and Low Reuse Modes

If you use the incremental implementation flow without enabling automatic incremental implementation mode, the reuse level triggers one of the following reuse modes:

- **High Reuse Mode:** High reuse mode is enabled when cell reuse is equal to or greater than 75%. This mode triggers incremental algorithms to be run and is the standard mode for incremental implementation.
- **Low Reuse Mode:** Low reuse mode is enabled when cell reuse is less than 75%. This mode reuses the cell placement of certain cells but runs the default algorithms. This mode can be effective when targeting block placement of DSPs, BRAMs, or a hierarchical cell.

Incremental Directives and Target WNS

You can use incremental directives to specify the target WNS for the implementation flow. The target WNS determines whether the implementation tools try to close timing or only try to achieve the same level of timing closure as the reference checkpoint. When the implementation flow uses the default algorithms, the incremental directive is ignored and the `place_design` and `route_design` directives are used.

The following table shows each incremental directive and the corresponding target WNS behavior.

Table 5: Incremental Directive and Target WNS Behavior

Incremental Directive	Target WNS Behavior
RuntimeOptimized	Same as the reference checkpoint
TimingClosure	0.000
Quick	Flow is not timing driven, and placement is driven by related logic

Note: Incremental directives replace directive mapping from previous releases.

Parallel Runs

To improve your chances of meeting timing using the default flow, it is common to implement many parallel runs, each with different placer directives. For incremental flows, the directive indicates whether to close or maintain timing. To achieve a spread of results, target the desired incremental directive with different reference checkpoints.

Compile Time Considerations

When using the `RuntimeOptimized` directive with automatic incremental implementation mode or high reuse mode, compile times can be reduced by half if 95% or more of the design is reused. As reuse declines, the benefit to compile time also declines. This is typically predictable unless changes impact critical paths.

When using the `TimingClosure` directive with automatic incremental implementation mode or high reuse mode, time is spent on running extra algorithms to close timing. Compile time can increase using this mode, especially when it is difficult to close timing or there are timing failures in a congested areas. When the reference checkpoint meets timing, compile time reduction is similar to using the `RuntimeOptimized` directive as described previously.

In low reuse mode, compile time is not predictable. When the place and route runs get closer to meeting timing, the Vivado tools might increase compile time to meet timing. In other cases, the Vivado tools might decrease compile time if existing placement and routing data is reused efficiently.

Opening the Synthesized Design

The first steps after synthesis are to read the netlist from the synthesized design into memory and apply design constraints. You can open the synthesized design in various ways, depending on the flow used. For more information, see this [link](#) in the *Vivado Design Suite User Guide: Implementation* (UG904).

Logic Optimization (`opt_design`)

Vivado Design Suite logic optimization optimizes the current in-memory netlist. Because this is the first view of the assembled design (RTL and IP blocks), the design can usually be further optimized. By default the `opt_design` command performs logic trimming, removing of cells with no loads, propagating constant inputs, and block RAM power optimization. It also optionally performs other optimizations such as remap, which combines LUTs in series into fewer LUTs to reduce path depth.

Optimization Analysis

The `opt_design` command generates messages detailing the results for each optimization phase. After optimization you can run `report_utilization` to analyze utilization improvements. To better analyze optimization results, rerun `opt_design` with the `-verbose` and `-debug_log` options for complete details on how each optimization affects the logic and how user constraints prevent some optimizations. For more information, see this [link](#) and this [link](#) in the *Vivado Design Suite User Guide: Implementation (UG904)*.

Placement (`place_design`)

The Vivado Design Suite placer engine positions cells from the netlist onto specific sites in the target Xilinx device.

Placement Analysis

Use the timing summary report after placement to check the critical paths.

- Paths with very large negative setup time slack may require that you check the constraints for completeness and correctness, or logic restructuring to achieve timing closure.
- Paths with very large negative hold time slack are most likely due to incorrect constraints or bad clocking topologies and should be fixed before moving on to route design.
- Paths with small negative hold time slack are likely to be fixed by the router. You can also run `report_clock_utilization` after `place_design` to view a report that breaks down clock resource and load counts by clock region.

For more information on placement, see this [link](#) in the *Vivado Design Suite User Guide: Implementation (UG904)*.

Related Information

[Timing Closure](#)

Physical Optimization (`phys_opt_design`)

Physical optimization is an optional step of the flow. It performs timing-driven optimization on the negative-slack paths of a design. Optimizations involve replication, retiming, hold fixing, and placement improvement. Because physical optimization automatically performs all necessary netlist and placement changes, `place_design` is not required after `phys_opt_design`.

Need for Physical Synthesis

To determine if a design would benefit from physical synthesis, evaluate timing after placement. Analyze failing paths for fanout. High fanout critical paths can benefit from fanout optimization. Additionally, high-fanout data, address and control nets of large RAM blocks involving multiple block RAMs that fail timing after `route_design` might benefit from Forced Net Replication. For more information on physical synthesis, see this [link](#) in the *Vivado Design Suite User Guide: Implementation* (UG904).

Routing (`route_design`)

The Vivado Design Suite router performs routing on the placed design and performs optimization on the routed design to resolve hold time violations. By default, the router performs optimization using a balance between runtime and design performance while alleviating congestion. Some router directives sacrifice runtime for better design performance and more aggressive congestion reduction. For more information on routing, see this [link](#) in the *Vivado Design Suite User Guide: Implementation* (UG904).

Route Analysis

Nets that are routed sub-optimally are often the result of incorrect timing constraints. Before you experiment with router settings, make sure that you have validated the constraints and the timing picture seen by the router. Validate timing and constraints by reviewing timing reports from the placed design before routing.

Common examples of poor timing constraints include cross-clock paths and incorrect multicycle paths causing route delay insertion for hold fixing. Congested areas can be addressed by targeted fanout optimization in RTL synthesis or through physical optimization. You can preserve all or some of the design hierarchy to prevent cross-boundary optimization and reduce the netlist density. Or you can use floorplan constraints to ease congestion.

Related Information

[Timing Closure](#)

Route Compile Time

You can use the `route_design -ultrathreads` option to reduce compile time at the expense of repeatability. This option gives the router extra freedom to execute multiple threads, which allows routing to finish faster but with slightly different results each time. The slack between identical subsequent runs differs by a fractional percentage, but the compile time savings are significant. Consider this option to reduce router compile time only if your environment does not require strictly repeatable results.

Design Closure

Design closure consists of meeting all system performance, timing, and power requirements, and successfully validating the functionality in hardware. During the design closure phase where you are starting to run the design through the implementation tools, both timing and power considerations should be your top priorities.

At this stage of design closure, estimation of design utilization, timing and power gain more accuracy. This presents an opportunity to reaffirm that the timing and power goals are achievable. To confirm the design can meet its requirements, Xilinx recommends conducting both a timing and power baseline. A timing baseline is largely about evaluating timing paths after accurate timing constraints have been defined. A power baseline needs to provide Vivado with the right toggle information to determine accurate dynamic power information.

By combining the analysis of power requirements and timing requirements, if one item is off significantly, a measure taken to resolve it can significantly impact the other. For example:

- An extreme measure might be necessary to meet a power budget such as scaling back features. This will make timing closure significantly easier as the part is less congested.
- A less extreme measure might involve adding logic to reduce switching. This might make timing closure more difficult, particularly if in a congested area of the die.

While many power saving items do not impact timing closure, it is possible that other items might make timing closure harder. Applying the required power saving techniques early will help you understand the true magnitude of the timing closure task.

Once you start to iterate from the baseline, you should recheck the power numbers when you make an improvement to timing. This ensures that you understand what change caused a regression. Generally, turning on wholesale power saving features early and then scaling back on individual items that are causing timing issues helps to strike the right balance of meeting design closure goals.

Conducting both power and timing analysis together and early in the design closure implementation phase will save engineering time and enable more accurate project planning. In addition, it creates time to allow engineering solutions to be explored than when this is realized later in the design cycle.



TIP: For more information on reports mentioned in this chapter, see *Vivado Design Suite User Guide: Design Analysis and Closure Techniques (UG906)*.

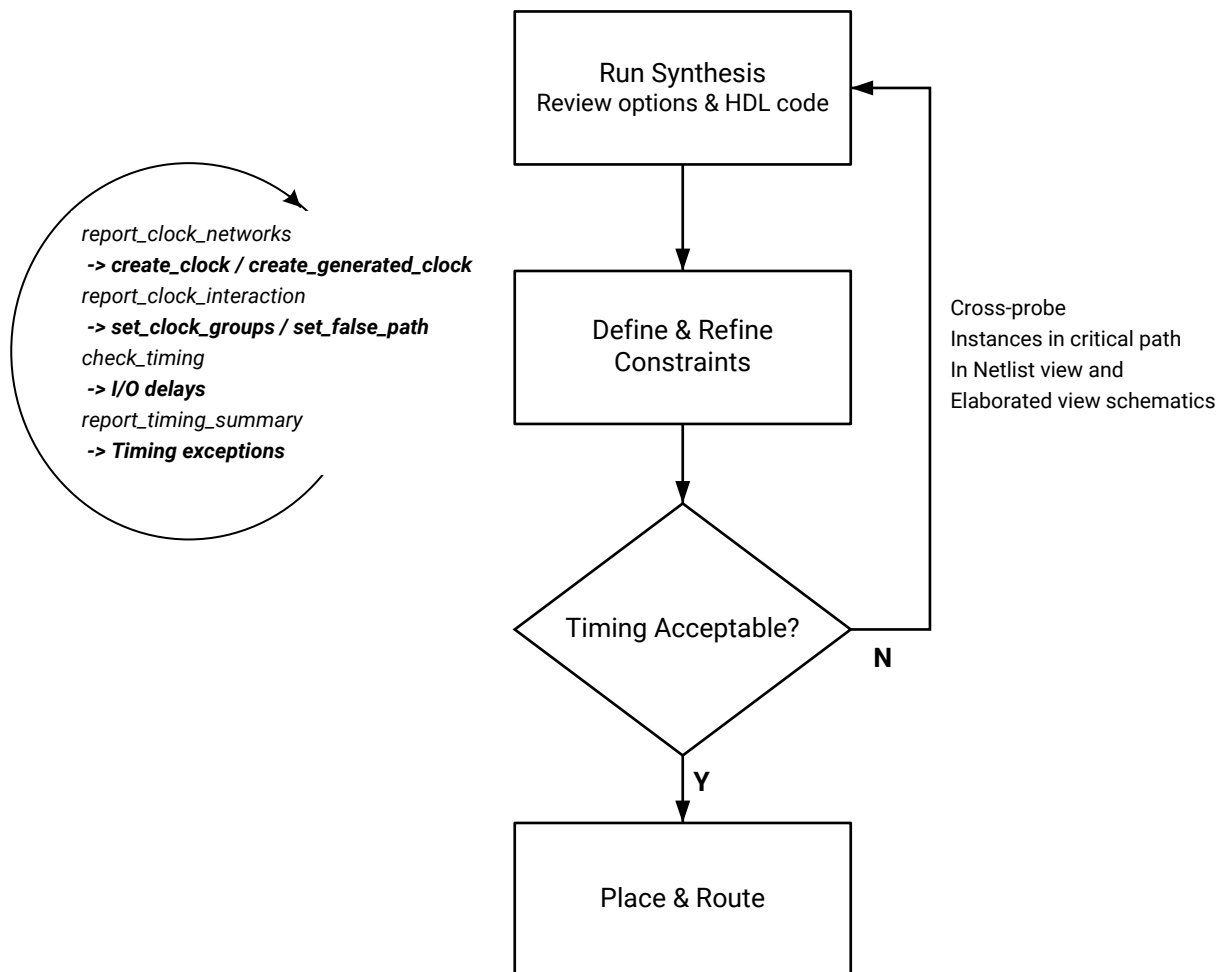


TIP: See the *UltraFast Design Methodology Timing Closure Quick Reference Guide (UG1292)* for a condensed version of the techniques described in this chapter, including running initial design checks, baselining the design, and resolving timing violations.

Timing Closure

Timing closure consists of the design meeting all timing requirements. It is easier to reach timing closure if you have the right HDL and constraints for synthesis. In addition, it is important to iterate through the synthesis stages with improved HDL, constraints, and synthesis options, as shown in the following figure.

Figure 121: Design Methodology for Rapid Convergence



X13422-121919

To successfully close timing, follow these general guidelines:

- When initially not meeting timing, evaluate timing throughout the flow.

- Focus on worst negative slack (WNS) of each clock as the main way to improve total negative slack (TNS).
- Review large worst hold slack (WHS) violations (<-1 ns) to identify missing or inappropriate constraints.
- Revisit the trade-offs between design choices, constraints, and target architecture.
- Know how to use the tool options and Xilinx® design constraints (XDC).
- Be aware that the tools do not try to further improve timing (additional margin) after timing is met.

The following sections provide recommendations for reviewing the completeness and correctness of the timing constraints using methodology design rule checks (DRCs) and baselining, identifying the timing violation root causes, and addressing the violations using common techniques.

Note: Timing results after synthesis use estimated net delays and not the actual routing delays. To get the final timing results, run implementation and then check the Report Timing Summary.

Understanding Timing Closure Criteria

Timing closure starts with writing valid constraints that represent how the design will operate in hardware. Review the Timing Summary report as described in the following sections.

Checking for Valid Constraints



POWER TIP: When you have a design run with clean timing, consider using the Create Runs command in the Vivado® IDE to run multiple strategies. Run the `report_power` command on each design with accurate switching activity XDC constraints to find the best run from both a timing and power perspective.

Review the Check Timing section of the Timing Summary report to quickly assess the timing constraints coverage, including the following:

- All active clock pins are reached by a clock definition.
- All active path endpoints have requirement with respect to a defined clock (setup/hold/recovery/removal).
- All active input ports have an input delay constraint.
- All active output ports have an output delay constraint.
- Timing exceptions are correctly specified.



CAUTION! Excessive use of wildcards in constraints can cause the actual constraints to be different from what you intended. Use the `report_exceptions` command to identify timing exception conflicts and to review the netlist objects, timing clocks, and timing paths covered by each exception.

In addition to `check_timing`, the Methodology report (TIMING and XDC checks) flags timing constraints that can lead to inaccurate timing analysis and possible hardware malfunction. You must carefully review and address all reported issues.

Note: When baselining the design, you must use all Xilinx IP constraints. Do not specify user I/O constraints, and ignore the violations generated by `check_timing` and `report_methodology` due to missing user I/O constraints. For more information on baselining the design, see [Baselining the Design](#).

Related Information

[Baselining the Design](#)

Checking for Positive Timing Slacks

The following timing metrics reflect the design timing score. Numbers must be positive to meet timing.

- Setup/Recovery (max delay analysis): $WNS > 0$ ns and $TNS = 0$ ns
- Hold/Removal (min delay analysis): $WHS > 0$ ns and $THS = 0$ ns
- Pulse Width: $WPWS > 0$ ns and $TPWS = 0$ ns

Understanding Timing Reports

The Timing Summary report provides high-level information on the timing characteristics of the design compared to the constraints provided. Review the timing summary numbers during signoff:

- **Total Negative Slack (TNS):** The sum of the setup/recovery violations for each endpoint in the entire design or for a particular clock domain. The worst setup/recovery slack is the worst negative slack (WNS).
- **Total Hold Slack (THS):** The sum of the hold/removal violations for each endpoint in the entire design or for a particular clock domain. The worst hold/removal slack is the worst hold slack (WHS).
- **Total Pulse Width Slack (TPWS):** The sum of the violations for each clock pin in the entire design or a particular clock domain for the following checks:
 - Minimum low pulse width
 - Minimum high pulse width
 - Minimum period
 - Maximum period
 - Maximum skew (between two clock pins of a same leaf cell)
- **Worst Pulse Width Slack (WPWS):** The worst slack for all pulse width, period, or skew checks on any given clock pin.

The Total Slack (TNS, THS or TPWS) only reflects the violations in the design. When all timing checks are met, the Total Slack is null.

The timing path report provides detailed information on how the slack is computed on any logical path for any timing check. In a fully constrained design, each path has one or several requirements that must all be met in order for the associated logic to function reliably.

The main checks covered by WNS, TNS, WHS, and THS are derived from the sequential cell functional requirements:

- **Setup time:** The time before which the new stable data must be available before the next active clock edge to be safely captured.
- **Hold requirement:** The amount of time the data must remain stable after an active clock edge to avoid capturing an undesired value.
- **Recovery time:** The minimum time required between the time the asynchronous reset signal has toggled to its inactive state and the next active clock edge.
- **Removal time:** The minimum time after an active clock edge before the asynchronous reset signal can be safely toggled to its inactive state.

A simple example is a path between two flip-flops that are connected to the same clock net.

After a timing clock is defined on the clock net, the timing analysis performs both setup and hold checks at the data pin of the destination flip-flop under the most pessimistic, but reasonable, operating conditions. The data transfer from the source flip-flop to the destination flip-flop occurs safely when both setup and hold slacks are positive.

For more information on timing analysis, see this [link](#) in the *Vivado Design Suite User Guide: Design Analysis and Closure Techniques (UG906)*.

Checking That Your Design is Properly Constrained

Before looking at the timing results to see if there are any violations, be sure that every synchronous endpoint in your design is properly constrained.

Run `check_timing` to identify unconstrained paths. You can run this command as a standalone command, but it is also part of `report_timing_summary`. In addition, `report_timing_summary` includes an Unconstrained Paths section where N logical paths without timing requirements are listed by the already defined source or destination timing clock. N is controlled by the `-max_path` option.

After the design is fully constrained, run the `report_methodology` command and review the TIMING and XDC checks to identify non-optimal constraints, which will likely make timing analysis not fully accurate and lead to timing margin variations in hardware. To identify and correct unrealistic target clock frequencies or setup path requirement, use the `report_qor_assessment` command.



IMPORTANT! To address missing or incomplete constraints, use the Timing Constraints wizard or see the Vivado Design Suite User Guide: Using Constraints (UG903).

Fixing Issues Flagged by `check_timing`

The `check_timing` Tcl command reports that something is missing or wrong in the timing definition. When reviewing and fixing the issues flagged by `check_timing`, focus on the most important checks first. Following are the checks listed from most important to least important.

No Clock and Unconstrained Internal Endpoints

This allows you to determine whether the internal paths in the design are completely constrained. You must ensure that the unconstrained internal endpoints are at zero as part of the Static Timing Analysis signoff quality review.

Zero unconstrained internal endpoints indicate that all internal paths are constrained for timing analysis. However, the correct value of the constraints is not yet guaranteed.

Generated Clocks

Generated clocks are a normal part of a design. However, if a generated clock is derived from a master clock that is not part of the same clock tree, this can cause a serious problem. The timing engine cannot properly calculate the generated clock tree delay. This results in erroneous slack computation. In the worst case situation, the design meets timing according to the reports but does not work in hardware.

Loops and Latch Loops

A good design does not have any combinational loops, because timing loops are broken by the timing engine. The broken paths are not reported during timing analysis or evaluated during implementation. This can lead to incorrect behavior in hardware, even if the overall timing requirements are met.

No Input/Output Delays and Partial Input/Output Delays

All I/O ports must be properly constrained.



RECOMMENDED: Start by validating baselining constraints and then complete the constraints with the I/O timing.

Multiple Clocks


Multiple clocks are usually acceptable. Xilinx recommends that you ensure that these clocks are expected to propagate on the same clock tree. You must also verify that the paths requirement between these clocks does not introduce tighter requirements than needed for the design to be functional in hardware.

If this is the case, you must use `set_clock_groups` or `set_false_path` between these clocks on these paths. Any time that you use timing exceptions, you must ensure that they affect only the intended paths.

Fixing Issues Flagged by `report_methodology`


The `report_methodology` command reports additional constraints and timing analysis issues, which you must carefully review before and after running the place and route tools. This section describes the main XDC and TIMING categories of checks, along with their relative impact on timing closure and hardware stability. You must focus on resolving the checks that impact timing closure first.

For more information on some of these checks, see this [link](#) in the *Vivado Design Suite User Guide: Design Analysis and Closure Techniques (UG906)*. Also, see the [Adoption Of The Methodology Report](#) blog series for more information on how `report_methodology` helps to resolve issues and save time.

 **IMPORTANT!** To increase visibility, the summary of the methodology violations is also included in the timing summary text report, because addressing these issues is critical for having proper signoff timing.

Methodology DRCs with Impact on Timing Closure

The DRCs shown in the following table flag design and timing constraint combinations that increase the stress on implementation tools, leading to impossible or inconsistent timing closure. These DRCs usually point to missing clock domain crossing (CDC) constraints, inappropriate clock trees, or inconsistent timing exception coverage due to logic replication. They must be addressed with highest priority.

 **IMPORTANT!** Carefully verify timing checks with a severity of Critical Warning.

For more information on timing methodology checks, see this [link](#) in the *Vivado Design Suite User Guide: Design Analysis and Closure Techniques (UG906)*.

Table 6: Timing Closure Methodology DRCs

Check	Severity	Description
TIMING-6	Critical Warning	No common clock between related clocks
TIMING-7	Critical Warning	No common node between related clocks
TIMING-8	Critical Warning	No common period between related clocks
TIMING-14	Critical Warning	LUT on the clock tree
TIMING-15	Warning	Large hold violation on inter-clock path
TIMING-16	Warning	Large setup violation
TIMING-30	Warning	Sub-optimal master source pin selection for generated clock
TIMING-31	Critical Warning	Inappropriate multicycle path between phase shifted clocks

Table 6: Timing Closure Methodology DRCs (cont'd)

Check	Severity	Description
TIMING-32, TIMING-33, TIMING-34, TIMING-37, TIMING-38, TIMING-39	Warning	Non-recommended bus skew constraint
TIMING-36	Critical Warning	Missing master clock edge propagation for generated clock
TIMING-42	Warning	Clock propagation prevented by path segmentation
TIMING-44 TIMING-45	Warning	Unreasonable user intra and inter-clock uncertainty
TIMING-48	Advisory	Max Delay Datapath Only constraint on latch input
TIMING-49	Critical Warning	Unsafe enable or reset topology from parallel BUFGCE_DIV
TIMING-50	Warning	Unrealistic path requirement between same-level latches
XDCB-3	Warning	Same clock mentioned in multiple groups in the same <code>set_clock_groups</code> command
XDCH-1	Warning	Hold option missing in multicycle path constraint
XDCV-1	Warning	Incomplete constraint coverage due to missing original object used in replication
XDCV-2	Warning	Incomplete constraint coverage due to missing replicated objects

Methodology DRCs with Impact on Signoff Quality and Hardware Stability

The DRCs shown in the following table do not usually flag issues that impact the ease of closing timing. Instead, these DRCs flag problems with timing analysis accuracy due to non-recommended constraints. Even when setup and hold slacks are positive, the hardware might not function properly under all operating conditions. Most checks refer to clocks not defined on the boundary of the design, clocks with unexpected waveform, missing timing requirements, or inappropriate CDC circuitry. For this last category, use the `report_cdc` command to perform a more comprehensive analysis.



IMPORTANT! Carefully verify timing checks with a severity of Critical Warning.

Table 7: Signoff Quality Methodology DRCs

Check	Severity	Description
TIMING-1, TIMING-2, TIMING-3, TIMING-4, TIMING-27	Critical Warning	Non-recommended clock source point definition
TIMING-5, TIMING-25, TIMING-19	Critical Warning	Unexpected clock waveform
TIMING-9, TIMING-10	Warning	Unknown or incomplete CDC circuitry
TIMING-11	Warning	Inappropriate <code>set_max_delay -datapath_only</code> command
TIMING-12	Warning	Clock Reconvergence Pessimism Removal disabled
TIMING-13, TIMING-23	Warning	Incomplete timing analysis due to broken paths
TIMING-17	Critical Warning	Non-clocked sequential cell

Table 7: Signoff Quality Methodology DRCs (cont'd)

Check	Severity	Description
TIMING-18, TIMING-20, TIMING-26	Warning	Missing clock or input/output delay constraints
TIMING-21, TIMING-22	Warning	Issues with MMCM compensation
TIMING-24	Warning	Overridden <code>set_max_delay -datapath_only</code> command
TIMING-29	Warning	Inconsistent pair of multicycle paths
TIMING-35	Critical Warning	No common node in paths with the same clock
TIMING-40, TIMING-43	Warning	Inappropriate clock topologies or requirements
TIMING-41	Warning	Invalid forwarded clock defined on an internal pin
TIMING-46	Warning	Multicycle path with tied CE pins
TIMING-47	Warning	False path or asynchronous clock group between synchronous clocks
TIMING-51	Critical Warning	No common phase between related clocks from parallel MMCMs or PLLs
TIMING-52	Critical Warning	No common phase between related clocks from Spread Spectrum MMCM

Other Timing Methodology DRCs

Other TIMING and XDC checks identify constraints that can incur higher run time, override existing constraints, or are highly sensitive to netlist names change. The corresponding information is useful for debugging constraints conflicts. You must pay particular attention to the TIMING-28 check (Auto-derived clock referenced by a timing constraint), because the auto-derived clock names can change when modifying the design source code and resynthesizing. In this case, previously defined constraints will not work anymore or will apply to the wrong timing paths.

Assessing the Maximum Frequency of the Design

You can define and assess the maximum frequency (FMAX) with a design that runs on a given architecture and speed grade by iteratively increasing the target clock frequency and re-running both synthesis and implementation until small setup slack violations ($WNS < 0$) are reported by timing analysis on the fully routed design. Xilinx recommends using the Default or PerformanceOptimized synthesis directives along with the Explore implementation directives and strategy to get the best achievable FMAX. In some cases, alternate strategies can show higher FMAX depending on the size of the design and the nature of the critical logic paths. For the implementation results with small setup violations, the maximum frequency is computed as follows:

- $F_{MAX} \text{ (MHz)} = \max(1000 / (T_i - WNS_i))$

Where:

- T_i is the target clock period (ns) used during the implementation run "i"

- WNS_i is the worst negative slack (ns) of the target clock used during the implementation run
";"

Additional important considerations:

- Using overly tight clock periods can lead to automatic effort reduction in the Vivado Implementation tools to avoid high compilation time due to unrealistic target and large timing violations. Use reasonably tight clock constraints instead.
- For designs with multiple clocks, you must proportionally decrease all synchronous clock periods until one of them starts failing timing after implementation (preferably the fastest clock or the clock with the most timing paths).

Note: The FMAX value is not explicitly provided in the `report_timing` or `report_timing_summary` report.

For a given design implementation, the maximum operating frequency on hardware across temperature and voltage ranges supported by the target device speed grade is defined by $1000/(T - WNS)$, with WNS positive or negative. When operating under nominal temperature and voltage conditions, typically in a lab environment, it is usually possible to operate the design at a slightly higher frequency.

Note: To increase the maximum frequency of the design, you can leverage the techniques described in this chapter or use Intelligent Design Runs.

Related Information

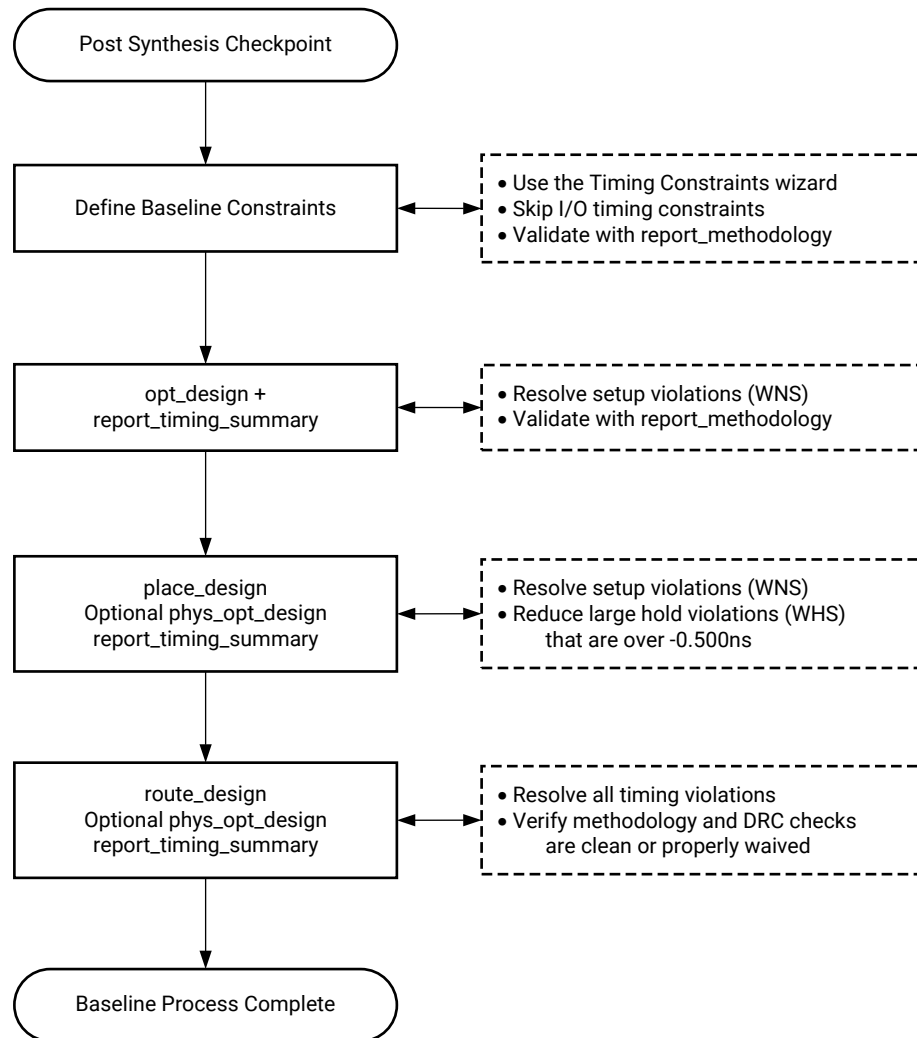
[Using Intelligent Design Runs](#)

Baselining the Design

Baselining is a process in which you create the simplest timing constraints and initially ignore I/O timing. After all clocks are completely constrained, all paths with start and endpoints within the design are automatically constrained. This provides an easy mechanism to identify internal device timing challenges, even while the design is evolving. Because the design might also have clock domain crossings, baseline constraints must also include the relationship among the specified clocks, including generated clocks.

When baselining the design, you must meet timing after each implementation step by analyzing and resolving timing challenges throughout the flow. First, you create simple and valid constraints to give a realistic picture of timing in the Vivado® implementation tools. Then, while iterating through different implementation steps, you solve timing violations before moving onto the next step. The following figure shows the baselining process.

Figure 122: Baselining the Design



X20037-021821

After baselining is complete, you can:

- Eliminate smaller timing violations
- Achieve full constraint coverage
- Individually baseline new modules before adding the modules to the top-level design



RECOMMENDED: Xilinx recommends that you create the baseline constraints very early in the design process, and plan any major change to the design HDL against these baseline constraints.

Defining Baseline Constraints

To create the simplest set of constraints, use a valid post-synthesis Vivado checkpoint without user timing constraints. With the checkpoint open, use the Timing Constraints wizard to define the constraints. The wizard guides you through the process of creating constraints in a structured manner.

Not all constraints need to be defined at this stage. The Vivado tools ignore I/O timing by default if there are no constraints. Therefore, you do not need to define I/O timing constraints at this point. Instead, define the I/O timing constraints later in the flow after the baselining process is complete.



TIP: When using the Timing Constraints wizard, deselect the suggested I/O timing constraints.

To get an accurate picture of internal timing in the device, define the following constraints:

- All clock constraints
- Clock domain crossings (CDC) constraints

CDC paths between synchronous clocks are safely timed by default, but you must use safe CDC circuitry and specify timing exceptions between asynchronous clocks.

After creating the constraints, identify the paths that cannot meet timing. Rewrite the corresponding RTL or relax the clock period.



IMPORTANT! All Xilinx IP and partner IP are delivered with specific XDC constraints that comply with the Xilinx constraints methodology. The IP constraints are automatically included during synthesis and implementation. You must keep the IP constraints intact when creating the baselining constraints.

If you do not use the Timing Constraints wizard to define the constraints, the following sections cover the steps you must take to define the baseline constraints manually.

Identifying Which Clocks Must Be Created

Begin by loading the post synthesized netlist or checkpoint into the Vivado IDE. In the Tcl Console, use the `reset_timing` command to ensure that all timing constraints are removed.

Use the `report_clock_networks` Tcl command to create a list of all the primary clocks that must be defined in the design. The resulting list of clock networks shows which clock constraints should be created. Use the **Timing Constraints Editor** to specify the appropriate parameters for each clock.

Verifying That No Clocks Are Missing

After the clock network report shows that all clock networks are constrained, you can begin verifying the accuracy of the generated clocks. Because the Vivado tools automatically propagate clock constraints through clock-modifying blocks, it is important to review the constraints that were generated. Use `report_clocks` to show which clocks were created with a `create_clock` constraint and which clocks were generated.

Note: MMCMs, PLLs, and clock buffers are clock-modifying blocks. For UltraScale™ devices, GTs are also clock-modifying blocks.

The `report_clocks` results show that all clocks are propagated. The difference between the primary clocks (created with `create_clock`) and the generated clocks is displayed in the attributes field:

- Clocks that are propagated (P) only are primary clocks.
- Clocks that are derived from other clocks are shown as both propagated (P) and generated (G).
- Clocks that are generated by a clock-modifying block are shown as auto-derived (A).
- Other attributes indicate that an auto-derived clock was renamed (R), a generated clock has an inverted waveform (I) relative to the incoming master clock, or a primary clock is virtual (V).

You can also create generated clocks using the `create_generated_clock` constraint. For more information, see the *Vivado Design Suite User Guide: Using Constraints (UG903)*.

Figure 123: Clock Report Shows the Clocks Generated from Primary Clocks

Attributes				
	P:	Propagated		
	G:	Generated		
	A:	Auto-derived		
	R:	Renamed		
	V:	Virtual		
	I:	Inverted		
Clock	Period(ns)	Waveform(ns)	Attributes	Sources
sysClk	10.000	{0.000 5.000}	P	{sysClk}
clkfbout	10.000	{0.000 5.000}	P,G,A	{clkgen/mmcm_adv_inst/CLKFBOUT}
cpuClk	20.000	{0.000 10.000}	P,G,A,R	{clkgen/mmcm_adv_inst/CLKOUT0}
wbClk_4	20.000	{0.000 10.000}	P,G,A	{clkgen/mmcm_adv_inst/CLKOUT1}
usbClk_3	10.000	{0.000 5.000}	P,G,A	{clkgen/mmcm_adv_inst/CLKOUT2}
phyClk0_2	10.000	{0.000 5.000}	P,G,A	{clkgen/mmcm_adv_inst/CLKOUT3}
phyClk1_1	10.000	{0.000 5.000}	P,G,A	{clkgen/mmcm_adv_inst/CLKOUT4}
fftClk_0	10.000	{0.000 5.000}	P,G,A	{clkgen/mmcm_adv_inst/CLKOUT5}



TIP: To verify that there are no unconstrained endpoints in the design, see the Check Timing report (*no_clock* category). The report is available from within the Report Timing Summary or by using the `check_timing` Tcl command.

Constraining Clock Domain Crossings

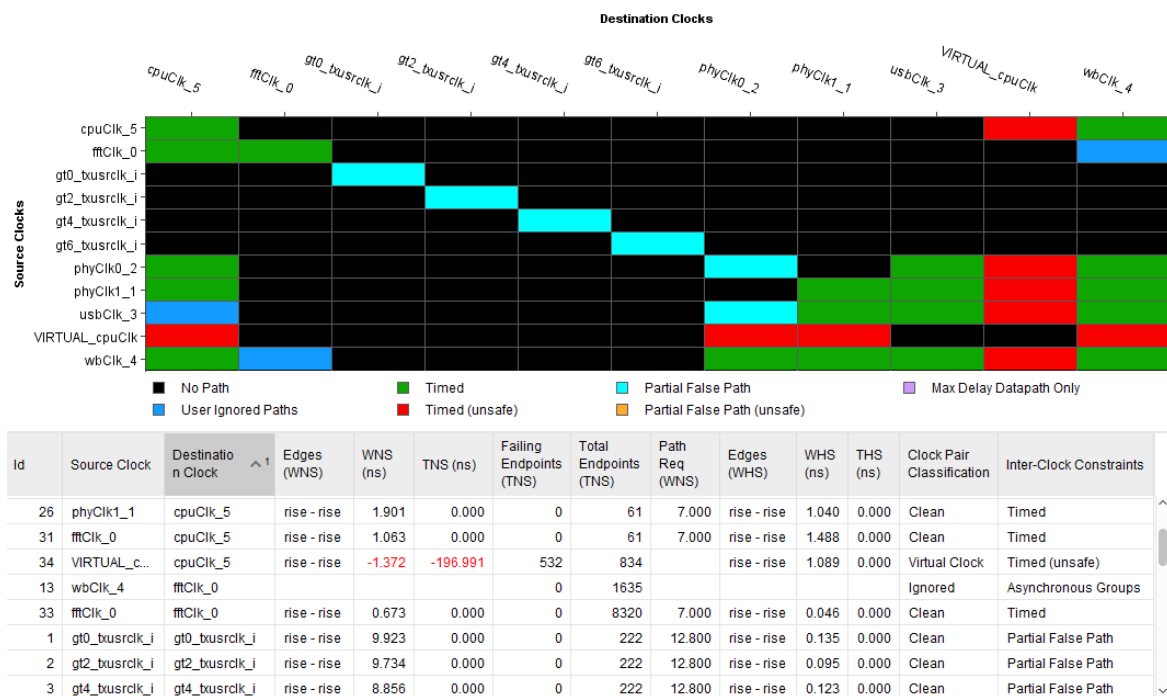
Upon verification of the clocking constraints, you must identify asynchronous and over-constrained clock domain crossing paths.

Note: This section does not explain how to properly cross clock region boundaries. Instead, it explains how to identify which crossings exist and how to constrain them.

Reviewing Clock Relationships

You can view the relationship between clocks using the `report_clock_interaction Tcl` command. The report shows a matrix of source clocks and destination clocks. The color in each cell indicates the type of interaction between clocks, including any existing constraints between them. The following figure shows a sample clock interaction report.

Figure 124: Sample Clock Interaction Report



The following table explains the meaning of each color in this report.

Table 8: report_clock_interaction Colors

Color	Label	Meaning	What Next
Black	No path	No interaction among these clock domains.	Primarily for information unless you expected these clock domains to be interacting.
Green	Timed	There is interaction among these clock domains, and the paths are getting timed.	Primarily for information unless you do not expect any interaction among the clock domains.

Table 8: report_clock_interaction Colors (cont'd)

Color	Label	Meaning	What Next
Cyan	Partial False Path	Some of the paths for the interacting domains are not being timed due to user exceptions.	Ensure that the timing exceptions are really desired.
Red	Timed (unsafe)	There is interaction among these clock domains, and the paths are being timed. However, the clocks appear to be independent (and hence, asynchronous).	Check whether these clocks are supposed to be declared as asynchronous, or whether they are supposed to be sharing a common primary source.
Orange	Partial False Path (unsafe)	There is interaction among these clock domains. The clocks appear to be independent (and hence, asynchronous). However, only some of the paths are not timed due to exceptions.	Check why some paths are not covered by timing exceptions.
Blue	User Ignored Paths	There is interaction among these clock domains, and the paths are not being timed due to clock groups or false path timing exceptions.	Confirm that these clocks are supposed to be asynchronous. Also, check that the corresponding HDL code is written correctly to ensure proper synchronization and reliable data transfer across clock domains.
Light blue	Max Delay Datapath Only	There is interaction among these clock domains, and the paths are getting timed through: <code>set_max_delay -datapath_only.</code>	Confirm that the clocks are asynchronous and that the specified delay is correct.

Before the creation of any false paths or clock group constraints, the only colors that appear in the matrix are black, red, and green. Because all clocks are timed by default, the process of decoupling asynchronous clocks takes on a high degree of significance. Failure to decouple asynchronous clocks often results in a highly over-constrained design.

Identifying Clock Pairs without Common Primary Clocks

The clock interaction report indicates whether or not each pair of interacting clocks has a common primary clock source. Clock pairs that do not share a common primary clock are frequently asynchronous to each other. Therefore, it is helpful to identify these pairs by sorting the columns in the report using the Common Primary Clock field. The report does not determine whether clock-domain crossing paths are or are not designed properly.

Use the `report_cdc` Tcl command for a comprehensive analysis of clock domain crossing circuitry between asynchronous clocks. For more information on the `report_cdc` command, see this [link](#) in the *Vivado Design Suite User Guide: Design Analysis and Closure Techniques (UG906)*. Also, see this [link](#) in the *Vivado Design Suite Tcl Command Reference Guide (UG835)*.

Identifying Tight Timing Requirements

For each clock pair, the clock interaction report also shows setup requirement of the worst path. Sort the columns by **Path Req (WNS)** to view a list of the tightest requirements in the design. Review these requirements to ensure that no invalid tight requirements exist.

The Vivado tools identify the path requirements by expanding each clock out to 1000 cycles, then determining where the closest, non-coincident edge alignment occurs. When 1000 cycles are not sufficient to determine the tightest requirement, the report shows Not Expanded, in which case you must treat the two clocks as asynchronous.

For example, consider a timing path that crosses from a 250 MHz clock to a 200 MHz clock:

- The positive edges of the 200 MHz clock are {0, 5, 10, 15, 20}.
- The positive edges of the 250 MHz clock are {0, 4, 8, 12, 16, 20}.

The tightest requirement for this pair of clocks occurs when the following is true:

- The 250 MHz clock has a rising edge at 4 ns.
- The next rising edge of the 200 MHz clock is at 5 ns.

This results in all paths timed from the 250 MHz clock domain into the 200 MHz clock domain being timed at 1 ns.

Note: The simultaneous edge at 20 ns is *not* the tightest requirement in this example, because the capture edge cannot be the same as the launch edge.

Because this is a fairly tight timing requirement, you must take additional steps. Depending on the design, one of the following constraints might be the correct way to handle these crossings:

- `set_clock_groups / set_false_path / set_max_delay -datapath_only`

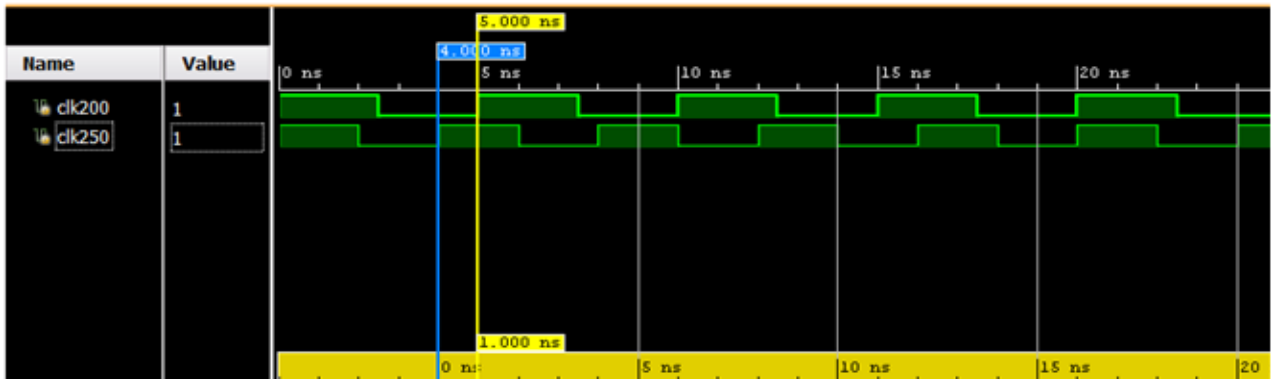
Use one of these constraints when treating the clock pair as asynchronous. Use the `report_cdc` Tcl command to validate that the clock domain crossing circuitry is safe.

- `set_multicycle_path`

Use this constraint when relaxing the timing requirement, assuming proper clock circuitry controls the launch and capture clock edges accordingly.

If nothing is done, the design might exhibit timing violations that cross these two domains. In addition, all of the best optimization, placement and routing might be dedicated to these paths instead of given to the real critical paths in the design. It is important to identify these types of paths before any timing-driven implementation step.

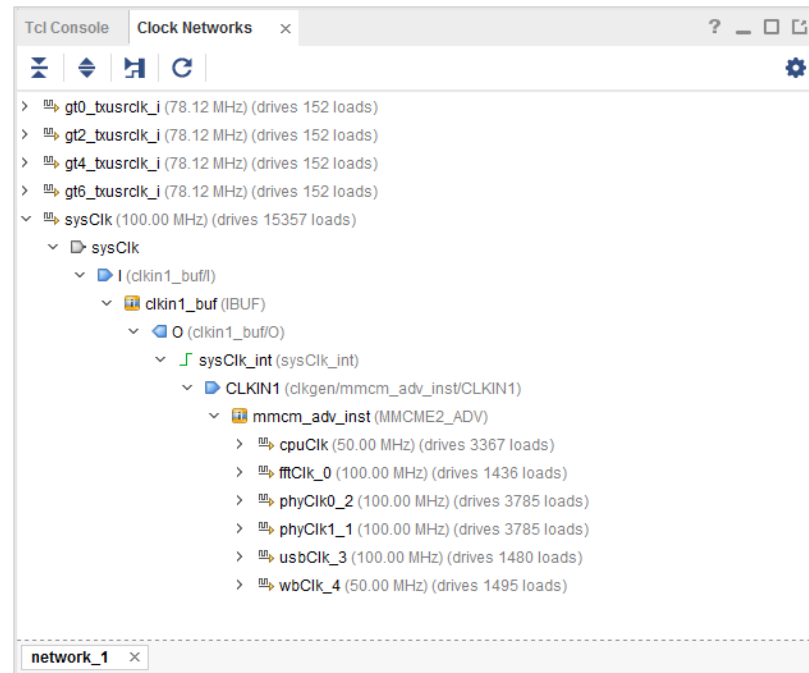
Figure 125: Clock Domain Crossing from 250 MHz to 200 MHz



Constraining Both Primary and Generated Clocks at the Same Time

Before any timing exceptions are created, it is helpful to go back to `report_clock_networks` to identify which primary clocks exist in the design. If all primary clocks are asynchronous to each other, you can use a single constraint to decouple the primary clocks from each other and to decouple their generated clocks from each other. Using the primary clocks in `report_clock_networks` as a guide, you can decouple each clock group and associated clocks as shown in the following figure.

Figure 126: Report Clock Networks



```

### Decouple asynchronous clocks
set_clock_groups -asynchronous \
-group [get_clocks sysClk -include_generated_clocks] \
-group [get_clocks gt0_txusrclk_i -include_generated_clocks] \
-group [get_clocks gt2_txusrclk_i -include_generated_clocks] \
-group [get_clocks gt4_txusrclk_i -include_generated_clocks] \
-group [get_clocks gt6_txusrclk_i -include_generated_clocks]
    
```

Limiting I/O Constraints and Timing Exceptions

Most timing violations are on internal paths. I/O constraints are not needed during the first baselining iterations, especially for I/O timing paths in which the launching or capturing register is located inside the I/O bank. You can add the I/O timing constraints after the design and other constraints are stable and the timing is nearly closed.



TIP: You can use the `config_timing_analysis -ignore_io_paths yes` Tcl command to ignore timing on all I/O paths during implementation and in reports that use timing information. You must manually enter this command before or immediately after opening a design in memory.

Based on recommendations of the RTL designer, timing exceptions must be limited and must not be used to hide real timing problems. Prior to this point, the false path or clock groups between clocks must be reviewed and finalized.

IP constraints must be entirely kept. When IP timing constraints are missing, known false paths might be reported as timing violations.

Evaluating Design WNS After Each Step

You must evaluate the design WNS after each synthesis and implementation step. If you are using the Tcl command line flow, you can easily incorporate `report_timing_summary` after each implementation step in your build script. If you are using the Vivado IDE, you can use simple `tcl.post` scripts to run `report_timing_summary` after each step. In both cases, when a significant degradation in WNS is noted, you must analyze the checkpoint immediately preceding that step.

In addition to evaluating the timing for the entire design after each implementation step, you can take a more targeted approach for individual paths to evaluate the impact of each step in the flow on the timing. For example, the estimated net delay for a timing path after the optimization step might differ significantly from the estimated net delay for the same path after placement. Comparing the timing of critical paths after each step is an effective method for highlighting where the timing of a critical path diverges from closure.

Post-Synthesis and Post-Logic Optimization

Estimated net delays are close to the best possible placement for all paths. To fix violating paths try the following:

- Change the RTL.
- Use different synthesis options.
- Add timing exceptions such as multicycle paths, if appropriate and safe for the functionality in hardware.

Pre- and Post-Placement

After placement, the estimated net delays are close to the best possible route, except for long and medium-to-high fanout nets, which use more pessimistic delays. In addition, congestion or hold fixing impact are not accounted for in the net delays at this point, which can make the timing results optimistic.

Clock skew is accurately estimated and can be used to review imbalanced clock trees impact on slack. You can estimate hold fixing by running min delay analysis. Large hold violations where the WHS is -0.500 ns or greater between slices, block RAMs or DSPs will need to be fixed. Small violations are acceptable and will likely be fixed by the router.

Note: Paths to/from dedicated blocks like the PCIe® block can have hold time estimates greater than -0.500 ns that get automatically fixed by the router. For these cases, check `report_timing_summary` after routing to verify that all corresponding hold violations are fixed.

Pre- and Post-Physical Optimization

Evaluate the need for running physical optimization to fix timing problems related to:

- Nets with high fanout (`report_high_fanout_nets` shows highest fanout non-clock nets)

- Nets with drivers and loads located far apart
- Digital signal processor (DSP) and block RAM with sub-optimal pipeline register usage

Pre- and Post-Route

Slack is reported with actual routed net delays except for the nets that are not completely routed. Slack reflects the impact of hold fixing on setup and the impact of congestion.

No hold violation should remain after route, regardless of the worst setup slack (WNS) value. If the design fails hold, further analysis is needed. This is typically due to very high congestion, in which case the router gives up on optimizing timing. This can also occur for large hold violations (over 4 ns) which the router does not fix by default. Large hold violations are usually due to improper clock constraints, high clock skew or, improper I/O constraints which can already be identified after placement or even after synthesis.

If hold is met ($WHS > 0$) but setup fails ($WNS < 0$), follow the analysis steps described in [Analyzing and Resolving Timing Violations](#).

Baselining and Timing Constraints Validation Procedure

The following procedure helps track your progress towards timing closure and identify potential bottlenecks:

1. Open the synthesized design.
2. Run `report_timing_summary -delay_type min_max`, and record the information shown in the following table.

Table 9: Timing Summary Report for Synthesized Design

	WNS	TNS	Num Failing Endpoints	WHS	THS	Num Failing Endpoints
Synth						

3. Open the post-synthesis `report_timing_summary` text report and record the `no_clock` section of `check_timing`.

Number of missing clock requirements in the design: _____

4. Run `report_clock_networks` to identify primary clock source pins/ports in the design. (Ignore `QPLLOUTCLK` and `QPLLOUTREFCLK` because they are pulse-width only checks.)

Number of unconstrained clocks in the design: _____

5. Run `report_clock_interaction -delay_type min_max` and sort the results by WNS path requirement.

Smallest WNS path requirement in the design: _____

- Sort the results of `report_clock_interaction` by WHS to see if there are large hold violations (>500 ps) after synthesis.

Largest negative WHS in the design: _____

- Sort results of `report_clock_interaction` by Inter-Clock Constraints and list *all* the clock pairs that show up as unsafe.

- Upon opening the synthesized design, how many Critical Warnings exist?

Number of synthesized design Critical Warnings: _____

- What types of Critical Warnings exist?

Record examples of each type.

- Run `report_high_fanout_nets -timing -load_types -max_nets 25`.

Number of high fanout nets *not* driven by FF: _____

Number of loads on highest fanout net *not* driven by FF: _____

Do any high fanout nets have negative slack? If yes, WNS = _____

- Implement the design. After each step, run `report_timing_summary` and record the information shown in the following table.

Table 10: Timing Summary Report

	WNS	TNS	Num Failing Endpoints	WHS	THS	Num Failing Endpoints
Opt						
Place						
Physopt						
Route						

- Run `report_exceptions -ignored` to identify if there are constraints that overlap in the design. Record the results.

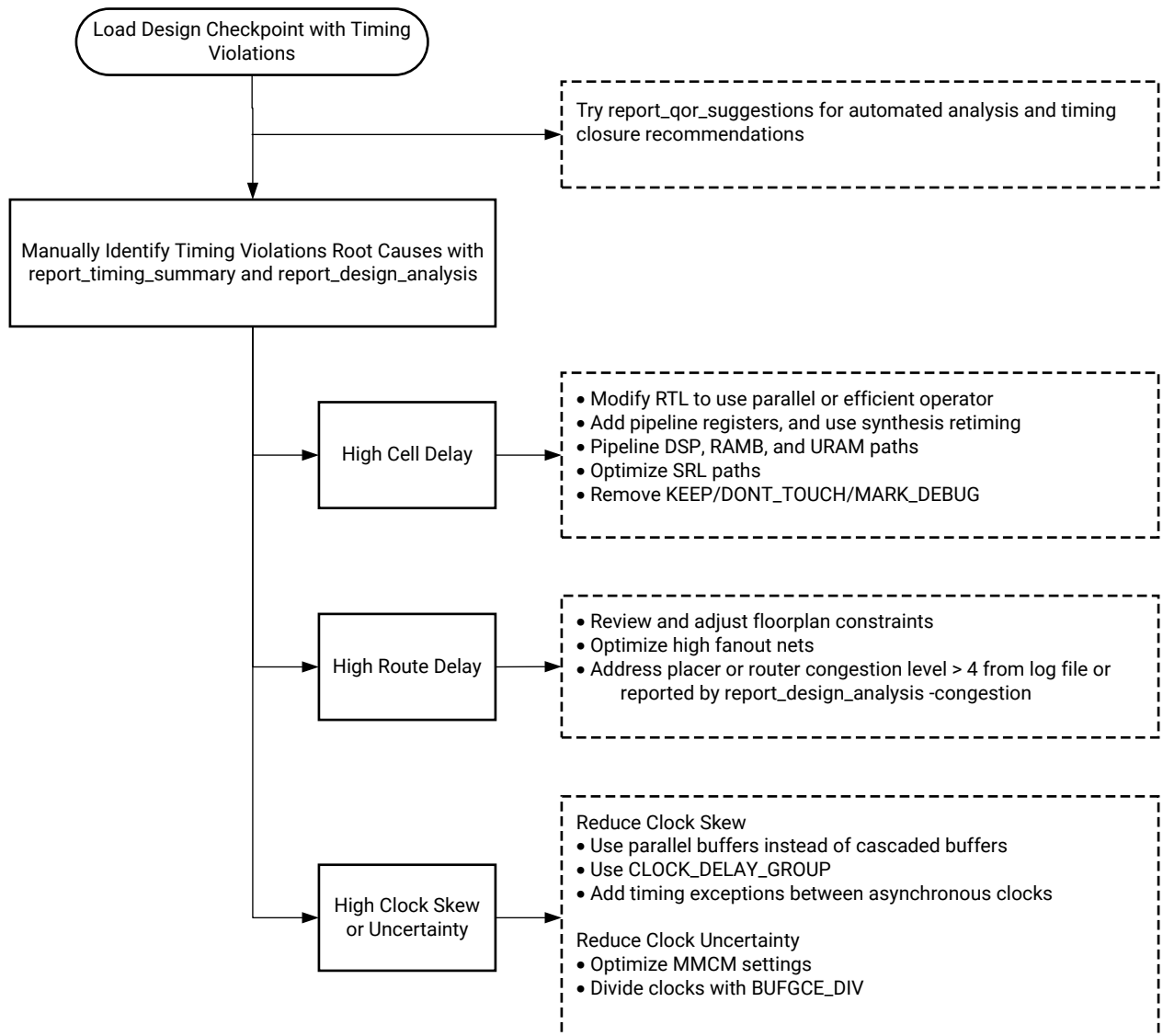
Analyzing and Resolving Timing Violations

The timing driven algorithms focus on the worst violations for each clock domain. When the worst violations are fixed, the tools typically resolve many of the less critical paths automatically when you rerun the implementation tools. You can assist in this process by focusing on resolutions that positively impact a high number of paths. For example, correcting suboptimal clocking typically impacts a high number of paths so Xilinx recommends focusing on these issues first before moving to path specific resolutions.

The Report QoR Suggestions command automatically identifies issues and orders suggestions based on criticality. You can determine the progress made towards timing closure by running the Report QoR Assessment command both before and after applying the suggestions. An increase in the QoR Assessment Score and a decrease in the detailed table marked for review indicates improvements.

The following figure shows the basic process for analyzing and resolving timing violations.

Figure 127: Analyzing and Resolving Timing Violations



X20036-110617

Identifying Timing Violations Root Cause

For setup, you must first analyze the worst violation of each clock group. A clock group refers to all intra, inter, and asynchronous paths captured by a given clock.

For hold, all violations must be reviewed as follows:

- Before routing, review only violations over 0.5 ns.
- After routing, start with the worst violation.

Reviewing Timing Slack

Several factors can impact the setup and hold slacks. You can easily identify each factor by reviewing the setup and hold slack equations when written in the following simplified form:

- **Slack (setup/recovery) = setup path requirement:**
 - datapath delay (max)
 - + clock skew
 - clock uncertainty
 - setup/recovery time
- **Slack (hold/removal) = hold path requirement:**
 - + datapath delay (min)
 - clock skew
 - clock uncertainty
 - hold/removal time

For timing analysis, clock skew is always calculated as follows:

- Clock Skew = destination clock delay - source clock delay (after the common node if any)

During the analysis of the violating timing paths, you must review the relative impact of each variable to determine which variable contributes the most to the violation. Then you can start analyzing the main contributor to understand what characteristic of the path influences its value the most and try to identify a design or constraint change to reduce its impact. If a design or constraint change is not practical, you must do the same analysis with all other contributors starting with the worst one. The following list shows the typical contributor order from worst to least.

For setup/recovery:

- **Datapath delay:** Subtract the timing path requirement from the datapath delay. If the difference is comparable to the (negative) slack value, then either the path requirement is too tight or the datapath delay is too large.
- **Datapath delay + setup/recovery time:** Subtract the timing path requirement from the datapath delay plus the setup/recovery time. If the difference is comparable to the (negative) slack value, then either the path requirement is too tight or the setup/recovery time is larger than usual and noticeably contributes to the violation.
- **Clock skew:** If the clock skew and the slack have similar negative values and the skew absolute value is over a few 100 ps, then the skew is a major contributor and you must review the clock topology.
- **Clock uncertainty:** If the clock uncertainty is over 100 ps, then you must review the clock topology and jitter numbers to understand why the uncertainty is so high.

For hold/removal:

- **Clock skew:** If the clock skew is over 300 ps, you must review the clock topology.
- **Clock uncertainty:** If the clock uncertainty is over 200 ps, then you must review the clock topology and jitter numbers to understand why the uncertainty is so high.
- **Hold/removal time:** If the hold/removal time is over a few 100 ps, you can review the primitive data sheet to validate that this is expected.
- **Hold path requirement:** The requirement is usually zero. If not, you must verify that your timing constraints are correct.

Assuming all timing constraints are accurate and reasonable, the most common contributors to timing violations are usually the datapath delay for setup/recovery timing paths, and skew for hold/removal timing paths. At the early stage of a design cycle, you can fix most timing problems by analyzing these two contributors. However, after improving and refining design and constraints, the remaining violations are caused by a combination of factors, and you must review all factors in parallel to identify which to improve.

See this [link](#) for more information on timing analysis concepts, and see this [link](#) for more information on timing reports (`report_timing_summary/report_timing`) in the *Vivado Design Suite User Guide: Design Analysis and Closure Techniques (UG906)*.

Using the Design Analysis Report

When timing closure is difficult to achieve or when you are trying to improve the overall performance of your application, you must review the main characteristics of your design after running synthesis and after any step of the implementation flow. The QoR analysis usually requires that you look at several global and local characteristics at the same time to determine what is suboptimal in the design and the constraints, or which logic structure is not suitable for the target device architecture and implementation tools. The `report_design_analysis` command gathers logical, timing, and physical characteristics in a few tables to simplify the QoR root cause analysis.

Note: The `report_design_analysis` command does not report on the completeness and correctness of timing constraints.



TIP: Run the Design Analysis Report in the Vivado IDE for improved visualization, automatic filtering, and convenient cross-probing.

The following sections only cover timing path characteristics analysis. The Design Analysis report also provides useful information about congestion and design complexity.

Related Information

Checking That Your Design is Properly Constrained

Analyze Path Characteristics

To report the 50 worst setup timing paths, you can use the Report Design Analysis dialog box in the Vivado IDE, or you can use the following command:

```
report_design_analysis -max_paths 50 -setup -name design_analysis_postRoute
```

The following figure shows an example of the Setup Path Characteristics table generated by this command. To see additional columns in the window, scroll horizontally.

Figure 128: Report Design Analysis Timing Path Characteristics Post-Route

Name	Requirement	Path Delay	Logic Delay	Net Delay	Clock Skew	Slack	Clock Relationship	Logic Levels	Routes	Logical Path	Start Point (
↳ Path 1	2.034	1.833	0.755	1.078	-0.292	-0.116	Safely Timed	1	2	URAM288_BASE LUT5 FDCE	clk_out5_s...
↳ Path 2	2.034	2.08	0.92	1.16	-0.008	-0.079	Safely Timed	6	5	FDCE CARRY8 CARRY8 LUT4 LUT6 LUT3 CARRY8 FDCE	clk_out5_s...
↳ Path 3	2.034	1.463	0.449	1.014	-0.234	-0.055	Safely Timed	4	3	FDCE LUT2 CARRY8 CARRY8 LUT2 RAMB18E2	clk_out5_s...
↳ Path 4	2.034	1.747	0.761	0.986	-0.302	-0.040	Safely Timed	1	1	URAM288_BASE LUT4 FDCE	clk_out5_s...
↳ Path 5	2.034	1.441	0.449	0.992	-0.234	-0.033	Safely Timed	4	3	FDCE LUT2 CARRY8 CARRY8 LUT2 RAMB18E2	clk_out5_s...
↳ Path 6	2.034	1.444	0.449	0.995	-0.231	-0.033	Safely Timed	4	3	FDCE LUT2 CARRY8 CARRY8 LUT2 RAMB18E2	clk_out5_s...
↳ Path 7	2.034	2.036	0.945	1.091	0	-0.028	Safely Timed	7	5	FDCE CARRY8 CARRY8 LUT4 LUT6 LUT3 CARRY8 CARRY8 FDCE	clk_out5_s...
↳ Path 8	2.034	1.963	0.949	1.014	-0.054	-0.008	Safely Timed	7	6	FDCE CARRY8 CARRY8 LUT4 LUT4 LUT6 LUT3 CARRY8 FDCE	clk_out5_s...

Following are tips for working with this table:

- Toggle between numbers and % by clicking the % (Show Percentage) button. This is particularly helpful to review proportion of cell delay and net delay.
- By default, columns with only null or empty values are hidden. Click the **Hide Unused** button to turn off filtering and show all columns, or right-click the table header to select which columns to show or hide.

From this table, you can isolate which characteristics are introducing the timing violation for each path:

- High logic delay percentage (Logic Delay)
 - Are there many levels of logic? (LOGIC_LEVELS)
 - Are there any constraints or attributes that prevent logic optimization? (DONT_TOUCH, MARK_DEBUG)
 - Does the path include a cell with high logic delay such as block RAM or DSP? (Logical Path, Start Point Pin Primitive, End Point Pin Primitive)
 - Is the path requirement too tight for the current path topology? (Requirement)
- High net delay percentage (Net Delay)
 - Are there any high fanout nets in the path? (High Fanout, Cumulative Fanout)
 - Are the cells assigned to several Pblocks that can be placed far apart? (Pblocks)
 - Are the cells placed far apart? (Bounding Box Size, Clock Region Distance)
 - For SSI technology devices, are there nets crossing SLR boundaries? (SLR Crossings)
 - Are one or several net delay values a lot higher than expected while the placement seems correct? Select the path and visualize its placement and routing in the Device window.
 - Is there a missing pipeline register in a block RAM or DSP cell? (Comb DSP, MREG, PREG, DOA_REG, DOB_REG)
- High skew (<-0.5 ns for setup and >0.5 ns for hold) (Clock Skew)
 - Is it a clock domain crossing path? (Start Point Clock, End Point Clock)
 - Are the clocks synchronous or asynchronous? (Clock Relationship)
 - Is the path crossing I/O columns? (IO Crossings)



TIP: For visualizing the details of the timing paths in the Vivado IDE, select the path in the table, and go to the Properties tab.

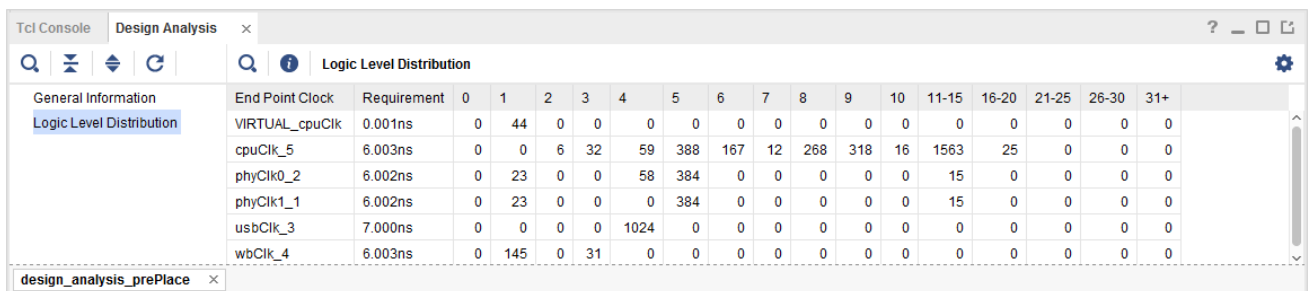
Review the Logic Level Distribution

The `report_design_analysis` command also generates a Logic Level Distribution table for the worst 1000 paths (default) that you can use to identify the presence of longer paths in the design. The longest paths are usually optimized first by the placer to meet timing, which will potentially degrade the placement quality of shorter paths. You must always try to eliminate the longer paths to improve the overall timing QoR. For this reason, Xilinx recommends reviewing the longest paths before placement.

The following figure shows an example of the Logic Level Distribution for a design where the worst 5000 paths include difficult paths with 17 logic levels while the clock period is 7.5 ns. Run the following command to obtain this report:

```
report_design_analysis -logic_level_distribution -logic_level_dist_paths
5000 -name design_analysis_prePlace
```

Figure 129: Report Design Analysis Timing Path Characteristics Pre-Place



End Point Clock	Requirement	0	1	2	3	4	5	6	7	8	9	10	11-15	16-20	21-25	26-30	31+
VIRTUAL_cpuClk	0.001ns	0	44	0	0	0	0	0	0	0	0	0	0	0	0	0	0
cpuClk_5	6.003ns	0	0	6	32	59	388	167	12	268	318	16	1563	25	0	0	0
phyClk0_2	6.002ns	0	23	0	0	58	384	0	0	0	0	0	15	0	0	0	0
phyClk1_1	6.002ns	0	23	0	0	0	384	0	0	0	0	0	15	0	0	0	0
usbClk_3	7.000ns	0	0	0	0	1024	0	0	0	0	0	0	0	0	0	0	0
wbClk_4	6.003ns	0	145	0	31	0	0	0	0	0	0	0	0	0	0	0	0

For logic levels above 10, you can use the `-min_level` and `-max_level` options to provide more distribution information for paths between the min and max level you identify. For example:

```
report_design_analysis -logic_level_distribution -min_level 16 -max_level 20
-logic_level_dist_paths 5000 -name design_analysis_1
```

Run the following command to generate the timing report of the longest paths:

```
report_timing -name longPaths -of_objects [get_timing_paths -setup -to [get_clocks
cpuClk_5] -max_paths 5000 -filter {LOGIC_LEVELS>=16 && LOGIC_LEVELS<=20}]
```

Based on what you find, you can improve the netlist by changing the RTL or using different synthesis options, or you can modify the timing and physical constraints.

Datapath Delay and Logic Levels

In general, the number of LUTs and other primitives in the path is most important factor in contributing to the delay. Because LUT delays are reported differently in different devices, separate cell delay and route delay ranges must be considered.

If the path delay is dominated by:

- Cell delay is >25% in 7 series devices and >50% in UltraScale devices.

Can the path be modified to be shorter or to use faster logic cells? See [Reducing Logic Delay](#).

- Route delay is >75% in 7 series devices and >50% in UltraScale devices.

Was this path impacted by hold fixing? You can determine this by running `report_design_analysis -show_all` and examining the **Hold Detour** column. Use the corresponding analysis technique.

- Yes - Is the impacted net part of a CDC path?
 - Yes - Is the CDC path missing a constraint?
 - No - Do the startpoint and endpoint of that hold-fixed path use a balanced clock tree? Look at the skew value.
- No - See the following information on congestion.

Was this path impacted by congestion? Look at each individual net delay, the fanout and observe the routing in the Device view with routing details enabled (post-route analysis only). You can also turn on the congestion metrics to see if the path is located in or near a congested area. Use the following analysis steps for a quick assessment or review [Reducing Net Delay Caused by Congestion](#) for a comprehensive analysis.

- Yes - For the nets with the highest delay value, is the fanout low (<10)?
 - Yes - If the routing seems optimal (straight line) but driver and load are far apart, the sub-optimal placement is related to congestion. Review [Addressing Congestion](#) to identify the best resolution technique.
 - No - Try to use physical logic optimization to duplicate the driver of the net. Once duplicated, each driver can automatically be placed closer to its loads, which will reduce the overall datapath delay. Review [Optimizing High Fanout Nets](#) for more details and to learn about alternate techniques.
- No - The design is spread out too much. Try one of the following techniques to improve the placement:
 - [Reducing Control Sets](#)
 - [Tuning the Compilation Flow](#)
 - [Considering Floorplan](#)

Clock Skew and Uncertainty

Xilinx devices use various types of routing resources to support most common clocking schemes and requirements such as high fanout clocks, short propagation delays, and extremely low skew. Clock skew affects any register-to-register path with either a combinational logic or interconnect between them.



RECOMMENDED: Run a design analysis report (`report_design_analysis`) to generate a timing report, which includes information on clock skew data. Verify that the clock nets do not contain excessive clock skew.

Clock skew in high performance clock domains (+300 MHz) can impact performance. In general, the clock skew should be no more than 500 ps. For example, 500 ps represents 15% of a 300 MHz clock period, which is equivalent to the timing budget of 1 or 2 logic levels. In cross domain clock paths the skew can be higher, because the clocks use different resources and the common node is located further up the clock trees. SDC-based tools time all clocks together unless constraints specify that they should not be (for example, `set_clock_groups`, `set_false_path`, or `set_max_delay -datapath_only`).

If the clock uncertainty is over 100 ps, then you must review the clock topology and jitter numbers to understand why the uncertainty is so high.

Related Information

[Reducing Clock Skew](#)

[Reducing Clock Uncertainty](#)

Reducing Logic Delay

Vivado implementation focuses on the most critical paths first, which often makes less difficult paths become critical after placement or after routing. Xilinx recommends identifying and improving the longest paths after synthesis or after `opt_design`, because it will have the biggest impact on timing and power QoR and will usually dramatically reduce the number of place and route iterations to reach timing closure.

Before placement, timing analysis uses estimated delays that correspond to ideal placement and typical clock skew. By using `report_timing`, `report_timing_summary`, or `report_design_analysis`, you can quickly identify the paths with too many logic levels or with high cell delays, because they usually fail timing or barely meet timing before placement. Use the methodology proposed in [Identifying Timing Violations Root Cause](#) to find the long paths which need to be improved before implementing the design.

Related Information

[Identifying Timing Violations Root Cause](#)

Optimizing Regular Fabric Paths

Regular fabric paths are paths between fabric registers or shift registers that traverse a mix of resources, such as LUTs. The `report_design_analysis` Timing Path Characteristics table provides the best logic path topology summary, where the following issues can be identified:

- Several small LUTs are cascaded

Mapping to LUTs is impacted by hierarchy, the presence of `KEEP_HIERARCHY`, `DONT_TOUCH`, or `MARK_DEBUG` attributes, or intermediate signals with some fanout (10 and higher). Run the `opt_design -remap` option or use the `AddRemap` or `ExploreWithRemap` directives to collapse smaller LUTs and reduce the number of logic levels. If `opt_design` is unable to optimize the longest paths due to a net fanout greater than one between the small LUTs, you can force the optimization by setting the `LUT_REMAP` property on the LUTs.

- Single CARRY cell is present in the path

CARRY primitives are most beneficial for timing QoR when cascaded. CARRY cells are more difficult to place than LUTs, and forcing synthesis to use LUTs rather than a single CARRY allows for better LUTs structuring and more flexible placement in many cases. Try the `FewerCarryChains` synthesis directive or the `PerfThresholdCarry` strategy (Project Mode only) to eliminate most single CARRY cells. Alternatively, use the `CARRY_REMAP` property to instruct `opt_design` to remap the tagged CARRY cells to LUTs.

Note: This optimization technique is automatically applied by the `report_qor_suggestions` Tcl command.

- Path ends at shift register (SRL)

Pull the first register out of the shift register by using the `SRL_STYLE` attribute in RTL. For details, see this [link](#) in the *Vivado Design Suite User Guide: Synthesis (UG901)*. Alternatively, you can use the `SRL_STAGES_TO_REG_INPUT` property applied prior to `opt_design` to implement the same optimization. For details, see this [link](#) in the *Vivado Design Suite User Guide: Implementation (UG904)*.

Note: This optimization technique is automatically applied by the `report_qor_suggestions` Tcl command.

- Path ends at a fabric register (FD) clock enable or synchronous set/reset

If the path ending at the data pin (D) has more margin and fewer logic levels, use the `EXTRACT_ENABLE` or `EXTRACT_RESET` attribute and set it to "no" on the signal in RTL. Alternatively, you can instruct `opt_design` to perform the same optimization by setting the `CONTROL_SET_REMAP` property on the registers to optimize.

Note: This optimization technique is automatically applied by the `report_qor_suggestions` Tcl command.



TIP: To cross-probe from a post-synthesis path to the corresponding RTL view and source code, see this [link](#) in the *Vivado Design Suite User Guide: Design Analysis and Closure Techniques (UG906)*.

Related Information

[Pushing the Logic from the Control Pin to the Data Pin](#)

Optimizing Paths with Dedicated Blocks and Macro Primitives

Paths from/to/between dedicated blocks and macro primitives (e.g., DSP, block RAM, or UltraRAM) need special attention because these primitives usually have the following timing characteristics:

- Higher setup/hold/clock-to-output timing arc values for some pins. For example, a block RAM has a clock-to-output delay around 1.5 ns without the optional output register and 0.4 ns with the optional output register. Review the data sheet of your target device series for complete details.
- Higher routing delays than regular FD/LUT connections.
- Higher clock skew variation than regular FD-FD paths.

Also, their availability and site locations are restricted compared to CLB slices, which usually makes their placement more challenging and often incurs some QoR penalty.

For these reasons, Xilinx recommends the following:

- Pipeline paths from and to dedicated blocks and macro primitives as much as possible.
- Restructure the combinational logic connected to these cells to reduce the logic levels by at least 1 or 2 cells if latency incurred by pipelining is a concern.
- Meet setup timing by at least 500 ps on these paths before placement.
- Replicate cones of logic connected to too many dedicated blocks or macro primitives if they need to be placed far apart.
- When the design has tight timing requirements to, within, or from a DSP block, run `opt_design -dsp_register_opt` to move registers to a more timing optimal position.

Note: Because timing is approximate during `opt_design`, you might also need to run `phys_opt_design -dsp_register_opt` to correct movements where timing was not accurately represented at the pre-placement stage.

Reducing Net Delay Caused by Physical Constraints

All designs come with a minimum set of physical constraints, especially for I/O location, and sometimes for clocking and logic placement. While I/O location cannot be modified when the design is ready for timing closure, physical constraints such as Pblocks and LOC must be analyzed. Use the `report_design_analysis` Timing Path Characteristics table to identify the presence of several Pblocks constraints on each critical path.

In the Vivado IDE Properties window, you can select the path in the Timing Path Characteristic table to review which Pblocks are constraining cells in the path. Consider removing one or several Pblock constraints if the constraints force logic spreading.

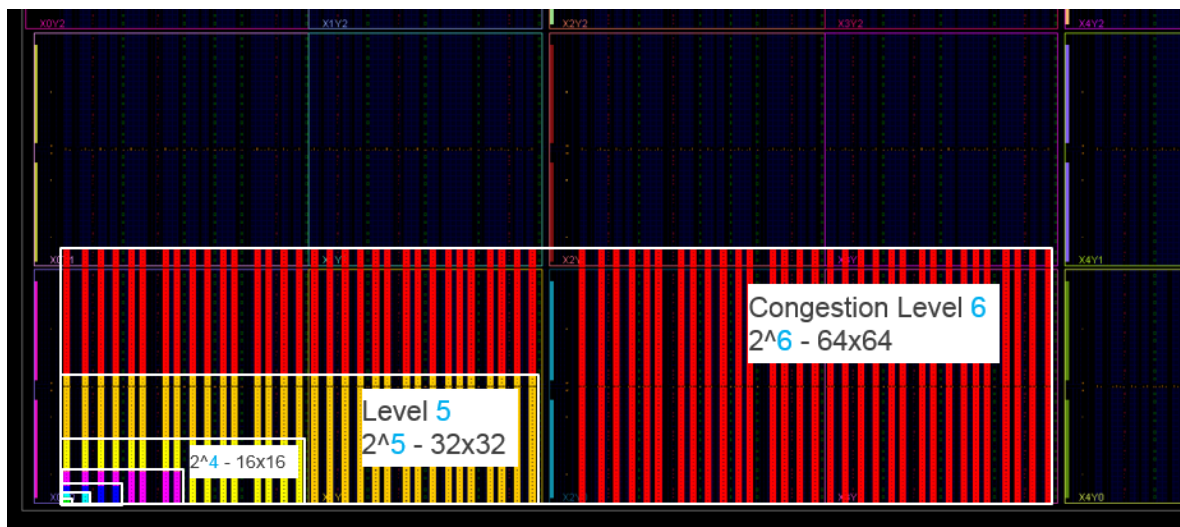
Reducing Net Delay Caused by Congestion

Device congestion can potentially lead to difficult timing closure if the critical paths are placed inside or next to a congested area or if the device utilization is high and the placed design is hardly routable. In many cases, congestion will significantly increase the router runtime. If a path shows routed delays that are longer than expected, analyze the congestion of the design and identify the best congestion alleviation technique.

Congestion Area and Level Definition

Xilinx device routing architecture comprises interconnect resources of various lengths in each direction: North, South, East, and West. A congested area is reported as the smallest square that covers adjacent interconnect tiles (INT_XnYm) or CLB tiles (CLE_M_XnYm) where interconnect resource utilization in a specific direction is close to or over 100%. The congestion level is the positive integer which corresponds to the side length of the square. The following figure shows the relative size of congestion areas on a Xilinx device versus clock regions.

Figure 130: Congestion Levels and Areas in the Device View



Congestion Level Ranges

When analyzing congestion, the level reported by the tools can be categorized as shown in the following table.

Note: Congestion levels of 5 or higher often impact QoR and always lead to longer router runtime.

Table 11: Congestion Level Ranges

Level	Area	Congestion	QoR Impact
1, 2	2x2, 4x4	None	None
3, 4	8x8, 16x16	Mild	Possible QoR degradation

Table 11: Congestion Level Ranges (cont'd)

Level	Area	Congestion	QoR Impact
5	32x32	Moderate	Likely QoR degradation
6	64x64	High	Difficulty routing
7, 8	128x128, 256x256	Impossible	Likely unroutable

Interconnect Congestion Level in the Device Window

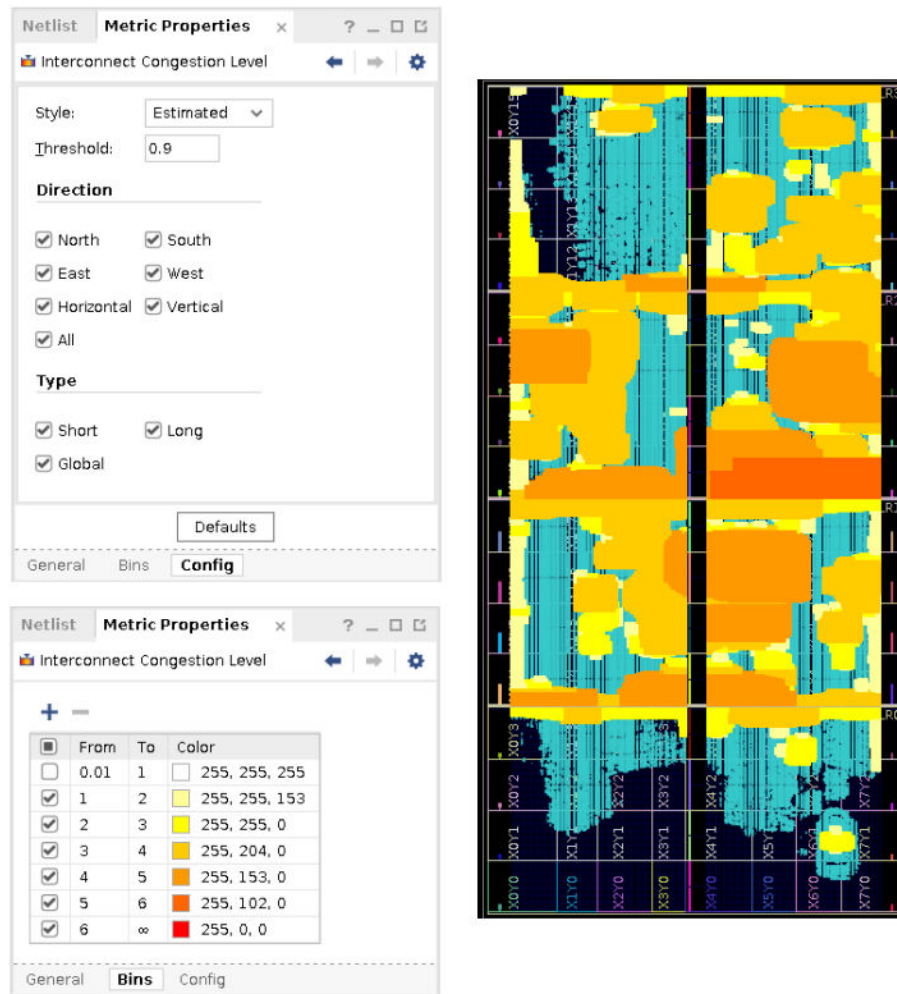
The Interconnect Congestion Level metric highlights the largest contiguous area in which routing resources are overused. By default, this metric is based on estimation, which is similar to the congestion level after initial routing. Actual routing can also be displayed if routing exists. After placement or after routing, you can display this congestion metric by right-clicking in the Device window and selecting **Metric → Interconnect Congestion Level**.

The Interconnect Congestion Level metric provides a quick visual overview of any congestion hotspots in the device. The following figure shows a placed design with several congested areas. This metric is based on the current interconnect demand and availability with a threshold of 0.9 (that is, 90% routing usage). The range is 0.1 to 0.9.

You can visualize congestion based on:

- Direction: North, South, East, West, Vertical, Horizontal
- Type: Short, Long, Global
- Style: Estimated, Routed, Mixed

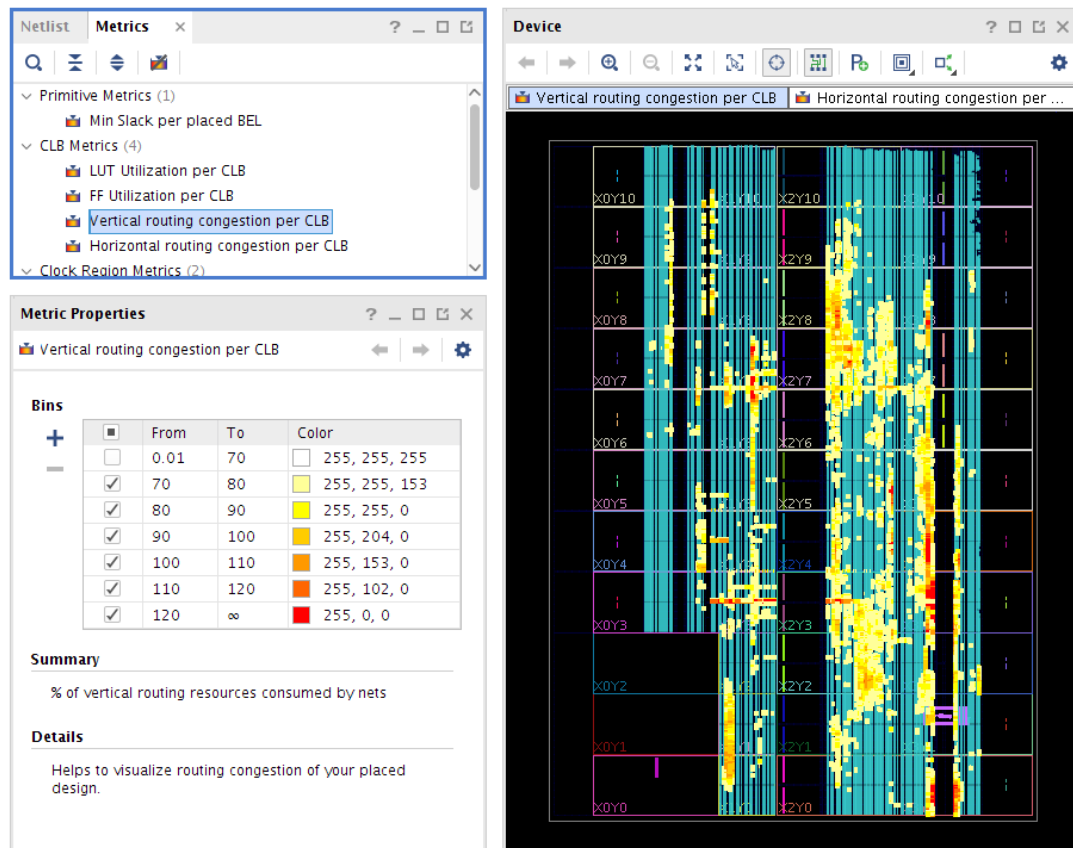
Figure 131: Example of Interconnect Congestion Level in the Device Window



Use the Routing Congestion per CLB, which is based on estimation and not actual routing. After placement or after routing, you can display this congestion metric by right-clicking in the Device window and selecting **Metric** → **Vertical and Horizontal Routing Congestion per CLB**. This provides a quick visual overview of any congestion hotspots in the device. The following figure shows a placed design with several congested areas due to high utilization and netlist complexity.

Note: Use this method for 7 series and UltraScale devices only.

Figure 132: Example of Congestion per CLB in the Device Window



Congestion in the Placer Log

The placer estimates congestion throughout the placement phases and spreads the logic in congested areas. This helps reducing the interconnect utilization to improve routability, and also the estimated versus routed delays correlation. However, when the congestion cannot be reduced due to high utilization or other reasons, the placer does not print congestion details but issues the following warning:

```
WARNING: [Place 46-14] The placer has determined that this design is highly congested
and may have difficulty routing. Run report_design_analysis -congestion for a
detailed report.
```

In that case the QoR is very likely impacted and it is prudent to address the issues causing the congestion before continuing on to the router. As stated in the message, use the `report_design_analysis` command to report the actual congestion levels, as well as identify their location and the logic placed in the same area.

Congestion in the Router Log

The router issues additional messages depending on the congestion level and the difficulty to route certain resources. The router also prints several intermediate timing summaries. The first one comes after routing all the clocks and usually shows WNS/TNS/WHS/TNS numbers similar to post-place timing analysis. The next router intermediate timing summary is reported after initial routing. If the timing has degraded significantly, the timing QoR has been impacted by hold fixing and/or congestion.

When congestion level is 4 or higher, the router prints an initial estimated congestion table which gives more details on the nature of the congestion:

- Global Congestion is similar to how the placer congestion is estimated and is based on all types of interconnects.
- Long Congestion only considers long interconnect utilization for a given direction.
- Short Congestion considers all other interconnect utilization for a given direction.

Any congestion area greater than 32x32 (level 5) will likely impact QoR and routability (highlighted in yellow in the table below). Congestion on Long interconnects increases usage of Short interconnects which results in longer routed delays. Congestion on Short interconnects usually induce longer runtimes and if their tile % is more than 5%, it will also likely cause QoR degradation (highlighted in red in the table below).

Figure 133: Initial Estimated Congestion Table

INFO: [Route 35-449] Initial Estimated Congestion

Direction	Global Congestion		Long Congestion		Short Congestion	
	Size	% Tiles	Size	% Tiles	Size	% Tiles
NORTH	16x16	1.95	32x32	1.68	32x32	11.58
SOUTH	8x8	1.90	16x16	2.00	32x32	9.23
EAST	8x8	0.93	2x2	0.20	32x32	9.14
WEST	8x8	1.37	2x2	0.15	32x32	14.50

During Global Iterations, the router first tries to find a legal solution with no overlap and also meet timing for both setup and hold, with higher priority for hold fixing. When the router does not converge during a global iteration, it stops optimizing timing until a valid routed solution has been found, as shown on the example below:

```
Phase 4.1 Global Iteration 0
Number of Nodes with overlaps = 1157522
Number of Nodes with overlaps = 131697
Number of Nodes with overlaps = 28118
Number of Nodes with overlaps = 10971
Number of Nodes with overlaps = 7324
WARNING: [Route 35-447] Congestion is preventing the router from routing all nets.
The router will prioritize the successful completion of routing all nets over timing
optimizations.
```

After a valid routed solution has been found, timing optimizations are re-enabled.

The route also flags CLB routing congestion and provides the name of the top most congested CLBs. An Info message is issued and the congested CLBs and nets are written to the text file listed in the message body. You can examine the text file for the list of CLB tiles and congested nets that are involved in the CLB pin-feed congestion, and use the congestion alleviation techniques listed in the Addressing Congestion section to resolve the CLB congestion before routing the design.

```
INFO: [Route 35-443] CLB routing congestion detected. Several CLBs have high routing
utilization, which can impact timing closure. Congested CLBs and Nets are dumped in:
iter_200_CongestedCLBsAndNets.txt
```



TIP: Localized CLB routing congestion can lead to routing failures even when the reported congestion levels for Global, Long, or Short congestion are within the acceptable range (less than 5). Look for the message above and in generated text files for localized congestion hotspots.

Finally, when the router cannot find a legally routed solution, several Critical Warning messages, as shown below, indicate the number of nets that are not fully routed and the number of interconnect resources with overlaps.

```
CRITICAL WARNING: [Route 35-162] 44084 signals failed to route due to routing
congestion. Please run report_route_status to get a full summary of the design's
routing.
...
CRITICAL WARNING: [Route 35-2] Design is not legally routed. There are 91566 node
overlaps.
```



TIP: During routing, nets are spread around the congested areas, which usually reduces the final congestion level reported in the log file when the design is successfully routed.

Report Design Analysis Congestion Report

To help you identify congestion, the Report Design Analysis command allows you to generate a congestion report that shows the congested areas of the device and the name of design modules present in these areas. The congestion tables in the report show the congested area seen by the placer and router algorithms. The following figure shows an example of the congestion table.

Figure 134: Congestion Table

General Information	Window	Direction	Congestion Level	Congestion	Cell Names			Combined LUTs	LUT6	LUT5	Flop	MUXF	RAMB
					Top Cell 1	Top Cell 2	Top Cell 3						
Placed Maximum	Window 1	North	4	120	inst_1022144/inst_1022144/inst_102inst_1022144/inst_1022134/inst_1018559/inst_990436 (10%)			26%	37%	22%	43%	0%	NA
Placed Tile Based (V)	Window 2	East	4	107	inst_1022144/inst_cv_33 (17%)			26%	29%	7%	60%	1%	50%
Placed Tile Based (H)	Window 3	South	4	131	inst_1022144/inst_1022144/inst_102inst_1022144/inst_1022134/inst_1018559/inst_990436/inst_979691 (6%)			32%	38%	18%	54%	0%	50%
	Window 4	West	2	143	inst_1022144/inst_1022144/inst_102inst_1022144/inst_1022134/inst_1018559/inst_887992 (9%)			84%	40%	1%	60%	0%	NA

The Placed Maximum, Initial Estimated Router Congestion, and Router Maximum congestion tables provide information on the most congested areas in the North, South, East, and West direction. When you select a window in the table, the corresponding congested area is highlighted in the Device window.

The tables show the congestion at different stages of the design flow:

- **Placed Maximum:** Shows congestion based on the location of the cells and a model of routing.
- **Initial Estimated Router Congestion:** Shows congestion after a quick router iteration. This is the most useful stage to analyze congestion because it gives an accurate picture of congestion due to placement.
- **Router Maximum:** Shows congestion after the router has worked extensively to reduce congestion.

The Congestion percentages in the Congestion Table show the routing utilization in the congestion window. The top three hierarchical cells located in the congested window are listed and can be selected and cross-probed to the Device window or Schematic window. The cell utilization percentage in the congestion window is also shown.

With the hierarchical cells present in the congested area identified, you can use the congestion alleviating techniques discussed later in this guide to try reducing the overall design congestion.

For more information on generating and analyzing the Report Design Analysis Congestion report, see this [link](#) in the *Vivado Design Suite User Guide: Design Analysis and Closure Techniques (UG906)*.

Report Design Analysis Complexity Report

The Complexity Report shows the Rent Exponent, Average Fanout, and distribution per type of leaf cells for the top-level design and/or for hierarchical cells. The Rent exponent is the relationship between the number of ports and the number of cells of a netlist partition when recursively partitioning the design with a min-cut algorithm. It is computed with similar algorithms as the ones used by the placer during global placement. Therefore, it can provide a good indication of the challenges seen by the placer, especially when the hierarchy of the design matches well the physical partitions found during global placement.

A design with higher Rent exponent corresponds to a design where the groups of highly connected logic also have strong connectivity with other groups. This usually translates into a higher utilization of global routing resources and an increased routing complexity. The Rent exponent provided in this report is computed on the unplaced and unrouted netlist. After placement, the Rent exponent of the same design can differ as it is based on physical partitions instead of logical partitions.

Report Design Analysis runs in Complexity Mode when you do either of the following:

- Check the Complexity option in the Report Design Analysis dialog box Options tab.
- Execute the `report_design_analysis` Tcl command with the `-complexity` option.

The following figure shows the Complexity Report.

Figure 135: Complexity Report

Instance	Module	Rent	Average Fanout	Total Instances	LUT1	LUT2	LUT3	LUT4	LUT5	LUT6	Memory LUT	DSP	RAMB	MUXF	URAM
cv_33	cv_33	0.41	2.91	1131310	0.7%	11.9%	18.4%	15.7%	17.2%	36.1%	22141	125	913	23685	82
Inst_1022144 (cv_71)	cv_71	0.42	2.86	1011347	0.6%	11.7%	18.6%	15.8%	17.2%	36.1%	17807	122	810	21452	82
Inst_1029467 (cv_13905)	cv_13905	0.37	3.44	7236	0.7%	11.9%	10.2%	24.6%	16.2%	36.4%	1472	0	0	3	0
Inst_1036789 (cv_13934)	cv_13934	0.41	3.44	7236	0.7%	11.9%	10.2%	24.6%	16.2%	36.4%	1472	0	0	3	0
Inst_1051863 (cv_13963)	cv_13963	0.47	3.01	61384	1.6%	9.1%	19.6%	13.4%	17.7%	38.6%	22	0	68	1892	0
Inst_1052499 (cv_13973)	cv_13973	0.63	3.16	1366	0.7%	13.3%	12.6%	9.6%	27.5%	36.3%	8	1	4	9	0
Inst_1055086 (cv_13982)	cv_13982	0.42	2.64	2525	2.3%	25.4%	12.7%	21.7%	11.4%	26.4%	4	0	6	0	0
Inst_1059242 (cv_13998)	cv_13998	0.25	2.32	4076	2.7%	39.1%	18.6%	16.2%	9.7%	13.6%	0	0	12	0	0
Inst_1071723 (cv_14030)	cv_14030	0.5	3.3	10914	0.4%	12.2%	15.4%	11.7%	16.4%	43.8%	912	2	8	204	0
Inst_1075799 (cv_14081)	cv_14081	0.41	3.1	4001	0.2%	18.3%	10.7%	14.6%	21.2%	35.0%	128	0	0	72	0
Inst_1077925 (cv_14087)	cv_14087	0.67	3.43	2067	0.2%	12.5%	15.1%	10.1%	14.1%	48.1%	256	0	0	18	0
Inst_130 (cv_39)	cv_39	0.16	4.2	17216	2.6%	26.4%	22.0%	13.6%	13.1%	22.4%	60	0	0	4	0

The following table shows the typical ranges for the Rent Exponent.

Table 12: Rent Exponent Ranges

Range	Meaning
0.0 to 0.65	This range is low to normal.
0.65 to 0.85	This range is high, especially when the total number of instances is above 15,000.
Above 0.85	This range is very high, indicating that the design might fail during implementation if the number of instances is also high.

The following table shows the typical ranges for the Average Fanout.

Table 13: Average Fanout Ranges

Range	Meaning
Below 4	This range is normal.

Table 13: Average Fanout Ranges (cont'd)

Range	Meaning
4 to 5	This range is high, indicating that placing the design without congestion might be difficult. When using SSI technology devices, if the total number of instances is above 100,000, it might be difficult for the placer to find a solution that fits in 1 SLR or is spread over 2 SLRs.
Above 5	This range is very high, indicating that the design might fail during implementation.

You must treat high Rent exponents and high Average Fanouts for larger modules with higher importance. Smaller modules, especially under 15,000 total instances, can have high Rent exponent and high Average Fanout and still be easy to place and route successfully. Therefore, you must review the Total Instances column along with the Rent exponent and Average Fanout.



TIP: Top-level modules do not necessarily have high complexity metrics even though some of the lower-level modules have high Rent exponents and high Average Fanouts. Use the `-hierarchical_depth` option to refine the analysis to include the lower-level modules.

For more information on generating and analyzing the Report Design Analysis Complexity report see this [link](#) in the *Vivado Design Suite User Guide: Design Analysis and Closure Techniques (UG906)*.

Reducing Clock Skew

To meet requirements such as high fanout clocks, short propagation delays, and low clock skew, Xilinx devices use dedicated routing resources to support the most common clocking schemes. Clock skew can severely reduce timing budget on high frequency clocks. Clock skew can also add excessive stress on implementation tools to meet both setup and hold when the device utilization is high.

The clock skew is typically less than 300 ps for intra-clock timing paths and less than 500 ps for timing paths between balanced synchronous clocks. When crossing resource columns, clock skew shows more variation, which is reflected in the timing slack and optimized by the implementation tools. For timing paths between unbalanced clock trees or with no common node, clock skew can be several nanoseconds, making timing closure almost impossible.

To reduce clock skew:

1. Review all clock relationships to ensure that only synchronous clock paths are timed and optimized.
2. Review the clock tree topologies and placement of timing paths impacted by higher clock skew than expected, as described in the following sections.
3. Identify the possible clock skew reduction techniques, as described in the following sections.

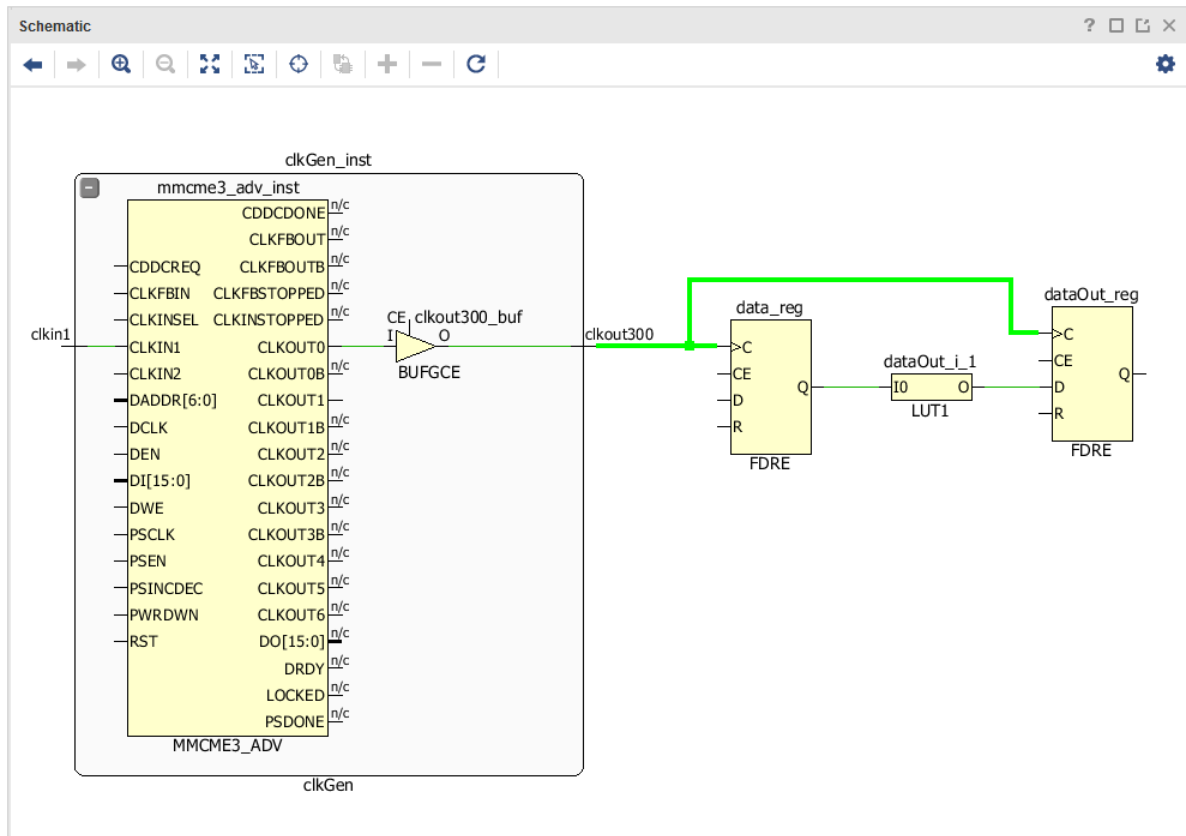
Related Information

[Defining Clock Groups and CDC Constraints](#)

Using Intra-Clock Timing Paths

Timing paths with the same source and destination clocks that are driven by the same clock buffer typically exhibit very low skew. This is because the common node is located on the dedicated clock network, close to the leaf clock pins, as shown in the following figure.

Figure 136: Typical Synchronous Clocking Topology with Common Node Located on Green Net



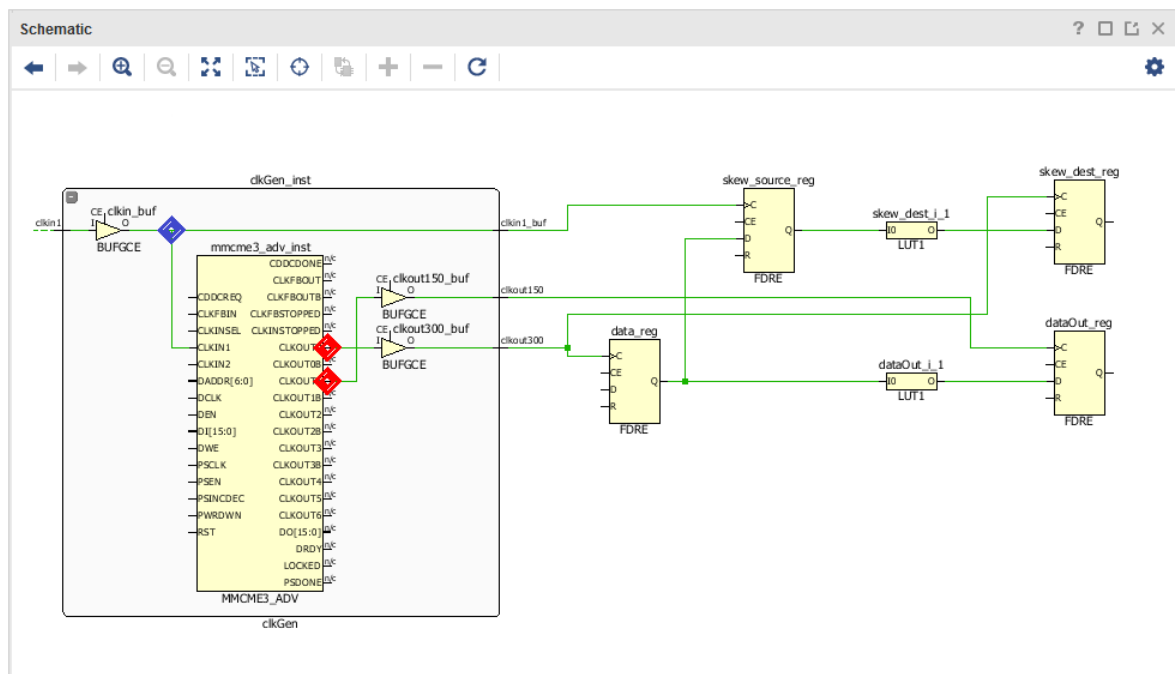
When analyzing the clock path in the timing report, the delays before and after the common node are not provided separately because the common node only exists in the physical database of the design and not in the logical view. For this reason, you can see the common node in the Device window of the Vivado IDE when the Routing Resources are turned on but not in the Schematic window. The timing report only provides a summary of skew calculation with source clock delay, destination clock delay, and credit from clock pessimism removal (CPR) up to the common node.

Limiting Synchronous Clock Domain Crossing Paths

Timing paths between synchronous clocks driven by separate clock buffers exhibit higher skew, because the common node is located before the clock buffers. That is, the common node is farther from the leaf clock pins, resulting in higher pessimism in the timing analysis. The clock skew is even worse for timing paths between unbalanced clock trees due the delay difference between the source and destination clock paths. Although positive skew helps with meeting setup time, it hurts hold time closure, and vice versa.

In the following figure, three clocks have several intra and inter clock paths. The common node of the two clocks driven by the MMCM is located at the output of the MMCM (red markers). The common node of the paths between the MMCM input clock and MMCM output clocks is located on the net before the MMCM (blue marker). For the paths between the MMCM input clock and MMCM output clocks, the clock skew can be especially high depending on the `clkIn_buf` BUFGE location and the MMCM compensation mode.

Figure 137: Synchronous CDC Paths with Common Nodes on Input and Output of a MMCM



Xilinx recommends limiting the number of synchronous clock domain crossing paths even when clock skew is acceptable. Also, when skew is abnormally high and cannot be reduced, Xilinx recommends treating these paths as asynchronous by implementing asynchronous clock domain crossing circuitry and adding timing exceptions.

Adding Timing Exceptions between Asynchronous Clocks

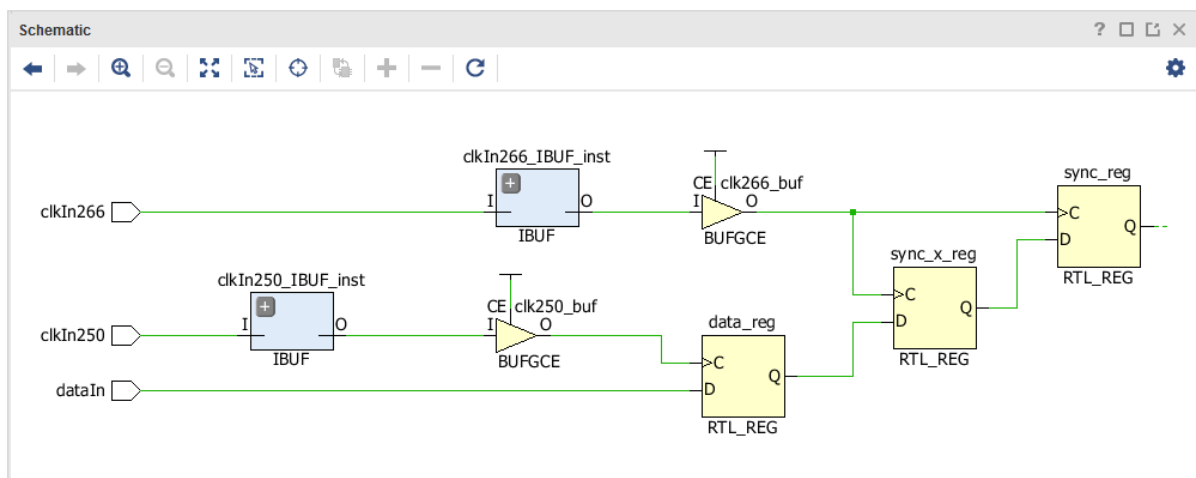
Timing paths in which the source and destination clocks originate from different primary clocks or have no common node, no common phase, or no common period must be treated as asynchronous clocks. In this case, the skew can be extremely large, making it impossible to close timing.

You must review all timing paths between asynchronous clocks to ensure the following:

- Proper asynchronous clock domain crossing circuitry (`report_cdc`)
- Timing exception definitions that ignore timing analysis (`set_clock_groups`, `set_false_path`) or ignore skew (`set_max_delay -datapath_only`)

You can use the Clock Interaction Report (`report_clock_interaction`) to help identify clocks that are asynchronous and are missing proper timing exceptions.

Figure 138: Asynchronous CDC Paths with Proper CDC Circuitry and No Common Node



Related Information

[Defining Clock Groups and CDC Constraints](#)

Applying Common Techniques for Reducing Clock Skew

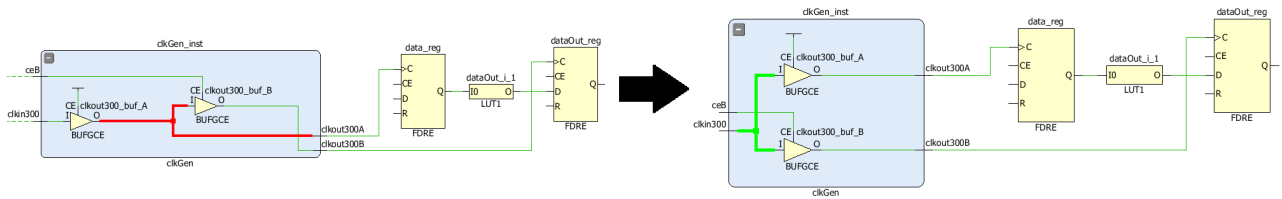


TIP: Given the flexibility of the UltraScale device clocking architecture, the `report_methodology` command contains checks to aid you in creating an optimal clocking topology.

The following techniques cover the most common scenarios:

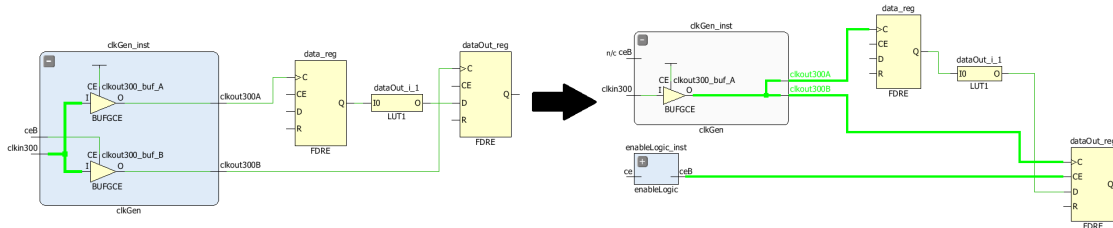
- Avoid timing paths between cascaded clock buffers by eliminating unnecessary buffers or connecting them in parallel as shown in the following figure.

Figure 139: Synchronous Clocking Topology with Cascaded BUFG Reconnected in Parallel



- Combine parallel clock buffers into a single clock buffer and connect any clock buffer clock enable logic to the corresponding sequential cell enable pins, as shown on figure below. If some of the clocks are divided by the buffer's built-in divider, implement the equivalent division with clock enable logic and apply multicycle path timing exceptions as needed. When both rising and falling clock edges are used by the downstream logic or when power is an important factor, this technique might not be applicable.

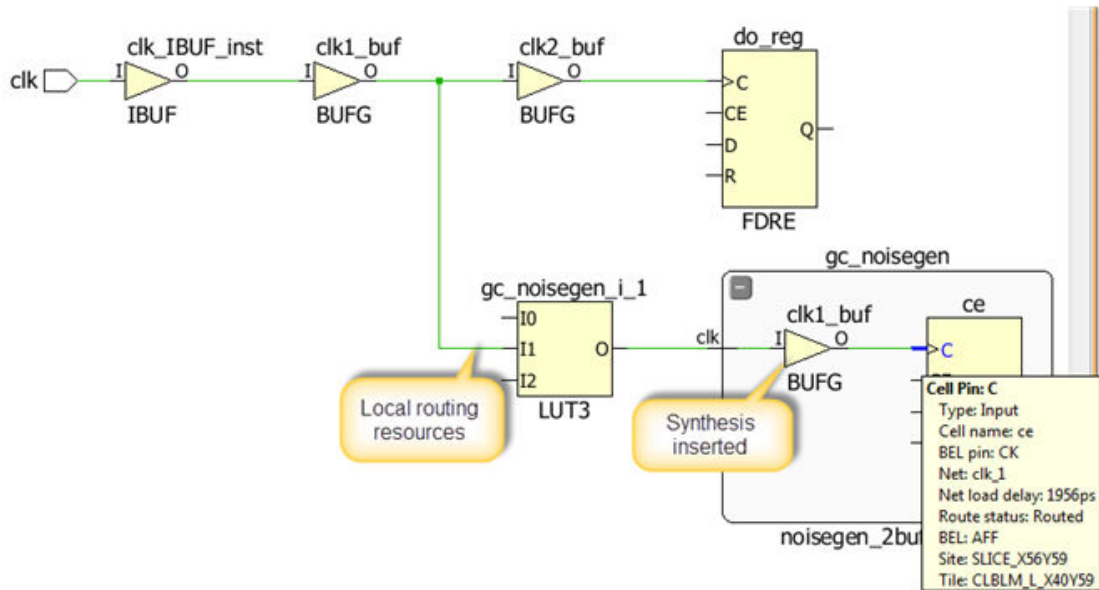
Figure 140: Synchronous Clocking Topology with Parallel Clock Buffer Recombined into a Single Buffer



- Remove LUTs or any combinatorial logic in clock paths as they make clock delays and clock skew unpredictable during placement, resulting in lower quality of results. Also, a portion of the clock path is routed with general interconnect resources which are more sensitive to noise than global clocking resources. Combinatorial logic usually comes from sub-optimal clock gating conversion and can usually be moved to clock enable logic, either connected to the clock buffer or to the sequential cells.

In the following figure, the first BUFG (`clk1_buf`) is used in LUT3 to create a gated clock condition.

Figure 141: Skew Due to Local Routing on Clock Network



IMPORTANT! The 7 series and UltraScale device clocking architectures differ. You must follow the clocking guidelines for your targeted architecture and verify that your design complies.

Related Information

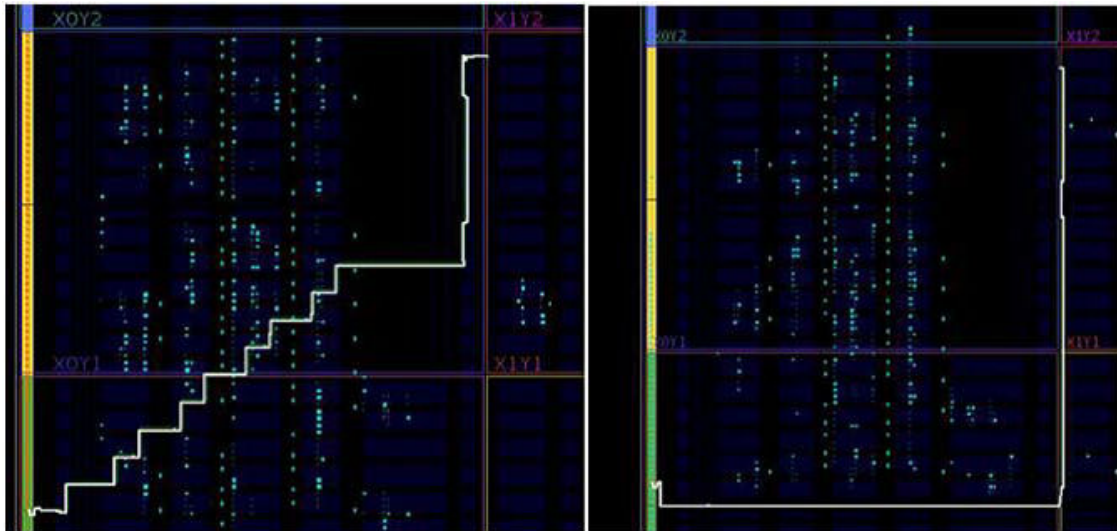
[Clocking Guidelines](#)

Applying Techniques for Improving Skew in 7 Series Devices

Although the 7 series and UltraScale architectures differ in terms of clock architectures, some general clock considerations apply to both families:

- Do not use the `CLOCK_DEDICATED_ROUTE=FALSE` constraint in a production 7 series design. Use `CLOCK_DEDICATED_ROUTE=FALSE` only as a temporary workaround to a clock failure *only* to produce an implemented design in order to view the clocking topology for debugging. Clock paths routed with fabric interconnect can have high clock skew and be impacted by switching noise, leading to poor performance or non-functional designs. In the following figure, the right side has a dedicated clock route, while on the left side, the dedicated route is disabled for clock.

Figure 142: Comparison of Fabric Clock Route versus Dedicated Clock Route

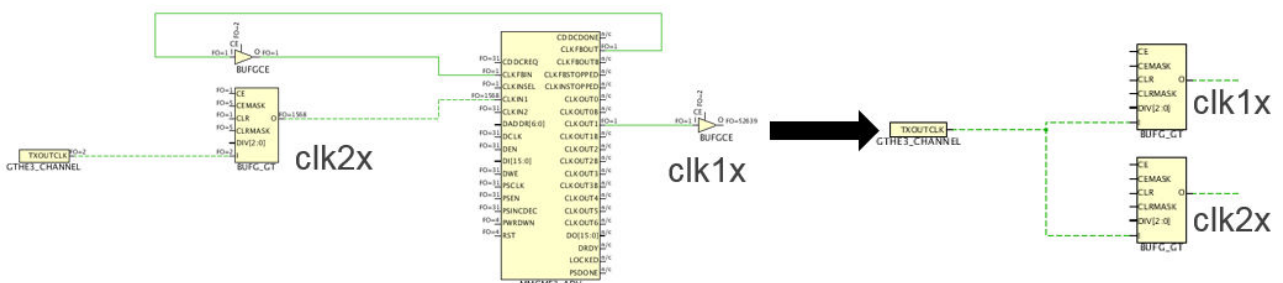


- Do not allow regional clock buffers (BUFR/BUFIO/BUFH) to drive logic in several clock regions as the skew between the clock tree branches in each region will be very high. Remove inappropriate LOC or Pblock constraints to resolve this situation.

Improving Skew in UltraScale and UltraScale+ Devices

- Avoid using an MMCM or PLL to perform simple division of a BUFG_GT clock. BUFG_GT cells have the ability to divide down the input clock. The following figure shows how to save an MMCM resource and implement balanced clock trees for two clocks originating from a GTHE3_CHANNEL cell.

Figure 143: Implementing Balanced Clock Trees using UltraScale BUFG_GT



- Use the CLOCK_DELAY_GROUP on the driver net of critical synchronous clocks to force CLOCK_ROOT and route matching during placement and routing. The buffers of the clocks must be driven by the same cell for this constraint to be honored.

Note: This optimization technique is automatically applied by the `report_qor_suggestions Tcl` command.

- If a timing path is having difficulty meeting timing and the skew is larger than expected, it is possible that the timing path is crossing an SLR or an I/O column. If this is the case, physical constraints such as Pblocks may be used to force the source and destination into a single SLR or to prevent the crossing of an I/O column.
- When working with high speed synchronous clock domain crossing timing paths, constraining the location of the clock modifying blocks, such as the MMCM/PLL, to the center of the clock loads can aid in meeting timing. The decreased delay on the clock networks will result in less timing pessimism on the clock domain crossing paths.
- Verify that clock nets with `CLOCK_DEDICATED_ROUTE=FALSE` constraint are routed with global clocking resources. Use `ANY_CMT_COLUMN` instead of `FALSE` to ensure the clock nets with routing waivers are routed with dedicated clocking resources only. If the clock net is routed with fabric interconnect, identify the design change or clocking placement constraint needed to resolve this situation and make the implementation tools use global clocking resources instead. Clock paths routed with fabric interconnect can have high clock skew or be impacted by switching noise, leading to poor performance or non-functional designs.

Related Information

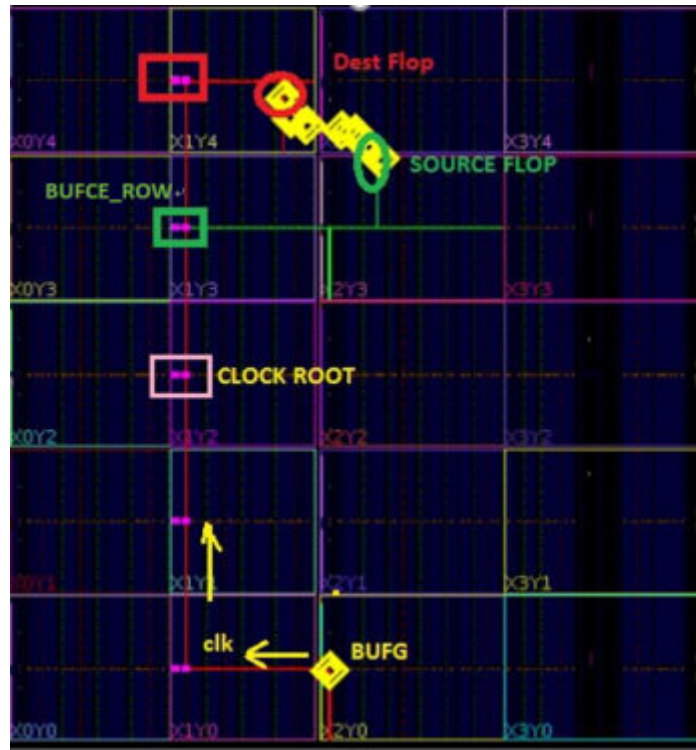
[Synchronous CDC
Clock Constraints](#)

Reducing Clock Delay in UltraScale and UltraScale+ Devices

In UltraScale and UltraScale+™ global clock routing, the clock net is first routed from a global clock buffer via the horizontal and vertical routing track to a central location called the clock root. From the clock root, the clock net spans out to drive clock rows in each clock region via the vertical distribution track. On each row, there are programmable delays in the clock network on `BUFCE_ROW` route-through sites that perform a coarse-grained deskew as the clock spans farther from the clock root.

The following figure shows a clock path from the global clock buffer (BUFG) to the clock root. The clock routing switches from routing to the vertical distribution track, through the `BUFCE_ROW` in each clock region row that drives the horizontal distribution tracks, and then to the leaf level. The source is shown in green and the destination in red.

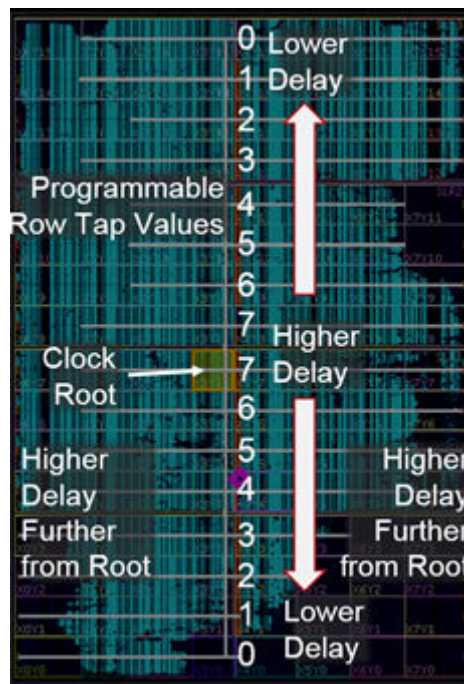
Figure 144: Clock Path from BUFG to the Leaf Level via BUFCE_ROW



The row programmable tap delay is the largest near the clock root. This delay decreases by one tap for one clock region as the clock reaches farther away from the root in the vertical direction, eventually decreasing to zero.

The following figure shows the topology of the programmable row tap values decreasing from the root. Higher tap values mean higher delays and higher crossing SLR clock skew, because the higher tap values add additional uncertainty for timing due to the minimum/maximum delay variation introduced by the manufacturing process variation. This makes it more difficult to meet timing near the root where programmable tap delay values are higher. Farther from the root in the vertical direction, there is less uncertainty, and it is generally easier to fix hold violations on crossing SLR buses. For SLR crossing buses that are farther from the root in the horizontal direction, the clock row delays increase. This additional delay introduces more minimum/maximum delay variation and reduces the performance of SLR crossings.

Figure 145: Row Programmable Tap Delay Settings Across an UltraScale+ SSI Technology Device



For UltraScale+ SSI technology devices, you can improve SLR crossing speed using either of the following methods:

- Move the clock root close to the SLR crossings in the horizontal direction
- Limit the maximum row programmable tap delay value to reduce the uncertainty

Note: Timing paths farther from the root in the vertical direction might become slightly slower due to increased delay from hold fixing route detours. However, using these methods results in an overall performance gain.

You can review the row programmable tap delay settings that the Vivado tool chose for each global clock in your design in the Device Cell Placement Summary for Global Clock sections in the Clock Utilization Report. Following is an example that shows the row programmable tap delay settings for the g13 global clock in the HORIZONTAL PROG DELAY column, which is highlighted in yellow.

Figure 146: Global Clock Row Programmable Tap Delay Settings in the Clock Utilization Report

22. Device Cell Placement Summary for Global Clock g13

Global Id	Driver Type/Pin	Driver Region (D)	Clock	Period (ns)	Waveform (ns)	Root (R)	Slice Loads
g13	BUFGCE/0	X4Y10	Multiple	4.926	{0.000 2.463}	X3Y8	12511

* Slice Loads column represents load cell count of all cell types other than IO, GT and clock resources
 ** IO Loads column represents load cell count of IO types
 *** Clocking Loads column represents load cell count that are clock resources (global clock buffer, MMCM, PLL, etc)
 **** GT Loads column represents load cell count of GT types

	X0	X1	X2	X3	X4	X5	X6	X7	HORIZONTAL PROG DELAY
Y15	2820	4086	308	0	0	0	0	0	0
Y14	713	392	0	0	3	0	0	0	0
Y13	0	0	0	0	0	0	0	0	0
Y12	0	0	0	0	0	0	0	0	0
Y11	0	0	0	0	0	62	94	3	3
Y10	0	0	801	3	{D} 45	224	216	0	4
Y9	0	0	252	91	361	185	10	0	5
Y8	0	0	66	{R} 261	241	2	0	0	5
Y7	0	17	365	117	16	0	0	0	4
Y6	0	165	275	39	2	0	0	0	3
Y5	0	120	66	0	0	0	0	0	2
Y4	0	27	7	0	0	0	0	0	1
Y3	21	21	0	3	0	0	0	0	0
Y2	11	1	0	0	0	0	0	0	0
Y1	0	0	0	0	0	0	0	0	0
Y0	0	0	0	0	0	0	0	0	0

For UltraScale+ SSI technology devices, the placer limits the maximum row programmable tap delay value to reduce minimum/maximum delay variation and reduce SLR crossing clock skew near the clock root, while also ensuring that clock regions on either side of SLR crossings have an increasing or decreasing tap delay value to balance the clock skew on SLR crossing paths farther from the root. The MAX_PROG_DELAY property value of the clock net can be queried to find the maximum row programmable tap delay value used by the placer.

You can also limit the row programmable tap value using the USER_MAX_PROG_DELAY property. Following is an example. To set the USER_MAX_PROG_DELAY property, the value must be applied to the net segment directly driven by the global clock buffer. If the USER_MAX_PROG_DELAY property is not set, the placer can use the maximum possible tap setting of 7.

```
set_property USER_MAX_PROG_DELAY <0-7> [get_nets -of [get_pins BUFG/0]]
```

Following are tips when using the USER_MAX_PROG_DELAY property:

- The recommended USER_MAX_PROG_DELAY tap value is 3 or 4 for clocks that span the majority of UltraScale+ SSI technology devices. When clock roots are near GT, PCIe®, or CMAC blocks that are off-center in the device, SLR crossing performance on the opposite device side is heavily impacted, because the common node for the launch and capture clock is farther away from the SLR crossing.
- For clock groups using the CLOCK_DELAY_GROUP for clock network matching, ensure that all clocks within the clock group use the same USER_MAX_PROG_DELAY value.

Reducing Clock Uncertainty

Clock uncertainty is the amount of uncertainty relative to an ideal clock. Uncertainty can come from user-specified external clock uncertainty (`set_clock_uncertainty`), system jitter, or duty cycle distortion. Clock-modifying blocks such as the MMCM and PLL also contribute to clock uncertainty in the form of Discrete Jitter, and Phase Error if multiple related clocks are used.

The Clocking Wizard provides accurate uncertainty data for the specified device and can generate various MMCM clocking configurations for comparing different clock topologies. To achieve optimal results for the target architecture, Xilinx recommends regenerating clock generation logic using the Clocking Wizard rather than using legacy clock generation logic from prior architectures.

Using MMCM Settings to Reduce Clock Uncertainty

Note: The `report_qor_suggestions` Tcl command flags this issue.

When configuring an MMCM for frequency synthesis, Xilinx recommends configuring the MMCM to achieve the lowest output jitter on the clocks. Optimize the MMCM settings to run at the highest possible voltage-controlled oscillator (VCO) frequency that meets the allowed operating range for the device. The following equations show the relationship between VCO frequency, M (multiplier), D (divider), and O (output divider) settings to both the input and output clock frequencies:

$$F_{VCO} = F_{CLKIN} \times \frac{M}{D}$$

$$F_{OUT} = F_{CLKIN} \times \frac{M}{D \times O}$$



TIP: You can increase the VCO frequency by increasing M, lowering D, or both and compensating for the change in frequency by increasing O. Increases in VCO frequency negatively affects the power dissipation from the MMCM or PLL. You can also make small increases in the VCO frequency when you switch from multiple MMCM clock outputs using BUFGs to one MMCM clock output using BUFGCE_DIVs, which allows more clocks to use the fractional divider. When selecting between MMCM and PLL, MMCMs are preferred because they are able to operate at a higher VCO frequency, have improved granularity for selecting M and D values, and have fractional dividers (CLKOUT0).

Different architectures have different VCO frequency maximums. Therefore, Xilinx recommends regenerating clocking components to be optimal for your target architecture. Xilinx recommends using the Clocking Wizard to automatically calculate M and D values along with the VCO frequency to properly configure an MMCM for the target device.



TIP: When using the Clocking Wizard from the IP catalog, make sure that Jitter Optimization Setting is set to the Minimize Output Jitter, which provides the higher VCO frequency. In addition, performing marginal changes to the desired output clock frequency can allow for an even higher VCO frequency to further reduce clock uncertainty.

The following MMCM frequency synthesis example uses an input clock of 62.5 MHz to generate an output clock of approximately 40 MHz. There are two solutions, but the MMCM_2 with a higher VCO frequency generates less clock uncertainty due to reduced jitter and phase error.

Table 14: MMCM Frequency Synthesis Example

	MMCM_1	MMCM_2
Input clock	62.5 MHz	62.5 MHz
Output clock	40.0 MHz	39.991 MHz
CLKFBOUT_MULT_F(M)	16	22.875
DIVCLK_DIVIDE(D)	1	1
VCO Frequency	1000.000 MHz	1429.688
CLKOUT0_DIVIDE_F(O)	25	35.750
Jitter (ps)	167.542	128.632
Phase Error (ps)	384.432	123.641

Using BUFGE_DIV to Reduce Clock Uncertainty

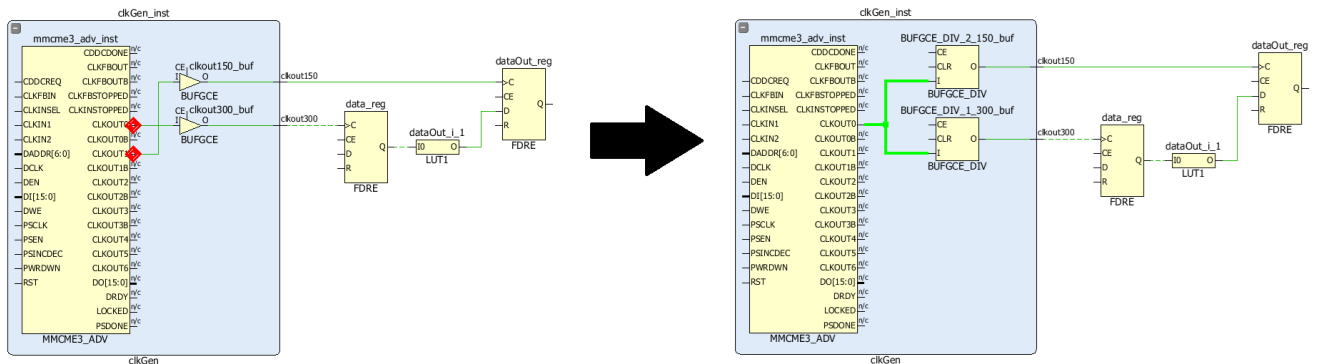


TIP: The `report_qor_suggestions` Tcl command flags this issue.

In UltraScale devices, BUFGE_DIV cells can be used to reduce clock uncertainty on synchronous clock domain crossings by eliminating MMCM Phase Error. For example, consider a path between a 300 MHz and 150 MHz clock domains, where both clocks are generated by the same MMCM.

In this case, the clock uncertainty includes 120 ps of Phase Error for both Setup and Hold analysis. Instead of generating the 150 MHz clock with the MMCM, a BUFGE_DIV can be connected to the 300 MHz MMCM output and divide the clock by 2. For optimal results, the 300 MHz clock needs to also use a BUFGE_DIV with BUFGE_DIVIDE set to 1 to match the 150 MHz clock delay accurately, as shown in the following figure.

Figure 147: Improving the Clock Topology for an UltraScale Synchronous CDC Timing Path



With the new topology:

- For setup analysis, clock uncertainty does not include the MMCM phase error and is reduced by 120 ps.
- For hold analysis, there is no more clock uncertainty (only for same edge hold analysis).
- The common node moves closer to the buffers, which saves some clock pessimism.

By applying the `CLOCK_DELAY_GROUP` constraint on the two clock nets, the clock paths will have matched routing.

Note: The `report_qor_suggestions` Tcl command provides these constraints.

The following tables compare the clock uncertainty for setup and hold analysis of an UltraScale synchronous CDC timing path.

Table 15: Comparison of Clock Uncertainty for Setup Analysis of an UltraScale Synchronous CDC Timing Path

Setup Analysis	MMCM Generated 150 MHz Clock	BUFGCE_DIV 150 MHz Clock
Total System Jitter (TSJ)	0.071 ns	0.071 ns
Discrete Jitter (DJ)	0.115 ns	0.115 ns
Phase Error (PE)	0.120 ns	0.000 ns
Clock Uncertainty	0.188 ns	0.068 ns

Table 16: Comparison of Clock Uncertainty for Hold Analysis of an UltraScale Synchronous CDC Timing Path

Hold Analysis	MMCM Generated 150 MHz Clock		BUFGCE_DIV 150 MHz Clock
	Total System Jitter (TSJ)	0.071 ns	0.000 ns
	Discrete Jitter (DJ)	0.115 ns	0.000 ns
	Phase Error (PE)	0.120 ns	0.000 ns
	Clock Uncertainty	0.188 ns	0.000 ns

Related Information

[Synchronous CDC](#)

Applying Common Timing Closure Techniques

The following techniques can help with design closure on challenging designs. Before attempting these techniques, ensure that the design is properly constrained and that you identify the main issue that affects the top violating paths.



RECOMMENDED: Xilinx recommends running the `report_qor_suggestions` Tcl command to identify and apply many of these techniques automatically. For more information, see this [link](#) in the Vivado Design Suite User Guide: Design Analysis and Closure Techniques (UG906).

Improving the Netlist with Block-Level Synthesis Strategies

Although most designs can meet timing requirements with the default Vivado synthesis settings, larger and more complex designs usually require a mix of synthesis strategies for different hierarchies to close timing.

For example, one module might require the use of MUXF* resources to implement a timing critical function, but the rest of the design might benefit from implementation of logic in LUTs rather than MUXF* to reduce congestion. In this case, set the PERFORMANCE_OPTIMIZED strategy for the timing-critical module, and synthesize the rest of the design using the Flow_AlternateRoutability strategy to reduce congestion.

Related Information

[Block-Level Synthesis Strategy](#)

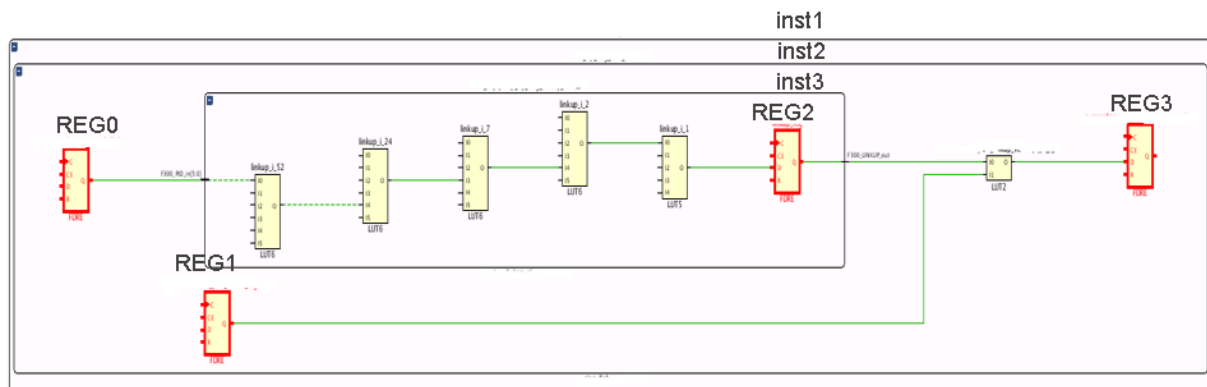
Improving Logic Levels

Throughout the design cycle, you must verify that the logic level distribution fits the clock frequency goals for the target Xilinx device family and device speed grade. Although a limited number of paths with a high number of logic levels do not always introduce a timing closure challenge, you can improve the timing QoR by optimizing the longest paths in the design with the Vivado synthesis retiming option.

Using the retiming option globally is usually runtime intensive and can negatively impact power. Therefore, Xilinx recommends that you identify a specific hierarchy with violations on paths with a high number of logic levels after synthesis or with optimal placement. When the paths in the fanin or fanout of the longest paths have fewer logic levels and are contained within a small or medium hierarchical module, you can use the `BLOCK_SYNTH.RETIMING` block-level synthesis strategy.

The following figure shows a critical paths with five LUTs, constrained by a 600 MHz clock. The REG2 destination flop drives a timing path with a single LUT that is included one hierarchy up from REG2.

Figure 148: Schematic Showing Critical Path with Five Logic Levels



In addition to using the Schematic window in the Vivado IDE, you can use the `report_design_analysis -logic_level_distribution` command to review the distribution of logic levels for specific paths. This allows you to determine how many paths need to be rebalanced to improve the timing QoR.

You can use the `retiming_forward` and `retiming_backward` attributes available in Vivado synthesis to control the optimization on a specific register or a path. Using these attributes applies retiming optimization on a specific set of paths rather than on the top module or submodules, which reduces the area overhead. You can apply these attributes in the RTL or in the XDC file. For more information, including usage and restrictions, see the *Vivado Design Suite User Guide: Synthesis* (UG901).

The following figure shows 58 paths with five logic levels within the inst1/inst2 hierarchy constrained with the 600 MHz clock and 32 paths with only one logic level.

Figure 149: Logic Level Distribution with Default Synthesis Optimization

```
current_instance inst1/inst2
report_design_analysis -timing -logic_level_distribution -of_timing_paths [get_timing_paths -max_paths 100 -group core_clk_600]
```

End Point Clock	Requirement	0	1	2	3	4	5	6	7	8	9	10	11-15	16-20	21-25	26-30	31+
core_clk_600	1.667ns	0	32	10	0	0	58	0	0	0	0	0	0	0	0	0	0

Vivado synthesis can rebalance the logic levels by moving the registers in the low logic level paths into the high logic level paths. In this example, you can add the following constraint to the synthesis XDC file to perform retiming on the inst1/inst2 hierarchy:

```
set_property BLOCK_SYNTH.RETIMING 1 [get_cells inst1/inst2]
```

After rerunning synthesis with the same global settings and the updated XDC file, you can run regular timing analysis on the inst1/inst2 timing paths or rerun the `report_design_analysis` command to verify that the longest paths have fewer logic levels, as shown in the following figure. The critical path is now **REG0 → 3 LUTs → REG2** (backward retimed), and the path from REG2 to REG4 has three logic levels.

Figure 150: Logic Level Distribution with Retiming Enabled for Synthesis Optimization

```
current_instance inst1/inst2
report_design_analysis -timing -logic_level_distribution -of_timing_paths [get_timing_paths -max_paths 100 -group core_clk_600]
```

End Point Clock	Requirement	0	1	2	3	4	5	6	7	8	9	10	11-15	16-20	21-25	26-30	31+
core_clk_600	1.667ns	0	0	0	100	0	0	0	0	0	0	0	0	0	0	0	0

Reducing Control Sets

Note: This optimization technique is automatically applied by the `report_qor_suggestions Tcd` command.

Often not much consideration is given to control signals such as resets or clock enables. Many designers start HDL coding with "if reset" statements without deciding whether the reset is needed or not. While all registers support resets and clock enables, their use can significantly affect the end implementation in terms of performance, utilization, and power.

The first factor to consider is the number of control sets. A control set is the group of clock, enable, and set/reset signals used by a sequential cell. For example, two cells connected to the same clock have different control sets if only one cell has a reset or if only one cell has a clock enable. Constant or unused enable and set/reset register pins also contribute to forming control sets.

The second factor to consider is the targeted architecture. The number of control sets that can be packed together depends on the architecture:

- A 7 series device slice (or half-CLB) comprises eight registers, which all share one clock, one set/reset, and one clock enable. Only one control set can be used per group of eight registers.
- An UltraScale device half-CLB comprises two groups of four registers, which share one clock and one set/reset. In addition, each group of four registers has one clock enable and can ignore the set/reset. A constant set/reset signal is not routed and can be ignored. A constant enable signal is treated like a dynamic enable signal and needs to be routed. Under optimal conditions, up to two control sets can be used per group of eight registers.

CLB packing restrictions caused by control sets force the placer to move some registers, including their input LUT. In some cases, the registers are moved to less optimal locations. The additional distance can negatively impact not only utilization but also placement QoR and power consumption, due to logic spreading (longer net delays) and higher interconnect resources utilization. This is mainly of concern in designs with many low fanout control signals, such as clock enables that feed single registers.

Despite the higher UltraScale device CLB control set capacity, typical designs show a control set utilization similar to 7 series designs. Therefore, Xilinx recommendations are the same for both architectures.

Follow Control Set Guidelines

The following table provides a guideline for the recommended number of control sets, depending on the target device size, for both 7 series and UltraScale devices.

Table 17: Control Set Guidelines

Guideline	Percentage of Control Sets
Acceptable	Less than 7.5% of the total number of control sets in the device
Reduction Recommended	Between 7.5% and 15% of the total number of control sets in the device
Reduction Required	Greater than 15% of the total number of control sets in the device

These guidelines assume the following:

- Typical control set capacity: 1 per 8 CLB registers
- Total number of control sets in a device: CLB registers / 8

To determine the number of control sets in a design:

- Before placement: Use `report_control_sets -verbose`
- After placement: Use `report_utilization` (text mode only)



TIP: The number of unique control sets can be a problem in a small portion of the design, resulting in longer net delays or congestion in the corresponding device area. Identifying the high local density of unique control sets requires detailed placement analysis in the Vivado IDE Device window, which includes highlighted control signals in different colors.

Reduce the Number of Control Sets

If the number of control sets is high, use one of the following strategies to reduce their number:

- Remove the MAX_FANOUT attributes that are set on control signals in the HDL sources or constraint files. Replication on control signals dramatically increases the number of unique control sets. Xilinx recommends relying on `place_design` to perform coarse replication and using `phys_opt_design -directive Explore` for finer replication after placer. This prevents unnecessary replication and equivalent control sets from crossing each other, which can lead to routing congestion.
- Increase the control set threshold of Vivado synthesis (or other synthesis tool). Review the control sets fanout distribution table in `report_control_sets -verbose` to determine a more appropriate control sets threshold to use during synthesis. Note that increasing `control_set_opt` can have negative impacts on power by eliminating clock enables that can actively reduce power. For example:

```
synth_design -control_set_opt_threshold 16
```



TIP: Use the BLOCK_SYNTH synthesis constraints to change the control sets threshold on modules that are the most impacted by placement spreading or congestion.

- Use `opt_design -control_set_merge` or `opt_design -merge_equivalent_drivers` to merge equivalent control sets after synthesis.
- Use the CONTROL_SET_REMAP property to map low-fanout control signals driving the synchronous set/reset and/or CE pin of a register to the D-input. For more information, see this [link](#) in the *Vivado Design Suite User Guide: Implementation (UG904)*.
- Avoid low fanout asynchronous set/reset (preset/clear), because they can only be connected to dedicated asynchronous pins and cannot be moved to the datapath by synthesis. For this reason, the synthesis control set threshold option does not apply to asynchronous set/reset.
- Avoid using both active-High and active-Low of a control signal for different sequential cells.
- Only use clock enable and set/reset when necessary. Often data paths contain many registers that automatically flush uninitialized values, and where set/reset or enable signals are only needed on the first and last stages.

Related Information

[Control Signals and Control Sets](#)

Optimizing High Fanout Nets

High fanout nets often lead to implementation issues. Because die sizes increase with each device family, fanout problems also increase. It is often difficult to meet timing on nets that have many thousands of endpoints, especially if there is additional logic on the paths, or if they are driven from non-sequential cells, such as LUTs or distributed RAMs.

Allow Register Replication

Most tools can replicate registers to reduce high fanout nets on critical paths. Alternatively, you can apply attributes on specific registers or levels of hierarchy to specify which registers can or cannot be replicated. For example, the presence of a LUT1 on a replicated net indicates that an attribute or constraint is partly preventing the optimization. During synthesis, a KEEP_HIERARCHY attribute on a hierarchical cell traversed by the optimized net or a KEEP attribute on net segment in a different hierarchy can alter the replication optimizations. During synthesis and implementation, a DONT_TOUCH constraint also prevents beneficial replications.

Sometimes, designers address the high fanout nets in RTL or synthesis by using a MAX_FANOUT attribute on a specific net. This does not always result in the most optimal routing resource usage, especially if the MAX_FANOUT attribute is set too low or is set on a net connected to several major hierarchies. In addition, if the high fanout signal is a register control signal and is replicated more than necessary, this can lead to a higher number of control sets and increase design power by unnecessarily adding additional registers that may not be necessary for timing closure.

Often, a better approach to reducing fanout is to use a balanced tree for the high fanout signals. Consider manually replicating registers based on the design hierarchy, because the cells included in a hierarchy are often placed together.

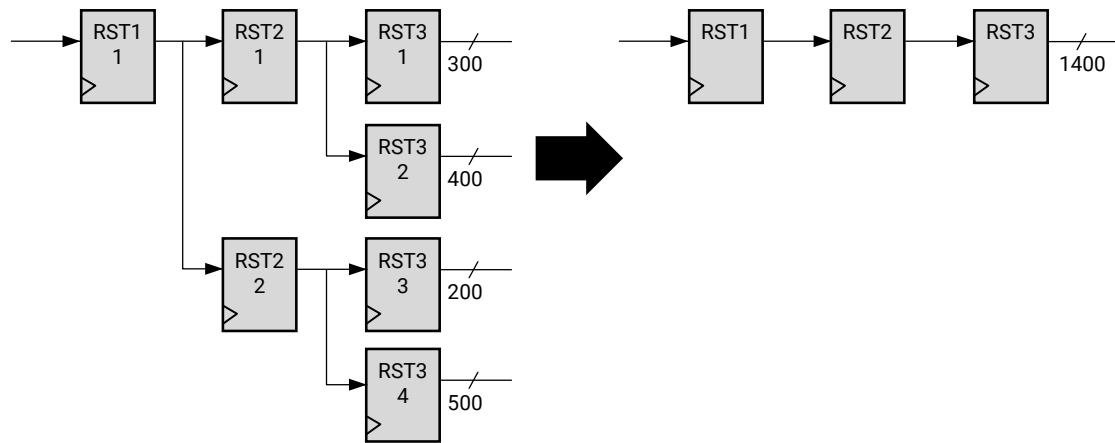
To restructure and reduce the number of control set trees and high fanout nets, you can use the `opt_design` Tcl command with one of the following options:

- `-control_set_merge`: This option aggressively combines the drivers of logically-equivalent control signals to a single driver.
- `-merge_equivalent_drivers`: This option merges logically-equivalent signals, including control signals, to a single driver.

Note: Try this option first, because the tools are aware of major hierarchies and Pblock constraints when you run this option.

These options are the reverse of fanout replication and result in nets that are better suited for module-based replication. This merge also works across multi-stage reset trees as shown in the following figure.

Figure 151: Control Set Merging Using `opt_design -control_set_merge`



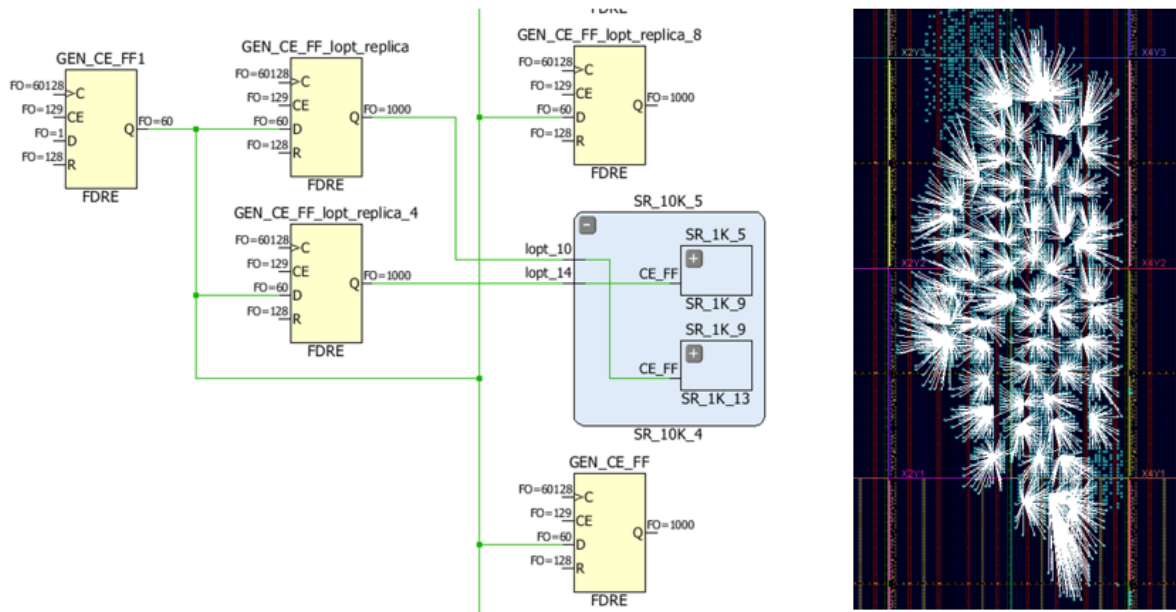
X20035-122019

After reducing the number of replicated objects, you can use the `opt_design` Tcl command to perform limited replication based on the hierarchy characteristics, with the following option:

- `-hier_fanout_limit <arg>`: This option replicates registers according to the hierarchy where `<arg>` represents the fanout limit for the replication according to the logical hierarchy. For each hierarchical instance driven by the high fanout net, if the fanout within the hierarchy is greater than the specified limit, the net within the hierarchy is driven by a replica of the driver of the high fanout net. The replicated driver is placed in the same level of hierarchy as the original driver, and replication is not limited to control set registers.

The following figure shows replication on a clock enable net with a fanout of 60000 using `opt_design -hier_fanout_limit 1000`. Because each module `SR_1K` contains 1000 loads, the driver is replicated 59 times.

Figure 152: Module-Based Replication on a High-Fanout Clock Enable Net



Fanout optimization is enabled by default in `place_design`. Replication occurs early in the placer flow and is based on placement information. Registers that drive more than 1000 loads and registers that drive DSPs, block RAMs, and UltraRAMs are considered for replication and are co-located with the loads if replication occurs. You can force the replication of a register or a LUT driving a net by adding the `FORCE_MAX_FANOUT` property to the net. The value of the `FORCE_MAX_FANOUT` specifies the maximum physical fanout the nets should have after the replication optimization.

You can force replication based on physical device attributes with the `MAX_FANOUT_MODE` property. Supported `MAX_FANOUT_MODE` properties are `CLOCK_REGION`, `SLR`, `MACRO`. For example, the `MAX_FANOUT_MODE` property with a value of `CLOCK_REGION` replicates the driver based on the physical clock region, the loads placed into same clock region will be clustered together. For more information, see this [link](#) in the *Vivado Design Suite User Guide: Implementation (UG904)*.

For SSI technology devices, high-fanout drivers can be replicated for each SLR and optionally assigned to SLR-aligned Pblocks along with their loads. This technique helps reduce the impact of the SLR crossing delay and gives more freedom to place the replicated high fanout nets independently in each SLR.

Related Information

[Replicate High Fanout Net Drivers](#)

Promote High Fanout Nets to Global Routing

Note: This optimization technique is automatically applied by the `report_qor_suggestions Tcl` command.

Lower performance high fanout nets can be moved onto the global routing by inserting a clock buffer between the driver and the loads. This optimization is automatically performed in `opt_design` for nets with a fanout greater than 25000 only when a limited number of clock buffers are already used and the clock period of the logic driven by the net is above the limit specific to the targeted device and speed grade.

You can force `synth_design` and `opt_design` to insert a clock buffer when setting the `CLOCK_BUFFER_TYPE` attribute on a net in the RTL file or in the constraint file (XDC). For example:

```
set_property CLOCK_BUFFER_TYPE BUFG [get_nets netName]
```

Using global clocking ensures optimal routing at the cost of higher net delay. For best performance, clock buffers must drive sequential loads directly, without intermediate combinatorial logic. In most cases, `opt_design` reconnects non-sequential loads in parallel to the clock buffer. If needed, you can prevent this optimization by applying a `DONT_TOUCH` on the clock buffer output net. Also, if the high fanout net is a control signal, you must identify why some loads are not dedicated clock enable or set/reset pins.

The placer also automatically routes high fanout nets (fanout > 10000) on any global routing tracks available after clock routing is performed. This optimization occurs towards the end of the placer flow and is only performed if timing does not degrade. You can disable this feature using the `-no_bufg_opt` option.

Related Information

[Control Signals and Control Sets](#)

Use Physical Optimization

Physical optimization (`phys_opt_design`) automatically replicates the high fanout net drivers based on slack and placement information, and usually significantly improves timing. Xilinx recommends that you drive high fanout nets with a fabric register (FD*), which is easier to replicate and relocate during physical optimization.

In some cases, the default `phys_opt_design` command does not replicate all critical high fanout nets. Use a different directive to increase the command effort: `Explore`, `AggressiveExplore` or `AggressiveFanoutOpt`. Also, when a high fanout net becomes critical during routing, you can add an iteration of `phys_opt_design` to force replication on specific nets before trying to route the design again. For example:

```
phys_opt_design -force_replication_on_nets [get_nets [list netA netB netC]]
```


Prioritize Critical Logic Using the group_path Command

You can use the `group_path` command with the `-weight` option to give higher priority to the path endpoints defined in a clock group. For example, to assign a higher priority to group of logic clocked by a specific clock, use the following command:

```
group_path -name [get_clocks clock] -weight 2
```

In this example, the implementation tools give higher priority to the paths that belong to clock group `clock` with a weight of 2 over other paths in the design.

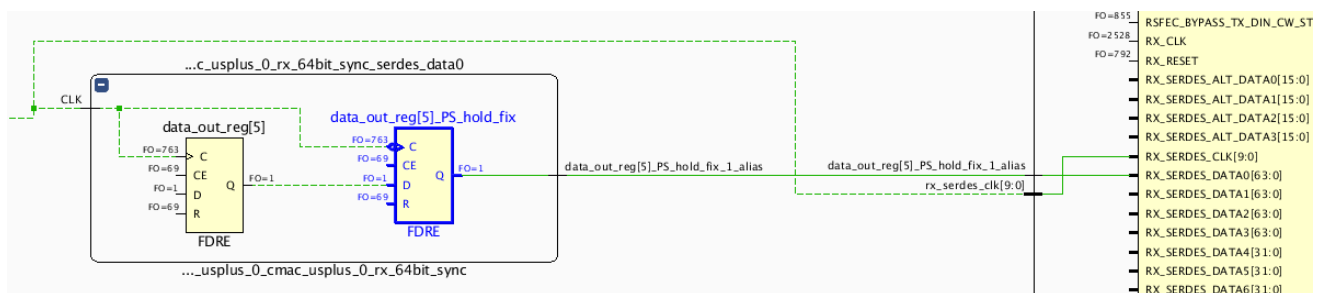
Fixing Large Hold Violations Prior to Routing

For paths that have large hold violations (> 0.4 ns), it is advantageous to reduce the hold violations prior to routing the design, making it easier for the router to fix the remaining smaller hold violations using route detours. Reducing hold violations prior to routing can be beneficial if hold fixing has been identified as a source of routing congestion. The `phys_opt_design` hold fixing options each use different resources and have specific targets. It is important to use the proper option depending upon the device utilization and desired impact. Prior to running `phys_opt_design` for hold fixing, it is important to validate that the design has properly constrained clocktrees for minimal skew.

The insertion of negative-edge triggered registers between sequential elements can split a timing path into two half period paths and significantly reduce hold violations. You can insert the negative-edge triggered registers using the `-insert_negative_edge_ffs` option during the `phys_opt_design` implementation step. Only paths with flip-flop drivers and at most one LUT in between the sequential elements are considered for this optimization. The setup slack on the paths must be sufficiently positive after the optimization or else the optimization is discarded.

The following figure shows a negative-edge triggered register inserted after a flip-flop driving a CMAC block. Before the optimization, the hold slack between the flip-flop and the driver was -0.492 ns. After the insertion of the negative-edge triggered register (highlighted in blue), the setup and hold slack are both positive.

Figure 153: Fixing Hold Violation with Negative Edge Register Insertion



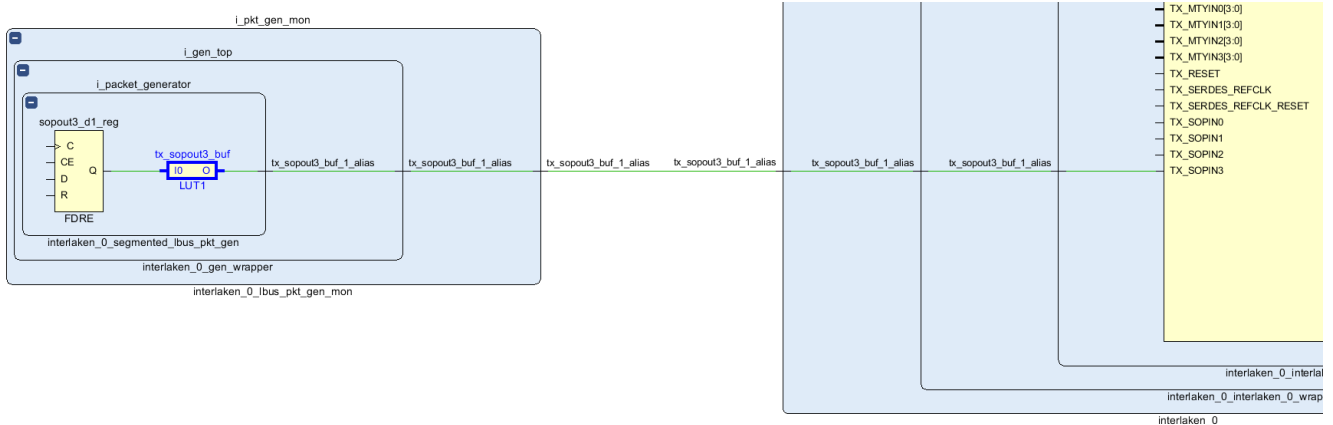
You can also insert LUT1 delays onto datapaths to reduce hold violations. To insert LUT1 delays, use one of the following options during the `phys_opt_design` implementation step:

- `-hold_fix`: Performs LUT1 insertion and only considers paths that are the largest WHS violators with sufficient positive setup slack.
- `-aggressive_hold_fix`: Performs LUT1 insertion in a more aggressive manner than the standard `-hold_fix` option. The `-aggressive_hold_fix` optimization considers many hold violating paths for LUT1 insertion and can be used to significantly reduce design THS at the expense of LUT utilization.

Note: The `phys_opt_design -directive ExploreWithAggressiveHoldFix` directive runs the `Explore` directive along with the `-aggressive_hold_fix` as a single optimization.

The following figure shows a LUT1 delay inserted after a flip-flop driving an ILKN block. Before the optimization, the path from the flip-flop to the ILKN is the WHS path in the design with -0.277 ns hold slack. After the insertion of the LUT1 delay (highlighted in blue), the hold slack is positive and the setup slack remains positive.

Figure 154: Fixing Hold Violation with LUT1 Delay Insertion



Addressing Congestion

Congestion can be caused by a variety of factors and is a complex problem that does not always have a straightforward solution. The `report_design_analysis` congestion report helps you identify the congested regions and the top modules that are contained within the congestion window. Various techniques exist to optimize the modules in the congested region. The `report_qor_suggestions` can automate the resolution of many of the items that cause congestion.



TIP: Before you try to address congestion with the techniques discussed in the following sections, make sure that you have clean constraints and you followed the clocking guidelines recommended by Xilinx. Excessive hold time failures (or negative hold slack) and clock uncertainties require the router to detour, which can lead to congestion. Avoid overlapping Pblocks, which can also cause congestion.

Lower Device Utilization

When several fabric resource utilization percentages are high (on average > 75%), placement becomes more challenging if the netlist complexity is also high (high top-level connectivity, high Rent exponent, high average fanout). High performance designs also come with additional placement challenges. In such situations, revisit the design features and consider removing non-essential modules until only one or two fabric resource utilization percentages are high. If logic reduction is not possible, review the other congestion alleviation techniques presented in this chapter.



TIP: Review resource utilization after `opt_design` to get more accurate numbers, once unused logic has been trimmed instead of after synthesis.

Use Alternate Placer and Router Directives

Because placement typically has the greatest impact on overall design performance, applying different placer directives is one of the first techniques that should be tried to reduce congestion. Consider running the alternate placer directives without any existing Pblock constraints in order to give more freedom to the placer to spread the logic as needed.

Several placer directives exist that can help alleviate congestion by spreading logic throughout the device to avoid congested regions. The SpreadLogic placer directives are:

- AltSpreadLogic_high
- AltSpreadLogic_medium
- AltSpreadLogic_low
- SSI_SpreadLogic_high
- SSI_SpreadLogic_low

When congestion is detected on SLR crossing, consider using:

- SSI_BalanceSLLs placer directive which helps with partitioning the design across SLRs while attempting to balance SLLs between SLRs.
- SSI_SpreadSLLs placer directive which allocates extra area for regions of higher connectivity when partitioning across SLRs.

Other placer directives or implementation strategies might also help with alleviating congestions and should also be tried after the placer directives mentioned above.

To compare congestion for different placer directives either run the Design Analysis Congestion report after `place_design`, or examine the initial estimated congestion in the router log file.

Routing has less impact on congestion than placer directives. However, in some cases it is useful to attempt different routing directives. The following directive ensures that the router works harder to access more routing and relieve congestion in the interconnect tiles:

- AlternateCLBRouting

Note: The AlternateCLBRouting routing directive is most effective when there is short congestion or both short and long congestion. This directive only applies to UltraScale devices.

For more information, see this [link](#) in the *Vivado Design Suite User Guide: Implementation (UG904)*.

Related Information

[Congestion Level Ranges](#)

Turn Off Cross-Boundary Optimization

Prohibiting cross-boundary optimization in synthesis prevents additional logic getting pulled into a module. This reduces the complexity of the modules but can also lead to higher overall utilization. This can be done globally with the `-flatten_hierarchy none` option in `synth_design`. This same technique can be applied on specific modules with the `KEEP_HIERARCHY` attribute in RTL.

Reduce MUXF Mapping



TIP: This optimization technique is automatically applied by the `report_qor_suggestions` Tcl command.

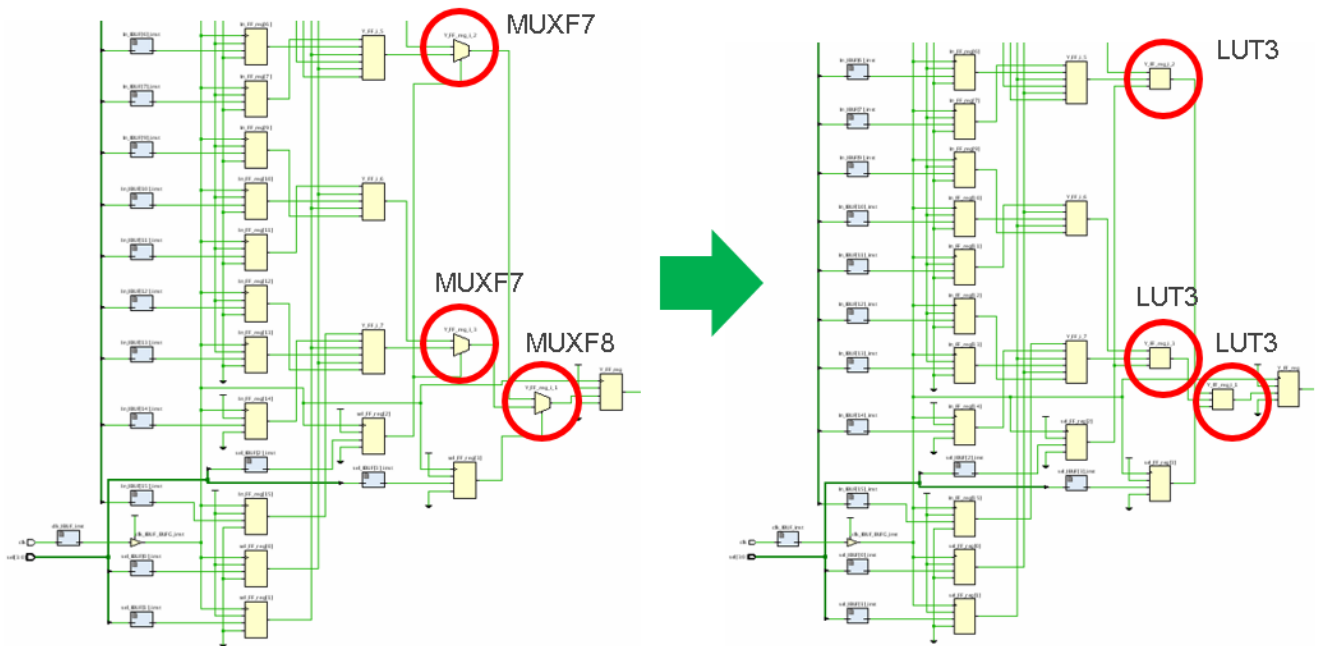
Using MUXF* primitives helps critical paths with many logic levels or a tight clock requirement while also reducing power. MUXF* includes MUXF7, MUXF8, and MUXF9, which are dedicated multiplexer resources located within the CLB. These resources are grouped with up to eight LUTs during placement. This grouping forces high CLB input utilization with higher routing demand and limits placement flexibility when the netlist connectivity is complex, leading to potential higher routing congestion and timing degradation.

In addition, the `opt_design` command provides an optional MUX optimization phase to remap MUXF* structures to LUT3 primitives to improve routability. You can use the `-muxf_remap` option to remap all of the MUXF* cells. Alternatively, set the `MUXF_REMAP` property to `TRUE` on a select number of cells in the congested region to limit the scope of the MUX remapping. Any MUXF* cells with the `MUXF_REMAP` property set to `TRUE` automatically trigger the MUX optimization phase during `opt_design` and are remapped to LUT3s.

Note: Disabling these resources can result in increased power. Use this method only when needed to achieve timing closure.

The following figure shows a 16-1 MUX before and after the MUXF* optimization.

Figure 155: Netlist Before and After MUX Optimization



To further optimize the netlist after performing MUX optimization, use the `-remap` option with the `-muxf_remap` option. This combines the LUT3 primitives that are generated by the MUXF* optimization with connected logic if possible.

You can determine whether timing closure is impacted by routing congestion by reviewing the Router Initial Estimated Congestion table in the log files or in the Design Analysis report (`report_design_analysis -congestion`) after place or route is complete.

In the following figure, the Design Analysis report shows that 7% of the device is impacted by Short congestion level 5 (32x32 CLBs) in the South direction while 26% MUXF are utilized in the corresponding congested area.

Figure 156: South Short Congestion in the report_design_analysis Congestion Table

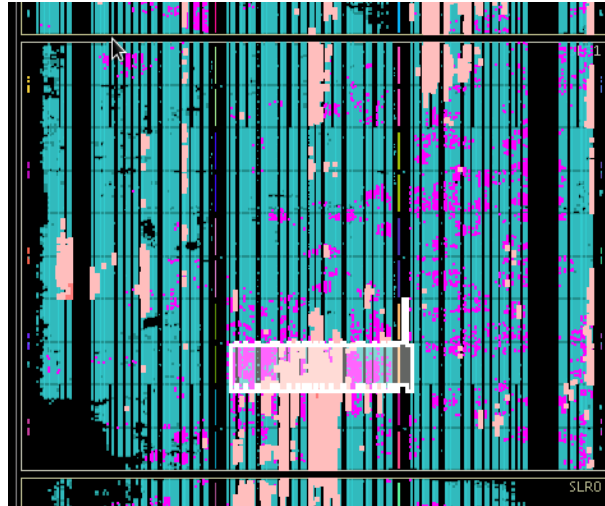
Direction	Type	Congestion Level	Percentage Tiles	Congestion Window	Cell Names	Combined LUTs	LUT6	MUXF
North	Short	4	6.983%	(CLEL_L_X48Y73,CLEL_R_X63Y88)	inst_name1(92%)	0%	22%	4%
South	Short	5	7.136%	(CLEL_L_X32Y297,CLEL_R_X63Y328)	inst_name2(99%)	2%	49%	26%
East	Short	4	6.005%	(CLEL_L_X48Y109,CLE_M_X63Y125)	inst_name3(94%)	0%	38%	10%
West	Short	4	6.541%	(CLEL_L_X32Y273,CLE_M_X47Y320)	inst_name4(92%)	1%	48%	23%

In the Vivado IDE, you can select a row in the table of the Design Analysis congestion report to highlight the corresponding congested area in the Device window. The following figure shows that the congestion overlaps with a higher MUXF density area. The MUXF cells are highlighted in magenta using the following command in the Vivado IDE Tcl Console:

```
highlight_objects -color magenta [get_cells -hier -filter REF_NAME=~MUXF*]
```

MUXF* includes MUXF7/MUXF8/MUXF9, which are dedicated multiplexer resources located within the CLB. These resources are grouped with up to 8 LUTs during placement, forcing high CLB input utilization with higher routing demand and limiting placement flexibility. The estimated congestion per CLB is displayed using the Vivado IDE metrics.

Figure 157: MUXF Congestion Highlighted in the Vivado IDE Device Window



When high MUXF* utilization overlaps with areas of higher congestion, Xilinx recommends reducing the number of MUXF* by mapping their corresponding functionality to LUTs, which have higher placement and routing flexibility. You can use the following command in the XDC synthesis constraints to modify the netlist:

```
set_property BLOCK_SYNTH.MUXF_MAPPING 0 [get_cells inst_name4]
```

After rerunning synthesis, place, and route, the updated congestion table in the Design Analysis report now shows that the South Short congestion is lower (level 4), which typically improves the timing quality of results.

Figure 158: Initial Router Congestion Table after Reducing MUXF Usage on a Module

Direction	Type	Congestion Level	Percentage Tiles	Congestion Window	Cell Names	Combined LUTs	LUT6	MUXF
North	Short	4	6.983%	(CLEL_L_X48Y73,CLEL_R_X63Y88)	inst_name1(92%)	0%	22%	4%
South	Short	4	9.040%	(CLEL_L_X34Y297,CLEL_R_X49Y312)	inst_name2(99%)	2%	54%	4%
East	Short	4	6.005%	(CLEL_L_X48Y109,CLE_M_X63Y125)	inst_name3(94%)	0%	38%	10%
West	Short	4	6.541%	(CLEL_L_X32Y273,CLE_M_X47Y320)	inst_name4(92%)	1%	48%	23%

Disable LUT Combining

Note: This optimization technique is automatically applied by the `report_qor_suggestions Tcl` command.

LUT combining reduces logic utilization by combining LUT pairs with shared inputs into single dual-output LUTs that use both O5 and O6 outputs. However, LUT combining can potentially increase congestion because it tends to increase the input/output connectivity for the slices. If LUT combining is high in the congested area (> 40%), you can try using a synthesis strategy that eliminates LUT combining to help alleviate congestion. The `Flow_AlternateRoutability` synthesis strategy and directive instructs the synthesis tool to not generate any additional LUT combining.

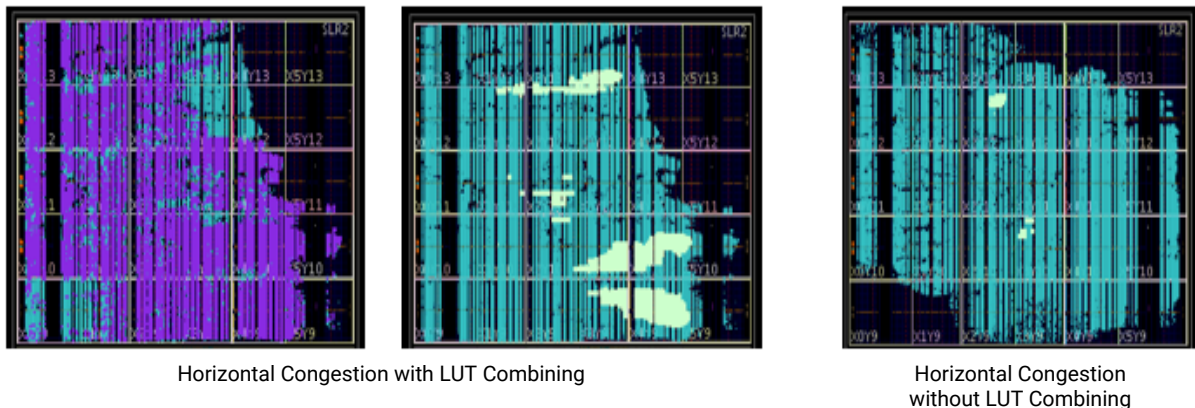
Note: If you are using Synplify Pro for synthesis, you can use the **Enable Advanced LUT Combining** option in the Implementation Options under the Device tab. This option is on by default. If you are modifying the Synplify Pro project file (*.prj), the following is specified: `set_option -enable_prepacking 1`.

You can use the following command to select cells with LUT combining enabled in your design:

```
select_objects [get_cells -hier -filter {SOFT_HLUTNM != "" || HLUTNM != ""}]
```

The following figure shows the horizontal congestion of a design with and without LUT combining. The cells with LUT combining are highlighted in purple.

Figure 159: Effect of LUT Combining on Horizontal Congestion



X18040-120519

To disable LUT combining on a module that overlaps with areas of higher congestion, use the following Tcl command:

```
reset_property SOFT_HLUTNM [get_cells -hierarchical -filter {NAME =~ <module name> && SOFT_HLUTNM != ""}]
```

Limit High-Fanout Nets in Congested Areas

Note: This optimization technique is automatically applied by the `report_qor_suggestions` Tcl command.

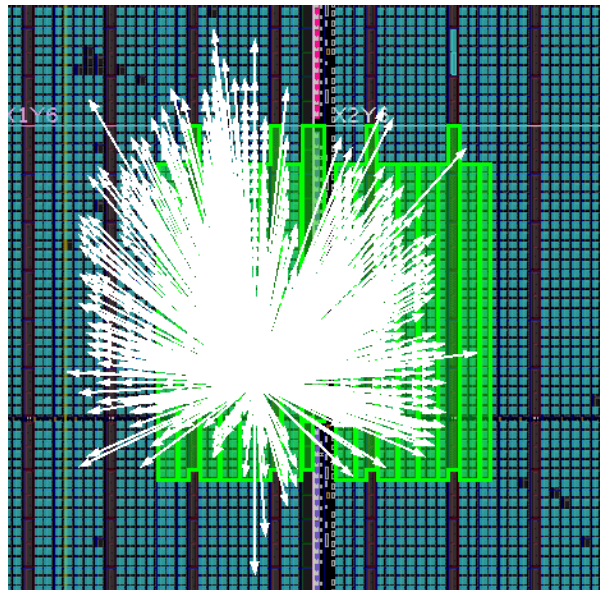
High fanout nets that have tight timing constraints require tightly clustered placement to meet timing. This can cause localized congestion as shown in the following figure. High fanout nets can also contribute to congestion by consuming routing resources that are no longer available for other nets in the congestion window.

To analyze the impact of high fanout non-global nets on routability in the congestion window you can:

- Select the leaf cells of the top hierarchical modules in the congestion window.
- Use the find command (**Edit → Find**) to select all of the nets of the selected cell objects (filter out Global Clocks, Power, and Ground nets).
- Sort the nets in decreasing Flat Pin Count order.
- Select the top fan-out nets to show them in relation to the congestion window.

This can quickly help you identify high-fanout nets which potentially contribute to congestion.

Figure 160: High-Fanout Nets in Congestion Window



For high fanout nets with tight timing constraints in the congestion window, replicating the driver will help relaxing the placement constraints and alleviate congestion.

High fanout nets (fanout > 5000) with sufficient positive timing slack can be routed on global clock resources instead of fabric resources. The placer automatically routes high fanout nets with fanout > 1000 on global routing resources if those resources are available towards the end of the placer step. This optimization only occurs if it does not degrade timing.

You can also set the property `CLOCK_BUFFER_TYPE=BUFG` on the net and let synthesis or logic optimization automatically insert the buffer prior to the placer step. Review the newly inserted buffer placement along with its driver and loads placement after `place_design` to verify that it is optimal. If it is not optimal, use the `CLOCK_REGION` constraint (UltraScale devices only) or `LOC` constraint (7 series devices only) on the clock buffer to control its placement.

Use Cell Bloating

You can use cell bloating to insert whitespace (increased cell spacing) during the `place_design` step. This leads to a lower density of cells in a given area of the die, which can reduce congestion by increasing available routing. This technique is particularly effective in small, congested areas of relatively high-performance logic.

To use cell bloating, apply the `CELL_BLOAT_FACTOR` property to hierarchical cells and set the value to `LOW`, `MEDIUM`, or `HIGH`. When working with smaller modules of several hundred cells, `HIGH` is the recommended setting.



CAUTION! *If the device already uses too many routing resources, cell bloating is not recommended. In addition, using cell bloating on larger cells might force placed cells to be too far apart.*

Tuning the Compilation Flow

The default compilation flow provides a quick way to obtain a baseline of the design and start analyzing the design if timing is not met. After initial implementation, tuning the compilation flow might be required to achieve timing closure.

Using Strategies and Directives

You can use strategies and directives to find the optimal solution for your design. Strategies are applied globally to a project implementation run. Directives can be set individually for each step of the implementation flow in both Project and Non-Project Modes.

ML Strategies

Machine learning (ML) strategies allow you to quickly obtain an optimized strategy for your design. If you are running multiple implementation strategies to generate implementation results, you can use ML strategies instead to help you predict which results are most likely to generate a good result.



RECOMMENDED: *Xilinx recommends performing three implementation runs with different strategy suggestions to identify and address errors in the prediction.*

You can generate strategy suggestion objects on a routed design by running the `report_qor_suggestions` command. Prior to running this command, you must run the implementation flow as follows:

- In Project Mode, use the Default or PerformanceExplore strategy.

- In Non-Project Mode, use the following Tcl commands:
 - `opt_design`: Set the `-directive` option to `Default` or `Explore`.
 - `place_design`, `phys_opt_design`, and `route_design`: Set the `-directive` option to `Default` or `Explore`. The option must match across all three Tcl commands.

After generating ML strategy suggestions, you must write the suggestions using `write_qor_suggestions -strategy_dir <directory>`. This writes one RQS file per strategy. To activate strategy objects, an RQS file with the strategy suggestion must be read using `read_qor_suggestions` prior to running `opt_design`, and the directives for all commands must be set to RQS (for example, `opt_design -directive RQS`).

Xilinx recommends the following when using ML strategies:

- For best results, resolve all methodology checks, and make sure the design has a QoR assessment score of three or higher. To verify, run `report_qor_assessment` after `synth_design` or `opt_design`.
- To further enhance performance, combine ML strategy suggestions with other QoR suggestions in the same RQS file.

Note: ML strategy suggestions are combined automatically when QoR suggestions are written. To disable this feature, use `write_qor_suggestions -of_objects [get_qor_suggestions ...]`, and filter only the desired suggestions.

For more information, see this [link](#) in the *Vivado Design Suite User Guide: Design Analysis and Closure Techniques (UG906)*.

Predefined Strategies

Xilinx provides a set of predefined strategies that are tuned to be effective solutions for the majority of designs.

Note: Xilinx does not recommend running the SSI technology strategies for a non-SSI technology device.

Custom Strategies

If timing cannot be met with the predefined strategies, you can manually explore a custom combination of directives. Because placement typically has a large impact on overall design performance, it can be beneficial to try various placer directives with only the I/O location constraints and with no other placement constraints. By reviewing both WNS and TNS of each placer run (these values can be found in the placer log), you can select two or three directives that provide the best timing results as a basis for the downstream implementation flow.



TIP: For a list of directives and a short description of their functions, enter the implementation command followed by the `-help` option (for example, `place_design -help`). For information on strategies, see this [link](#) in the *Vivado Design Suite User Guide: Implementation (UG904)*.

For each of these checkpoints, several directives for `phys_opt_design` and `route_design` can be tried and again only the runs with the best estimated or final WNS/TNS should be kept. In Non-Project Mode, you must explicitly describe the flow with a Tcl script and save the best checkpoints. In Project Mode, you can create individual implementation runs for each placer directive, and launch the runs up to the placement step. You would continue implementation for the runs that have the best results after the placer step (as determined by the Tcl-post script).

Physical constraints (Pblocks and DSP and RAM macro constraints) can prevent the placer from finding the most optimal solution. Xilinx therefore recommends that you run the placer directives without any Pblock constraints. The following Tcl command can be used to delete any Pblocks before placement with directives commences:

```
delete_pblock [get_pblocks *]
```

Running `place_design -directive <directive>` and analyzing placement of the best results can also provide a template for floorplanning the design or reusing the placement of block RAM macros or DSP macros, which can stabilize the flow from run to run.

Using Optimization Iterations

Sometimes it is advantageous to iterate through a command multiple times to obtain the best results. For example, it might be helpful to first run `phys_opt_design` with the `force_replication_on_nets` option to optimize some critical nets that appear to have an impact on WNS during route.

Next, run `phys_opt_design` with any of the directives to improve the overall WNS of the design.

In Non-Project Mode, use the following commands:

```
phys_opt_design -force_replication_on_nets [get_nets -hier *phy_reset*]
phys_opt_design -directive <directive name>
```

In Project Mode, the same results can be achieved by running the first `phys_opt_design` command as part of a Tcl-pre script for a `phys_opt_design` run step which will run using the `-directive` option.

Overconstraining the Design

When the design fails timing by a small amount after route, it is usually due to a small timing margin after placement. It is possible to increase the timing budget for the router by tightening the timing requirements during placement and physical optimization. To accomplish this, Xilinx recommends using the `set_clock_uncertainty` constraint for the following reasons:

- It does not modify the clock relationships (clock waveforms remain unchanged).
- It is additive to the tool-computed clock uncertainty (jitter, phase error).
- It is specific to the clock domain or clock crossing specified by the `-from` and `-to` options.

- It can easily be reset by applying a null value to override the previous clock uncertainty constraint.

In any case, Xilinx recommends that you:

- Overconstrain only the clocks or clock crossing that cannot meet setup timing.
- Use the `-setup` option to tighten the setup requirement only.

Note: If you do not specify this option, both setup and hold requirements are tightened.

- Reset the extra uncertainty before running the router step.

Overconstraining Example

A design misses timing by -0.2 ns on paths with the `clk1` clock domain and on paths from `clk2` to `clk3` by -0.3 ns before and after route.

1. Load netlist design and apply the normal constraints.
2. Apply the additional clock uncertainty to overconstrain certain clocks.
 - a. The value should be at least the amount of violation.
 - b. The constraint should be applied only to setup paths.

```
set_clock_uncertainty -from clk0 -to clk1 0.3 -setup
set_clock_uncertainty -from clk2 -to clk3 0.4 -setup
```

3. Run the flow up to the router step. It is best if the pre-route timing is met.
4. Remove the extra uncertainty.

```
set_clock_uncertainty -from clk0 -to clk1 0 -setup
set_clock_uncertainty -from clk2 -to clk3 0 -setup
```

5. Run the router.

After running the router, you can review the timing results to evaluate the benefits of overconstraining. If timing was met after placement but still fails by some amount after route, you can increase the amount of uncertainty and try again.



RECOMMENDED: Do not overconstrain beyond 0.5 ns. Overconstraining the design can result in increased power for the implementation as well as an increase in run time.



TIP: An alternative to overconstraining the design is to change the relative priority of each path group. By default, each clock and user-defined path group is analyzed independently with the same priority during implementation. You can set a higher priority for any clock-based path group using the `group_path_weight 2 -name <ClockName>` options. The priority of user-defined path groups cannot be changed.

Considering Floorplan

Floorplanning allows you to guide the tools, either through high-level hierarchy layout, or through detail placement. This can provide improved QoR and more predictable results. You can achieve the greatest improvements by fixing the worst problems or the most common problems. For example, if there are outlier paths that have significantly worse slack, or high levels of logic, fix those paths first by grouping them in a same region of the device through a Pblock. Limit floorplanning only to portions of design that need additional user intervention, rather than floorplanning the entire design.

Floorplanning logic that is connected to the I/O to the vicinity of the I/O can sometimes yield good results in terms of predictability from one compilation to the next. In general, it is best to keep the size of the Pblocks to a clock region. This provides the most flexibility for the placer. Avoid overlapping Pblocks, as these shared areas can potentially become more congested. Where there is a high number of connecting signals between two Pblocks consider merging them into a single Pblock. Minimize the number of nets that cross Pblocks.



TIP: When upgrading to a newer version of the Vivado Design Suite, first try compiling without Pblocks or with minimal Pblocks (i.e., only SLR level Pblocks) to see if there are any timing closure challenges. Pblocks that previously helped to improve the QoR might prevent place and route from finding the best possible implementation in the newer version of the tools.

For SSI technology devices, you can also consider using SLR Pblocks or soft floorplanning constraints (USER_SLR_ASSIGNMENT).

Related Information

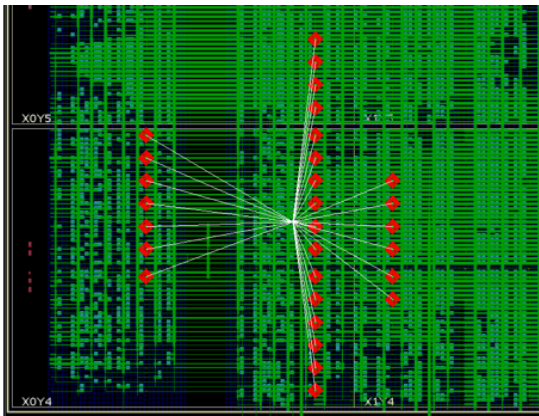
[SSI Technology Considerations](#)

Grouping Critical Logic

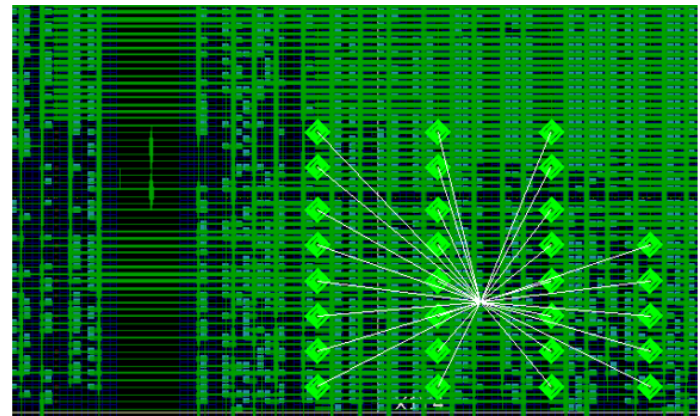
Grouping critical logic to avoid crossing SLR or I/O columns can help improve the critical path of a design. The following figure shows two examples of a large FIFO implemented with 29 FIFO36E2 primitives. The critical path is from the WRRSTBUSY pin of every FIFO36E2 in the group through 5 LUTs to the WREN pin of every FIFO36E2 in the group.

- On the left, the example shows that the placer was unable to find the most optimal placement of the path, because block RAM utilization was high. FIFO36E2 primitives are marked in red.
- On the right, the example shows that the placer was able to meet timing, because the FIFO36E2 blocks were grouped in a rectangle that avoided the configuration column crossing. FIFO36E2 primitives are marked in green.

Figure 161: Locations Avoiding the Configuration Column



Locations Not Avoiding the Configuration Column



Preassigned Locations Avoiding the Configuration Column

X18041-120219

Reusing Placement Results

It is fairly easy to reuse the placement of block RAM macros and DSP macros. Reusing this placement helps to reduce the variability in results from one netlist revision to the next. These primitives generally have stable names. The placement is usually easy to maintain. Some placement directives result in better block RAM and DSP macro placement than others. You can try applying this improved macro placement from one placer run to others using different placer directives to improve QoR. Following is a simple Tcl script that saves block RAM placement into an XDC file for UltraScale and UltraScale+ device designs.

```
set_property IS_LOC_FIXED 1 \
    [get_cells -hier -filter {PRIMITIVE_TYPE =~ BLOCKRAM.*.*}]
write_xdc bram_loc.xdc -exclude_timing
```

You can edit the `bram_loc.xdc` file to only keep block RAM location constraints and apply it for your consecutive runs.



IMPORTANT! Do not reuse the placement of general slice logic. Do not reuse the placement for sections of the design that are likely to change. Use the Incremental Compile flow if you make small changes to the design and want to re-use prior placement to achieve more predictable results and faster compile time.

Using Incremental Implementation

You can use incremental implementation to reduce implementation compile time and produce more predictable results. Xilinx recommends making incremental implementation part of your standard timing closure strategies. For more information, see this [link](#) in the *Vivado Design Suite User Guide: Implementation (UG904)* and this [link](#) in the *Vivado Design Suite User Guide: Design Analysis and Closure Techniques (UG906)*.

This section covers recommendations for automatic incremental implementation, including both high and low reuse modes.

Choose a High Quality Reference Checkpoint

Because the incremental implementation flow depends on reuse, the most important input to the flow is the reference checkpoint. When you use automatic incremental implementation in Project Mode, the Vivado tools manage the updating of the reference checkpoint. This ensures that reuse is high and timing is almost closed.

In all other use cases of the incremental implementation flow, you have control over the selection of the reference checkpoint. Following are guidelines to help improve your selection of the reference checkpoint:

- Use a reference checkpoint that meets timing or is close to meeting timing. If the reference checkpoint is close to meeting timing, it might be beneficial to improve timing as follows before running the incremental implementation flow.

Note: For automatic incremental implementation, the checkpoint is rejected unless WNS is less than -0.250 ns.

- Run `route_design -tns_cleanup` to optimize paths that are not the worst case path.
- Run the post-route `phys_opt_design` command to improve timing failures. Although this command might increase run time, these optimizations are replayed quickly in the incremental implementation run.
- Use the `report_qor_suggestions` command to generate suggestions to improve the design. New suggestions applied in the incremental implementation flow must be incremental implementation-friendly. Suggestions already applied in the reference checkpoint do not need to be incremental implementation-friendly. For suggestions that are not incremental implementation friendly, consider applying the suggestions and updating the checkpoint using the default flow.
- Select the checkpoint with the lowest congestion, which more readily accommodates changes than congested checkpoints.
- Maximize matching between reference and incremental checkpoints.

Note: For automatic incremental implementation, the checkpoint is rejected unless cell matching is at least 94% and net matching is at least 90%.

- Use incremental synthesis to reduce changes introduced into the netlist due to RTL changes. Enable incremental synthesis early in the design closure cycle rather than waiting until you are ready to use incremental implementation.
- Ensure that `synth_design` and `opt_design` options match for the reference checkpoint and the incremental implementation runs.
- Match tool versions. Although this is not a requirement, thresholds change and new optimizations are added, which can lead to reduced matching.
- Avoid using `opt_design AddRemap` and `ExploreWithRemap` directives unless these are the only directives that close timing. These directives have reduced naming consistency when changes are introduced to the codebase.

- Use `report_qor_assessment` to determine whether the design is ready for the incremental implementation flow to be run and whether it is preferable to switch from the default flow.



TIP: To adjust the incremental implementation thresholds, run `config_implementation -help` for information. To identify differences between the reference and the incremental checkpoints, run `report_incremental_reuse`.

Select Incremental Implementation Directives for High Reuse Mode

You can adjust the incremental implementation flow behavior using directives. The tools follow these directives when the incremental implementation algorithms are used on the implementation run. When flow reverts to the default algorithms, the tools follow the directives specified with the `place_design`, `phys_opt_design`, and `route_design` commands.

Following are the directives available for use with the incremental implementation flow:

- **RuntimeOptimized:** Targets the WNS from the reference checkpoint. This helps maintain consistency with the reference checkpoint and improves placer and router run time by at least 2x. If the reference checkpoint does not close timing, this directive does not attempt to close timing. This directive is the default.
- **TimingClosure:** Targets WNS = 0.000 ns. Use this directive when the reference run is very close to meeting timing, and you are willing to trade off consistency in results and run time with more effort to try to meet timing. This mode can improve WNS by up to 250 ps on difficult designs. Use this directive with QoR Suggestions for the best chance at closing timing. There is usually a run time hit with this directive.
- **Quick:** This option is intended for designs that easily meet timing with greater than 99% reuse. Typically, this option is used for ASIC emulation and prototype designs with minor changes that do not impact timing.

Following is an example command for Project Mode:

```
set_property -name INCREMENTAL_CHECKPOINT.MORE_OPTIONS -value {-directive
TimingClosure} -object [get_runs <runName>]
```

Following is an example command for Non-Project Mode:

```
read_checkpoint -incremental -directive TimingClosure <reference>.dcp
```

Note: The `RuntimeOptimized` directive replaces the `Default` mapping directive, and the `TimingClosure` directive replaces the `Explore` mapping directive from previous Vivado Design Suite releases.

Reduce QoR Variability for Low Reuse Mode

In low reuse mode, you can reuse particular cells (for example, a hierarchical cell in the design) or cell types (for example, DSPs or block RAMs). This can be effective when both of the following are true:

- Some design runs are showing that a design can meet timing but many runs do not.
- It is early in the design flow or significant changes are still being made.

Reusing hierarchical cells is effective when placement of a particular cell is influencing the WNS significantly. Reusing DSPs, block RAMs, or both is useful in designs that have a relatively high density of these blocks.

To reuse particular cell or cell types:

- Analyze the reference runs, including checking failing checkpoints to identify the difference between good and bad runs.
 - Identify runs that have a good WNS and low congestion levels.
 - Use floorplanning to define SLR placement.
- After determining the area to target, compare a set of runs using low reuse mode against a baseline set of runs using the default flow to evaluate effectiveness.
 - Use different `place_design` directives to generate multiple results for comparison.

Note: In low reuse mode, incremental implementation directives are ignored, and target WNS is always 0.000 ns.

To reuse only block memory placement, use the following Tcl script:

```
read_checkpoint -incremental routed.dcp \
-reuse_objects [all_rams] -fix_objects [all_rams]
```

To reuse only DSP placement, use the following Tcl script:

```
read_checkpoint -incremental routed.dcp \
-reuse_objects [all_dsps] -fix_objects [all_dsps]
```

To reuse both Block Memory and DSP placement, use the following Tcl script:

```
read_checkpoint -incremental routed.dcp \
-reuse_objects [all_rams] -reuse_objects [all_dsps] -fix_objects
[current_design]
```

To reuse hierarchy in a particular hierarchical cell and all hierarchies below the cell, use the following Tcl script:

```
read_checkpoint -incremental routed.dcp \
-only_reuse [get_cells <cell_name>] -fix_objects [get_cells <cell_name>]
```



RECOMMENDED: When reusing a hierarchical module, Xilinx recommends using out-of-context synthesis or incremental synthesis with a `PRESERVE_BOUNDARY` constraint to ensure cell matching is 100%.

Avoid Floorplanning and Overconstraining

When using the incremental implementation flow, avoid the following:

- Do not floorplan incremental implementation runs.
Pblock placement is overridden by reference checkpoint placement.
- Do not overconstrain the placer.
Overconstraining the design in the incremental implementation run can severely impact reuse, because the tools try to meet a target WNS that is artificially altered.

Related Information

[Overconstraining the Design](#)

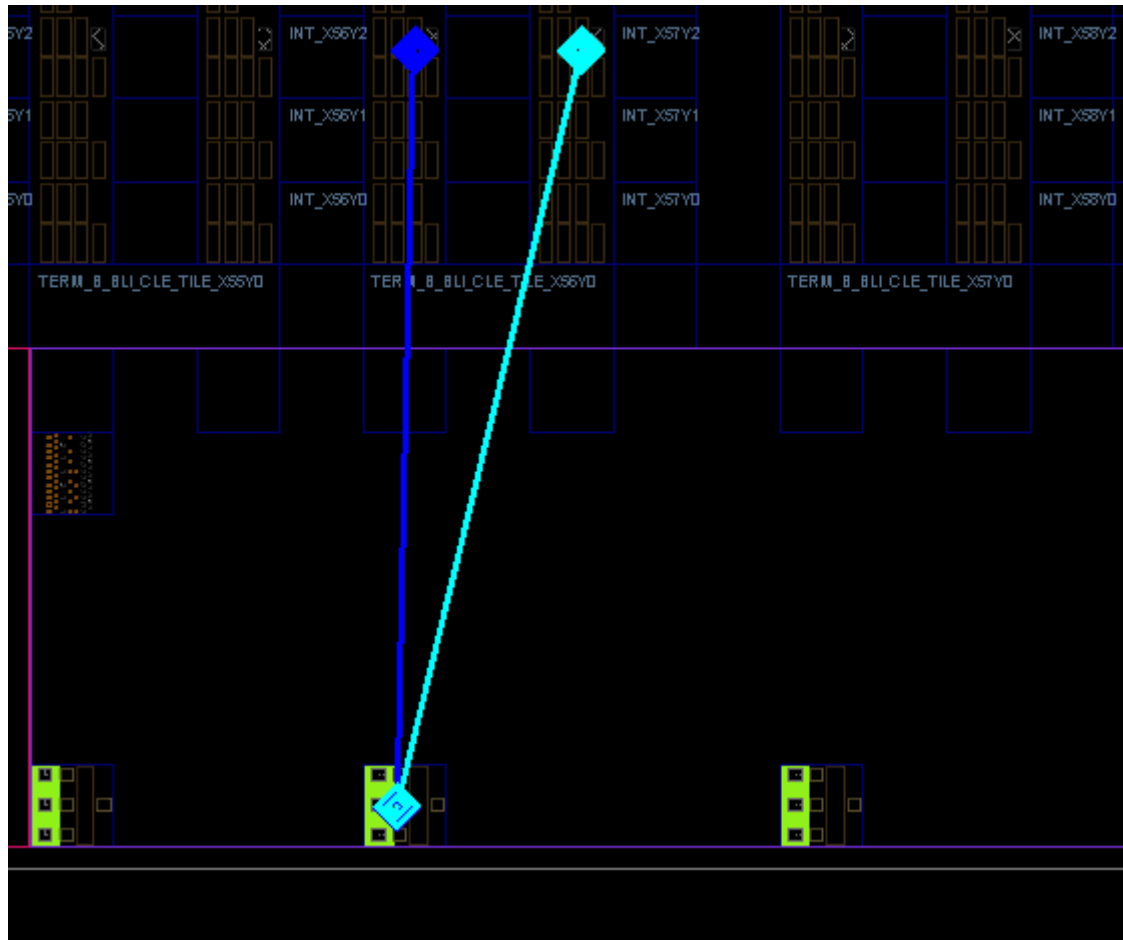
XPIO-PL Interface Techniques for Timing

Boundary logic interface flip-flops exist in hardware between the XPIO-programmable logic (PL) interface, which you can use to improve timing. Dedicated blocks in the XPIO such as the XPHY Logic, I/O Logic, and clock-modifying blocks have boundary logic interface flip-flops. You can apply boundary logic interface (BLI) constraints to flip-flops in your design to automatically take advantage of this hardware feature during design placement. In this example, the data paths to and from the I/O Logic cells ODDRE1 and IDDRE1 in the XPIO are taking advantage of the BLI FFs.

```
set_property BLI TRUE [get_cells {oddr_D1_BLI_reg oddr_D2_BLI_reg}]  
set_property BLI TRUE [get_cells {iddr_Q1_BLI_reg iddr_Q2_BLI_reg}]
```

The following figure shows the resulting placement and connectivity from setting the BLI property to TRUE.

Figure 162: Placement of XPIO-PL Interface BLI Flip-Flops for ODDRE1 and IDDRE1



SSI Technology Considerations

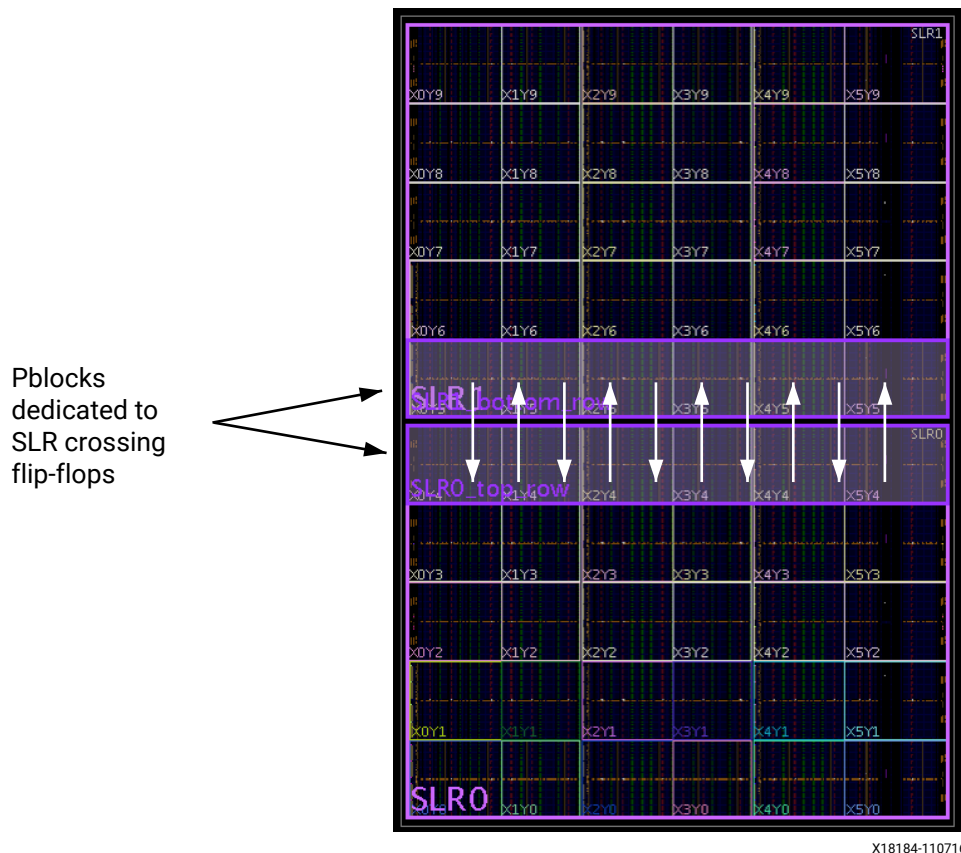
Stacked silicon interconnect (SSI) technology devices consist of multiple super logic regions (SLRs), joined by an interposer. The interposer connections are called super long lines (SLLs). There is some delay penalty when crossing from one SLR to another. To minimize the impact of the SLL delay on your design, floorplan the design so that SLR crossings are not part of the critical path. Minimizing SLR crossings through floorplanning by keeping a challenging module within one SLR only can also improve timing and routability of the design targeting SSI technology devices.

Using Hard SLR Floorplan Constraints

For high-performance designs, sufficient pipelining between the major hierarchies is required to ease global placement and SLR partitioning. When a design is challenging, SLR crossing points can change from run to run. In addition to defining SLR Pblocks, you can create additional Pblocks that are aligned to clock regions and located along the SLR boundary to constrain the crossing flip-flops. The following example shows an UltraScale ku115 SSI device with the following Pblocks:

- 2 SLR Pblocks: SLR0 and SLR1
- 2 SLR-crossing Pblocks: SLR0_top_row and SLR1_bottom_row

Figure 163: SLR-Crossing Pblock Example



IMPORTANT! Xilinx recommends using `CLOCKREGION` ranges instead of `LAGUNA` ranges for SLR-crossing Pblocks.



TIP: You can define SLR Pblocks by specifying a complete SLR. For example, `resize_pblock pblock_SLR0 -add SLR0`.

For more information, see this [link](#) in *Vivado Design Suite User Guide: Design Analysis and Closure Techniques (UG906)*.



VIDEO: For information on using floorplanning techniques to address design performance issues, see the [Vivado Design Suite QuickTake Video: Design Analysis and Floorplanning](#).

Using Soft SLR Floorplan Constraints

For large designs, logic for most of the major blocks fits in one SLR as expected and closes timing after a few design iterations. However, small portions of the logic, especially the connectivity across major blocks and across SLRs, is subject to QoR variation depending on the overall design placement. In such cases, the placer and physical optimization algorithms need additional flexibility to replicate or move some of the logic to a different SLR to address placement challenges and close timing.

You can use the `USER_SLR_ASSIGNMENT` property to floorplan the design by assigning large design blocks to SLRs. Set this property to a string value, which is applied to hierarchical cells and ignored on leaf cells. The value you set for this property influences the logic partitioning as follows:

- **SLR name:** When a hierarchical cell is assigned the name of an SLR (SLR0, SLR1, SLR2, etc.), the placer attempts to place the entire cell within the specified SLR.
- **String value:** When a hierarchical cell is assigned an arbitrary string value, the placer chooses the SLR. This prevents cells from being partitioned into multiple SLRs.

Note: If multiple cells have the same `USER_SLR_ASSIGNMENT` value, the placer attempts to group the cells in the same SLR.

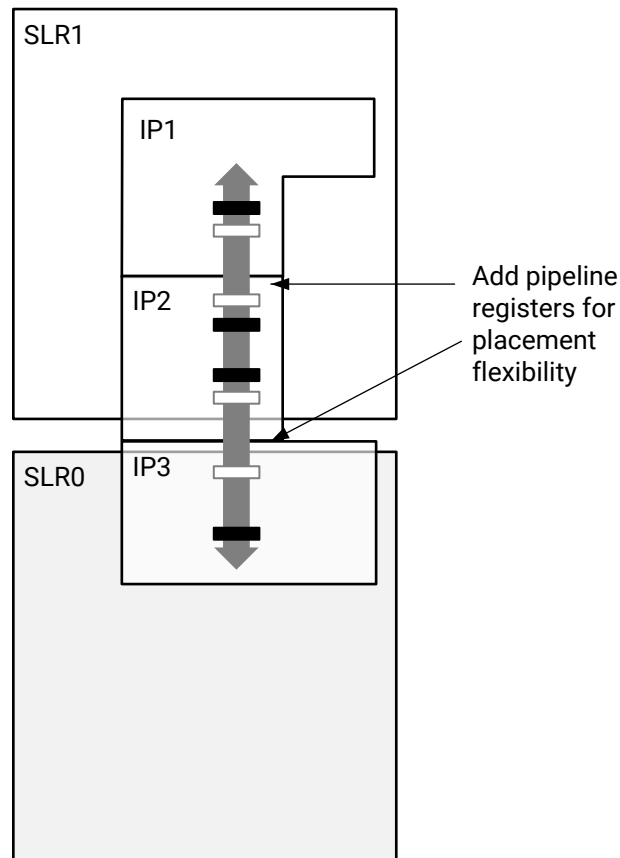
The `USER_SLR_ASSIGNMENT` property is a soft constraint during SLR partitioning while the Pblock is always a hard constraint during SLR partitioning and global placement. Unlike Pblocks, the `USER_SLR_ASSIGNMENT` can be ignored by the placer to find a valid SLR partitioning of the design. Both `USER_SLR_ASSIGNMENT` and Pblocks allow the detailed placer and physical optimization to make fine-tuned adjustments to leaf cell placement near the SLR boundaries to improve timing. These adjustments include moving pipeline registers across SLR boundaries if the moves improve timing. These register moves are not permitted across Pblock boundaries.

In the following example, a design contains three timing-critical hierarchical blocks with cell names IP1, IP2, and IP3 and targets a two-SLR device. To split the three blocks so that IP1 and IP2 are kept together in SLR1 while IP3 is placed in SLR0, the following XDC constraints are applied:

```
set_property USER_SLR_ASSIGNMENT SLR1 [get_cells {IP1 IP2}]
set_property USER_SLR_ASSIGNMENT SLR0 [get_cells IP3]
```

The following figure shows the resulting placement. To improve performance, you can incorporate extra pipeline stages to traverse distances within the device. This is particularly helpful along expected SLR crossings, between IP2 and IP3 in this example. During detail placement and `phys_opt_design`, the pipeline registers from IP2 and IP3 can automatically move across SLR boundaries if this improves timing.

Figure 164: Placement Example for the USER_SLR_ASSIGNMENT Property



X21199-121919

For cases in which you cannot set `USER_SLR_ASSIGNMENT` or the placer splits challenging paths across SLRs, you can use the `USER_CROSSING_SLR` property to direct where SLR crossings should or should not occur. Typically, you apply this property to nets or leaf pins where you want pins to be placed in the same SLR as the net driver, or where you want the SLR crossing for the case of a register chain. Set this property to a Boolean value, which is applied to nets and pins to constrain individual SLR crossings:

- **TRUE:** Indicates that the target net object should cross an SLR or the target pin object should be connected across an SLR. You can only apply the TRUE value to register-to-register connections with a single fanout in between.

Note: You cannot use the TRUE value for random logic. This option is useful for ensuring a chain of registers always crosses a SLR boundary on a specific register when trying multiple implementation strategies.

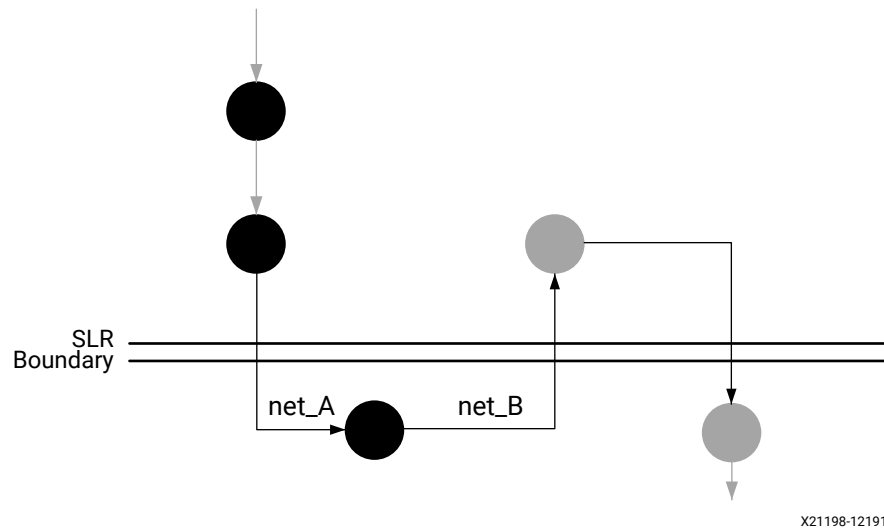
- **FALSE:** Indicates that the target net object should not cross an SLR or the target pin object should not be connected across an SLR. You can apply the FALSE value to any net or pin.

Note: Pins must not be inside macro primitives, because these pins are internal and cannot be constrained.

In the following example, a pipeline register chain crosses an SLR twice, resulting in an unintentional, inefficient zigzag path.

Note: In the next two figures, each dot represents a register stage.

Figure 165: Suboptimal SLR Crossings Before Setting the USER_CROSSING_SLR Property

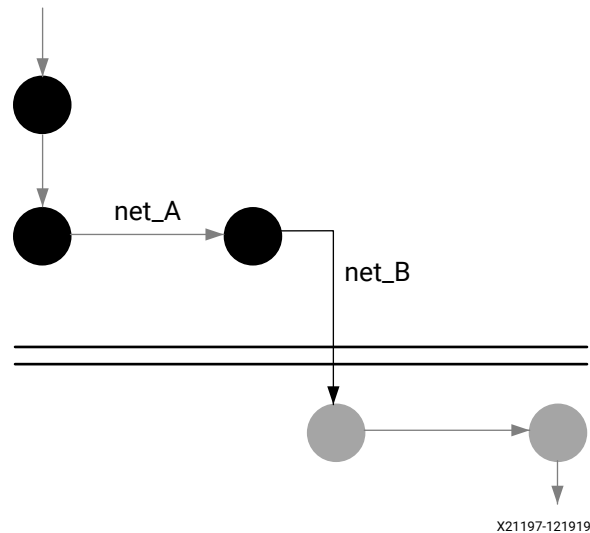


To achieve the optimal placement in which only net_B crosses the SLR, the following XDC constraints are applied:

```
set_property USER_CROSSING_SLR FALSE [get_pins -leaf -of [get_nets net_A]]
set_property USER_CROSSING_SLR TRUE [get_pins -leaf -of [get_nets net_B]]
```

The resulting placement contains just a single SLR crossing on net_B as shown in the following figure.

Figure 166: Optimal SLR Crossings After Setting the USER_CROSSING_SLR Property



Using SLR Crossing Registers

When targeting UltraScale+ SSI technology devices, you can map a register-to-register SLR crossing to a Laguna TX_REG driving a Laguna RX_REG directly. This type of connection is only possible in the UltraScale+ device family, where the Vivado router can fix hold time violations by setting local programmable clock delays. Using the TX_REG to RX_REG SLR crossing topology for pipeline register crossings offers the following performance advantages:

- The placement of SLR crossings spreads vertically, reducing routing congestion near SLR boundaries.
- Locating registers in Laguna sites improves delay estimation accuracy, resulting in higher timing QoR.
- SLR-crossing performance becomes faster and more consistent.

Note: When targeting UltraScale SSI technology devices, you can only use a Laguna TX_REG or RX_REG on a SLR crossing net, and you cannot use both at the same time. Performance advantages are similar to the ones listed above.

You can set the USER_SLL_REG property on registers that you expect to be placed at an SLR crossing boundary on a Laguna register site. The USER_SLL_REG constraint is ignored by `place_design` if the register D and Q pins are connected to a net that either does not cross an SLR boundary or drives loads placed in multiple SLRs. For example:

```
set_property USER_SLL_REG TRUE [get_cells {reg_A reg_B}]
```


A reliable method of mapping registered crossings to Laguna is to apply both BEL and LOC constraints to the registers to lock them in place. The LOC value assigns the Laguna site, and the BEL value chooses a particular Laguna register inside the site, one of six TX_REG registers or one of six RX_REG registers. Laguna crossing registers are a fixed distance apart, which means that each TX_REG register is paired with an RX_REG register for a direct connection.

In the following example, a register-to-register connection is manually placed onto a TX_REG to RX_REG connection. Pipeline register reg_A drives a single fanout with the single load of register reg_B. For a VU9P target device, the following XDC constraints are applied so that reg_A in SLR2 drives reg_B in SLR1 using a direct TX_REG to RX_REG connection:

```
set_property BEL TX_REG3 [get_cells reg_A]
set_property BEL RX_REG3 [get_cells reg_B]
set_property LOC LAGUNA_X2Y480 [get_cells reg_A]
set_property LOC LAGUNA_X2Y360 [get_cells reg_B]
```

The BEL assignments are applied first, and the register position (0, 1, ... 5) must match between TX_REG and RX_REG, which is 3 for this example. Finally, the distance between paired Laguna sites is 120 rows. The register reg_A drives from the bottom row of the SLR2 Laguna column across to the bottom row of the SLR1 Laguna column. When creating LAGUNA BEL and LOC constraints, try grouping registers with same clock, clock enable and reset signals to avoid control set compatibility issues.

Using Auto-Pipelining for SLR Crossings

Whether you use soft SLR floorplan constraints, hard SLR floorplan constraints, or no floorplan constraints, the number of pipeline stages required to meet timing between major portions of the design located in different SLRs varies based on the following:

- Target frequency
- Device floorplan
- Device speed grade

You can leverage the auto-pipelining feature to allow the placer algorithms to decide on the number of required stages and their optimal location, which helps timing closure across SLR boundaries. When using this feature, the Vivado placer automatically uses Laguna registers without additional intervention.

You can enable auto-pipelining by setting AUTOPIPELINING_* attributes on buses and handshake logic in your RTL, but make sure that the additional latency does not adversely affect the design functionality. Alternatively, you can use the Xilinx AXI Register Slice Memory Mapped or Streaming IP, configured in the SLR crossing.

Related Information

[Auto-Pipelining Considerations](#)

Using Intelligent Design Runs

To automatically address most timing closure challenges during implementation, you can use an Intelligent Design Run (IDR). An IDR is a special type of implementation run that leverages `report_qor_suggestions`, ML-based strategy predictions, and incremental compile. An IDR can run up to 6 iterations of place and route, which leads to a typical compile time of 4.5 times that of a standard run. However, using IDR can provide significant benefits by reducing the knowledge required to close timing and by saving hours of user analysis.



RECOMMENDED: *Because an IDR takes longer than a standard implementation run, Xilinx recommends using IDRs less frequently than standard runs. For example, use an IDR after you resolve all methodology warnings and after trying a few common implementation strategies, such as default and explore.*



TIP: *To iterate more quickly, you can extract the QoR suggestions and ML strategies from the IDR for use in a standard implementation run. If a significant design change is made, rerun the IDR to update the associated files.*

An IDR comprises the following stages:

1. Uses `report_qor_suggestions` to apply optimization properties to elements in the design in a predetermined order.
2. Uses machine learning (ML) strategies to generate tool options for `opt_design`, `place_design`, `phys_opt_design`, and `route_design` that are optimized for the design.
3. Uses a Last Mile Timing Closure feature to apply extensive effort on paths that are difficult to resolve to get the final result.

To ensure success when using IDR, follow these requirements:

- The implementation must be project based. For non-project users, the easiest method is to create a post-synthesis netlist-based project using a `pre-opt_design` checkpoint.
- The device must be from either an UltraScale or UltraScale+™- device-based family.
- The design must have a baseline with accurate and achievable constraints.
- All designs must comply with the recommended methodology, as reported by the `report_methodology Tcl` command.
- An SLR-based floorplan might be required for SSI technology-based devices.
- Apply only automatic implementation suggestions. Text-based suggestions or suggestions with `APPLICABLE_FOR = synth_design` must be applied *before* starting an IDR.

For more information see this [link](#) in the *Vivado Design Suite User Guide: Design Analysis and Closure Techniques (UG906)*.

Power Closure

Given the importance of power, the Vivado tools support methods for obtaining an accurate estimate for power, as well as providing some power optimization capabilities. For additional information, see the *Vivado Design Suite User Guide: Power Analysis and Optimization (UG907)*.



RECOMMENDED: When targeting UltraScale and UltraScale+™ devices and using the Explore directives or Explore-based strategies, you must manually enable block RAM power optimization by running `power_opt_design` or using `opt_design -bram_power_opt` after `opt_design` runs. Xilinx recommends targeting block RAMs to achieve power reduction.

Estimating Power Throughout the Flow

As your design flow progresses through synthesis and implementation, you must regularly monitor and verify the power consumption to be sure that thermal dissipation remains within budget, that the board voltage regulators remain within their current operating limits and the design stays within any system power limits. You can then take prompt remedial actions if the power approaches your budget too closely.

Specify a power budget to report the power margin using the XDC constraint:

```
set_operating_conditions -design_power_budget <value in watts>
```

This value is used by the `report_power` command. The difference between the calculated on-chip power and the power budget is the power margin, which is displayed in red in the Vivado IDE if the power budget is exceeded. This makes it easier to monitor power consumption throughout the flow.



TIP: For UltraScale+ devices, you can export an XDC file from XPE that contains the environment settings, including the XPE estimate that can be used as a power budget constraint. You can override the power budget using either XPE or the XDC. Add the XDC constraints for power margin reporting.

The accuracy of the power estimates varies depending on the design stage when the power is estimated. To estimate power post-synthesis through implementation, run the `report_power` command, or open the Power Report in the Vivado IDE.

- **Post Synthesis:** The netlist is mapped to the actual resources available in the target device.
- **Post Placement:** The netlist components are placed into the actual device resources. With this packing information, the final logic resource count and configuration becomes available. This accurate data can be exported to the Xilinx® Power Estimator spreadsheet. This allows you to:
 - Perform what-if analysis in XPE.
 - Provide the basis for accurately filling in the spreadsheet for future designs with similar characteristics.

- **Post Routing:** After routing is complete all the details about routing resources used and exact timing information for each path in the design are defined.

In addition to verifying the implemented circuit functionality under best and worst case logic and routing delays, the simulator can also report the exact activity of internal nodes and include glitching. Power analysis at this level provides the most accurate power estimation before you actually measure power on your prototype board.

Using the Power Constraints Advisor

The Power Constraint Advisor reports the tool-computed switching activity on all control signals in the design and is sorted starting with highest fanout. Review this list for Low confidence levels, which indicate resets with high switching activity and enables with very low or zero switching activity. Both factors contribute to erroneously optimistic power results. For more information, see [Power Constraints Advisor](#) in the *Vivado Design Suite User Guide: Power Analysis and Optimization* (UG907).

Recommended Power Constraints

Applying the correct power constraints to a design is critical to design closure. The Vivado tools `report_power` command reports the power margin based on the budget applied as well as additional constraints. Following is a list of the minimum recommended constraints. For more information, see the *Vivado Design Suite User Guide: Power Analysis and Optimization* (UG907).

Minimum Recommended Constraints

The following constraints ensure that the power estimation checks the power budget and uses the worst-case maximum process for static power analysis:

```
set_operating_conditions -design_power_budget <Power in Watts>
set_operating_conditions -process maximum
```

Additional Recommended Constraints

The following constraints define the thermal solution and allow the `report_power` command to estimate the junction temperature and therefore, the static power more accurately:

```
set_operating_conditions -ambient_temp <max Ambient requested for
application is Celsius>
set_operating_conditions -thetaja <the rise in junction temperature for
every watt dissipated, obtained from thermal simulation, C/W>
```

The Vivado tools `report_power` command also allows you to report power on a on a per regulator or voltage regulator module (VRM) basis using the following constraints.

Creating a New Power Rail

```
create_power_rail <power rail name> -power_sources {supply1, supply2, ...}
```

Adding Power Sources to an Existing Power Rail

```
add_to_power_rail <power rail name> -power_sources {supply1, supply2, ..}
```

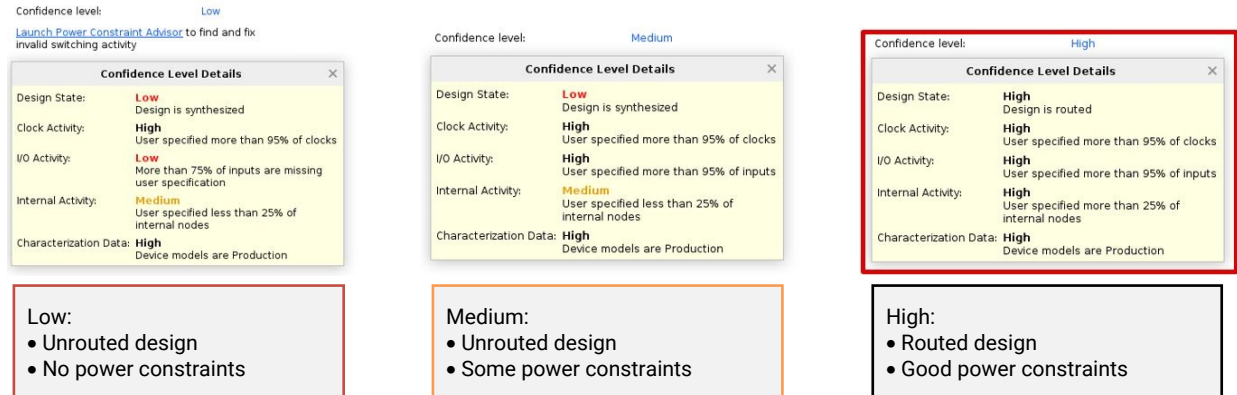
Defining Current Budget

```
set_operating_conditions -supply_current_budget {<supply rail name> <current budget in Amp>} -voltage {<supply rail name> <voltage>}
```

Best Practices for Accurate Power Analysis

For accurate power analysis, make sure you have accurate timing constraints, I/O constraints, and switching activity. The `report_power` command indicates a confidence level, as shown in the following figure. Target a High confidence level to ensure accurate power analysis. For more information, see this [link](#) in the *Vivado Design Suite User Guide: Power Analysis and Optimization (UG907)*.

Figure 167: Power Analysis Confidence Level



X25127-021621

Reviewing the Design Power Distribution After Running Power Analysis

You can review the total on-chip power and thermal properties as well as details of the power at the resource level to determine which parts of your design contribute most to the total power. For more information, see this [link](#) in the *Vivado Design Suite User Guide: Power Analysis and Optimization (UG907)*.



POWER TIP: Review and validate the decoupling requirement of the completed Vivado design against the current schematic/PCB. You can generate a `.xpe` file from Vivado tools `report_power` using the following Tcl commands:

```
set_operating_conditions -process maximum
```

```
set_operating_conditions -ambient_temp <max Ambient requested for  
application is Celsius>
```

```
set_operating_conditions -thetaja <the rise in junction temperature for  
every watt dissipated, obtained from thermal simulation, C/W>
```

```
report_power -xpe {C:/Design_Runs/Vivado_export.xpe} -name {Any_Name}
```

You can then import the `.xpe` file into XPE. The XPE Power Delivery sheet shows the decoupling requirement based on the power estimation and power delivery option.

Further Refining Control Signal Activity After Running Power Analysis

When SAIF-based annotation has not been used for accurate power analysis, you can fine-tune the power analysis after doing the first level analysis. For more information, see [Further Refining Control Signal Activity](#) in the *Vivado Design Suite User Guide: Power Analysis and Optimization (UG907)*.

Power Optimization

If the power estimates are outside the budget, you must follow the steps described in the following sections to reduce power.

Analyzing Your Power Estimation and Optimization Results

Once you have generated the power estimation report using `report_power`, Xilinx recommends the following:

- Examine the total power in the Summary section. Does the total power and junction temperature fit into your thermal and power budget?
- If the results are substantially over budget, review the power summary distribution by block type and by the power rails. This provides an idea of the highest power consuming blocks.
- Review the Hierarchy section. The breakdown by hierarchy provides a good idea of the highest power consuming module. You can drill down into a specific module to determine the functionality of the block. You can also cross-probe in the GUI to determine how specific sections of the module have been coded, and whether there are power efficient ways to recode it.

Note: If the design has a timing margin, conduct multiple runs to evaluate if any of the runs have a better total power. For example, a design that has 2 ps of margin can perform similarly to a design with 15 ps, but the 2 ps design might have lower power.

Running Power Optimization

Power optimization works on the entire design or on portions of the design (when `set_power_opt` is used) to minimize power consumption.

Power optimization can be run either pre-place or post-place in the design flow, but not both. The pre-place power optimization step focuses on maximizing power saving. This can result (in rare cases) in timing degradation. If preserving timing is the primary goal, Xilinx recommends the post-place power optimization step. This step performs only those power optimizations that preserve timing.

In cases where portions of the design should be preserved due to legacy (IP) or timing considerations, use the `set_power_opt` command to exclude those portions (such as specific hierarchies, clock domains, or cell types) and rerun power optimization.

Related Information

[Coding Styles to Improve Power](#)

Using the Power Optimization Report

To determine the impact of power optimizations, run the following command in the Tcl Console to generate a power optimization report:

```
report_power_opt -file myopt.rep
```

Using the Timing Report to Determine the Impact of Power Optimization

Power optimization works to minimize the impact on timing while maximizing power savings. However, in certain cases, if timing degrades after power optimization, you can employ a few techniques to offset this impact.

Where possible, identify and apply power optimizations only on non-timing critical clock domains or modules using the `set_power_opt` XDC command. If the most critical clock domain happens to cover a large portion of the design or consumes the most power, review critical paths to see if any cells in the critical path have the `IS_CLOCK_GATED` property with value `TRUE`, indicating that the paths are the result of a power optimization. To improve timing at the expense of increased power in a subsequent implementation, use the `set_power_opt` XDC constraint to disable power optimization on the power-optimized cells in the critical path. Then rerun implementation with the `set_power_opt` XDC constraints or Tcl commands.

The following Tcl example disables power optimization on cells in the top 100 failing paths:

```
set pwr_critical_cells [get_cells -of [get_timing_paths -slack_lesser_than 0 -max_paths 100] -filter {IS_CLOCK_GATED}]
set_power_opt -exclude_cells $pwr_critical_cells
```

Power Timing Slack

When closing a design for timing, it is more efficient and effective to simultaneously close the design from a power perspective. This approach allows for the best run selection that satisfies both criteria. To close both timing and power, add the report_power constraint to the script you are running. For more information and an example script, see [Xilinx Answer Record 76056](#).

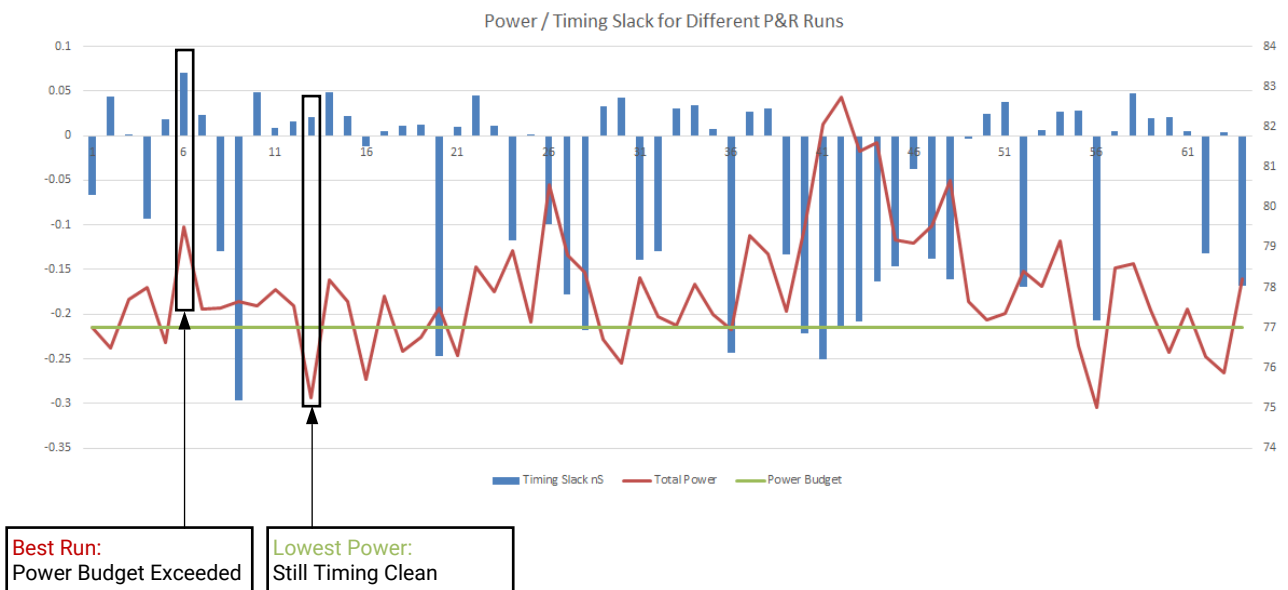
The following figure shows an example of this approach. For all 64 timing closure runs, report power was also run, and all runs are plotted together. From the graph, 36 runs were timing clean, and from a power perspective, the total power budget is 77W. The 64 runs were in the range of 75W to 83W, an 8W or ~10% range.

Looking at the best run from a timing perspective, run #6 had a power estimate of 79.5W, which exceeds the total power budget. However, from the timing clean runs, run #13 yielded the lowest power at 75W and was still timing clean. Understanding the design from both a timing and power perspective allows you to select the best run for both, without impacting the timing result. In this example, this approach enabled a 4W power saving.



POWER TIP: You can also improve design power by removing the DONT_TOUCH constraint to allow upfront logic trimming, including clocking primitives.

Figure 168: Power and Timing Slack for Different Place and Route Runs



X25400-060421

Configuration and Debug

After successfully completing the design implementation, the next step is to load the design into the device and run it on hardware. Configuration is the process of loading application-specific data into the internal memory of the device. Debug is required if the design does not meet expectations on the hardware.

See the following resources for details on configuration and debug software flows and commands:

- *Vivado Design Suite User Guide: Programming and Debugging* ([UG908](#))
- *Vivado Design Suite Tcl Command Reference Guide* ([UG835](#))
- *7 Series FPGAs Configuration User Guide* ([UG470](#))
- *UltraScale Architecture Configuration User Guide* ([UG570](#))
- [Vivado Design Suite QuickTake Video: How To Use the "write_bitstream" Command in Vivado](#)

Configuration

You must first successfully synthesize and implement your design to create a bitstream image. Once the bitstream has been generated and all DRCs are analyzed and corrected, you can load bitstream onto the device using one of the following methods:

- **Direct Programming :**

The bitstream is loaded directly to the device using a cable, processor, or custom solution.

- **Indirect Programming:** The bitstream is loaded into an external flash memory. The flash memory then loads the bitstream into the device.

You can use the Vivado tools to accomplish the following:

- Create the bitstream (`.bit` or `.rbit`).
- Select **Tools** → **Edit Device** to review the configuration settings for bitstream generation.
- Format the bitstream into flash programming files (`.mcs`).
- Program the device using either of the following methods:
 - Directly program the device.
 - Indirectly program the attached configuration flash device.

Flash devices are non-volatile devices and must be erased before programming. Unless a full chip erase is specified, only the address range covered by the assigned MCS is erased.



IMPORTANT! The Vivado Design Suite Device Programmer can use JTAG to read the Status register data on Xilinx devices. In case of a configuration failure, the Status register captures the specific error conditions that can help identify the cause of a failure. In addition, the Status register allows you to verify the Mode pin settings M[2:0] and the bus width detect. For details on the Status register, see the Configuration User Guide for your device.



TIP: If configuration is not successful, you can use a JTAG readback/verify operation on the device to determine whether the intended configuration data was loaded correctly into the device.

Debugging

In-system debugging allows you to debug your design in real time on your target device. This step is needed if you encounter situations that are extremely difficult to replicate in a simulator.

For debug, you provide your design with special debugging IP that allows you to observe and control the design. After debugging, you can remove the instrumentation or special IP to increase performance and logic reduction.

Debugging a design is a multistep, iterative process. Like most complex problems, it is best to break the design debugging process down into smaller parts by focusing on getting smaller sections of the design working one at a time rather than trying to get the whole design to work at once.

Though the actual debugging step comes after you have successfully implemented your design, Xilinx recommends planning how and where to debug early in the design cycle. You can run all necessary commands to perform programming of the devices and in-system debugging of the design from the Program and Debug section of the Flow Navigator in the Vivado IDE.

Following are the debug steps:

1. **Probing:** Identify the signals in your design that you want to probe and how you want to probe them.
2. **Implementing:** Implement the design that includes the additional debug IP attached to the probed nets.
3. **Analyzing:** Interact with the debug IP contained in the design to debug and verify functional issues.
4. **Fixing phase:** Fix any bugs and repeat as necessary.

For more information, see the *Vivado Design Suite User Guide: Programming and Debugging* (UG908).

Debugging the PL

Debugging the programmable logic (PL) can be necessary if you encounter situations that are difficult to replicate in PL logic simulation. This section covers the debugging tools that allow visibility into the PL domain.

Using ILA Cores

The Integrated Logic Analyzer (ILA) core allows you to perform in-system debugging of post-implementation designs on a device. Use this core when you need to monitor signals in the design. You can also use this feature to trigger on hardware events and capture data at system speeds.

Probing the Design

The Vivado tools provide several methods to add debug probes in your design. The table below explains the various methods, including the pros and cons of each method.

Table 18: Debugging Flows

Debugging Flow Name	Flow Steps	Pros/Cons
HDL instantiation probing flow	Explicitly attach signals in the HDL source or IP-Integrator canvas to an ILA debug core instance.	<ul style="list-style-type: none"> You have to add/remove debug nets and IP from your design manually, which means that you will have to modify your HDL source. This method provides the option to probe at the HDL design level. Allows for probing certain protocols such as AXI or AXI4-Stream at the interface level It is easy to make mistakes when generating, instantiating, and connecting debug cores.
Netlist insertion probing flow	Use one of the following two methods to identify the signal for debug: <ul style="list-style-type: none"> Use the MARK_DEBUG attribute to mark signals for debug in the source RTL code. Use the MARK_DEBUG right-click menu option to select nets for debugging in the synthesized design netlist. Once the signal is marked for debug, use the Set up Debug wizard to guide you through the Netlist Insertion probing flow.	<ul style="list-style-type: none"> This method is the most flexible with good predictability. This method allows probing at different design levels (HDL, synthesized design, system design). This method does not require HDL source modification.
Tcl-based netlist insertion probing flow	Use the <code>set_property</code> Tcl command to set the MARK_DEBUG property on debug nets then use netlist insertion probing Tcl commands to create debug cores and connect them to debug nets.	<ul style="list-style-type: none"> This method provides fully automatic netlist insertion. You can turn debugging on or off by modifying the Tcl commands. This method does not require HDL source modification.

Related Information

[Modifying the Implemented Netlist to Replace Existing Debug Probes](#)

Choosing Debug Nets

Xilinx makes the following recommendations for choosing debug nets:

- Probe nets at the boundaries (inputs or outputs) of a specific hierarchy. This method helps isolate problem areas quickly. Subsequently, you can probe further in the hierarchy if needed.
- Do not probe nets in between combinatorial logic paths. If you add MARK_DEBUG on nets in the middle of a combinatorial logic path, none of the optimizations applicable at the implementation stage of the flow are applied, resulting in sub-par timing QoR results.
- Probe nets that are synchronous to get cycle accurate data capture.

Retaining Names of Debug Probe Nets Using MARK_DEBUG

You can mark a signal for debug either at the RTL stage or post-synthesis. The presence of the MARK_DEBUG attribute on the nets ensures that the nets are not replicated, retimed, removed, or otherwise optimized. You can apply the MARK_DEBUG attribute on top level ports, nets, hierarchical module ports and nets internal to hierarchical modules. This method is most likely to preserve HDL signal names post synthesis. Nets marked for debugging are shown in the Unassigned Debug Nets folder in the Debug window post synthesis.

Add the `mark_debug` attribute to HDL files as follows:

VHDL:

```
attribute mark_debug : string;  
attribute keep : string;  
attribute mark_debug of sine : signal is "true";
```

Verilog:

```
(* mark_debug = "true" *) wire sine;
```

You can also add nets for debugging in the post-synthesis netlist. These methods do not require HDL source modification. However, there may be situations where synthesis might not have preserved the original RTL signals due to netlist optimization involving absorption or merging of design structures. Post-synthesis, you can add nets for debugging in any of the following ways:

- Select a net in any of the design views (such as the Netlist or Schematic window), then right-click and select **Mark Debug**.
- Select a net in any of the design views, then drag and drop the net into the Unassigned Debug Nets folder.
- Use the net selector in the Set Up Debug wizard.

- Set the MARK_DEBUG property using the Properties window or the Tcl Console.

```
set_property mark_debug true [get_nets -hier [list {sine[*]}]]
```

This applies the `mark_debug` property on the current, open netlist. This method is flexible, because you can turn MARK_DEBUG on and off through the Tcl command.

ILA Core and Timing Considerations

The configuration of the ILA core has an impact in meeting the overall design timing goals. Follow the recommendations below to minimize the impact on timing:

- Choose probe width judiciously. The bigger the probe width the greater the impact on both resource utilization and timing.
- Choose ILA core data depth judiciously. The bigger the data depth the greater the impact on both block RAM resource utilization and timing.
- Ensure that the clocks chosen for the ILA cores are free-running clocks. Failure to do so could result in an inability to communicate with the debug core when the design is loaded onto the device.
- Ensure that the clock going to the `dbg_hub` is a free running clock. Failure to do so could result in an inability to communicate with the debug core when the design is loaded onto the device. You can use the `connect_debug_port` Tcl command to connect the `clk` pin of the debug hub to a free-running clock.
- Close timing on the design prior to adding the debug cores. Xilinx does not recommend using the debug cores to debug timing related issues.
- If you still notice that timing has degraded due to adding the ILA debug core and the critical path is in the `dbg_hub`, perform the following steps:
 1. Open the synthesized design.
 2. Find the `dbg_hub` cell in the netlist.
 3. Go to the Properties window of the `dbg_hub`.
 4. Find property `C_CLK_INPUT_FREQ_HZ`.
 5. Set it to frequency (in Hz) of the clock that is connected to the `dbg_hub`.
 6. Find property `C_ENABLE_CLK_DIVIDER` and enable it.
 7. Re-implement design.
- Make sure the clock input to the ILA core is synchronous to the signals being probed. Failure to do so results in timing issues and communication failures with the debug core when the design is programmed into the device.
- Make sure that the design meets timing before running it on hardware. Failure to do so results in unreliable probed waveforms.

The following table shows the impact of using specific ILA features on design timing and resources.

Note: This table is based on a design with one ILA and does not represent all designs.

Table 19: Impact of ILA Features on Design Timing and Resources

ILA Feature	When to Use	Timing	Area
Capture Control/ Storage Qualification	To capture relevant data To make efficient use of data capture storage (block RAM)	Medium to High Impact	<ul style="list-style-type: none"> No additional block RAMs Slight increase in LUT/FF count
Advanced Trigger	When BASIC trigger conditions are insufficient To use complex triggering to focus in on problem area	High Impact	<ul style="list-style-type: none"> No additional block RAMs Moderate increase in LUT/FF count
Number of Comparators per Probe Port Note: Maximum is 4.	To use probe in multiple conditionals: <ul style="list-style-type: none"> 1-2 for Basic 1-4 for Advanced +1 for Capture Control 	Medium to High Impact	<ul style="list-style-type: none"> No additional block RAMs Slight to moderate increase in LUT/FF count
Data Depth	To capture more data samples	High Impact	<ul style="list-style-type: none"> Additional block RAMs per ILA core Slight increase in LUT/FF count
ILA Probe Port Width	To debug a large bus versus a scalar	Medium Impact	<ul style="list-style-type: none"> Additional block RAMs per ILA core Slight increase in LUT/FF count
Number of Probes Ports	To probe many nets	Low Impact	<ul style="list-style-type: none"> Additional block RAMs per ILA core Slight increase in LUT/FF count



TIP: In the early stages of the design, there are usually many spare resources in the device that can be used for debugging.

ILA Core Designs with High-Speed Clocks

For designs with high-speed clocks, consider the following:

- Limit the number and width of signals being debugged.
- Pipeline the input probes to the ILA (C_INPUT_PIPE_STAGES), which enables extra levels of pipe stages.

Note: For designs with limited MMCM/BUFG availability, consider clocking the debug hub with the lowest clock frequency in the design instead of using the clock divider inside the debug hub.

Using VIO Cores

The Virtual Input/Output (VIO) core allows you to monitor and drive internal device signals in real time. Use this core when it is necessary to drive or monitor low speed signals, such as resets or status signals. The VIO debug core must be instantiated in the design and can be used in both Vivado IP integrator block design and RTL. The VIO core is available in the IP catalog for RTL-based designs and in IP integrator.

For information on customizing the VIO core, see the *Virtual Input/Output LogiCORE IP Product Guide* (PG159). For information on taking measurements with a VIO core, see this [link](#) in the *Vivado Design Suite User Guide: Programming and Debugging* (UG908).

VIO Core Considerations

When using VIO cores, consider the following:

- Signals connected to VIO input probes must be synchronous to the clock connected to the VIO `clk` port on the VIO core. Connecting signals that are not synchronous to the `clk` port results in a clock domain crossing at the VIO input probe port.
- Signals driven from VIO output probes are asserted and deasserted synchronous to the clock connected to the VIO `clk` port on the VIO core.
- The VIO core has a relatively low refresh rate because it is intended to replace low speed board I/O, such as push-buttons or light-emitting diodes (LEDs). To capture high-speed signals, consider using the ILA core.

Debugging Designs in Vivado IP Integrator

The Vivado IP integrator provides different ways to set up your design for debugging. You can use one of the following flows to add debug cores to your IP integrator design. The flow you choose depends on your preference and the types of nets and signals that you want to debug.

- Debug interfaces, nets, or both in the block design using the System ILA core

Use this flow to:

- Perform hardware-software co-verification using the cross-trigger feature of a MicroBlaze™ device, Zynq®-7000 SoC, or Zynq UltraScale+ MPSoC.
- Verify the interface-level connectivity.

- Netlist insertion flow

Use this flow to analyze I/O ports and internal nets in the post-synthesized design.

Note: You can also use a combination of both flows to debug your design.

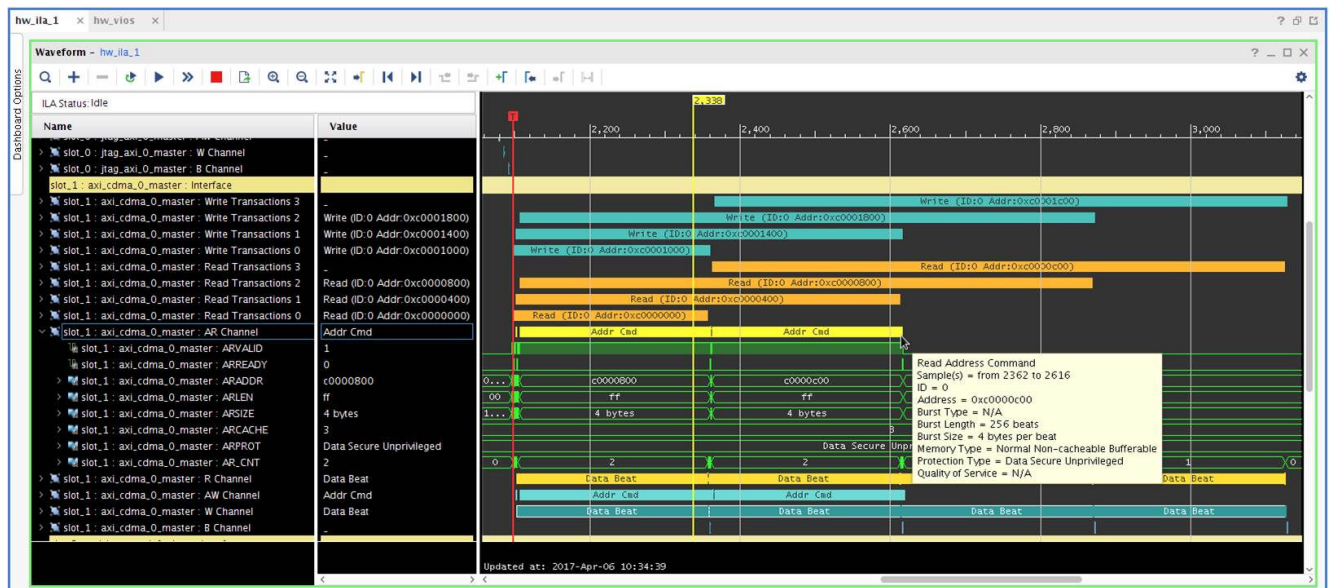
For more information on using System ILA in your IP integrator design, see the *Vivado Design Suite User Guide: Designing IP Subsystems Using IP Integrator* (UG994).

Debugging AXI Interfaces in Vivado Hardware Manager

The ILA allows you to perform in-system debugging of post-implemented designs on a Xilinx device. Use this feature when there is a need to monitor interfaces and signals in the design.

If you changed the ILA mode to **Interface**, you can debug and monitor AXI transactions and read and write events in the Waveform window shown in the following figure. The Waveform window displays the interface slots, transactions, events, and signal groups that correspond to the interfaces probed by the interface slots on the ILA.

Figure 169: Waveform Window



For more information on System ILA and debugging AXI interfaces in the Vivado Hardware Manager, see this [link](#) and this [link](#) in the *Vivado Design Suite User Guide: Programming and Debugging (UG908)*.

Using In-System IBERT

The In-System IBERT core provides RX margin analysis through eye scan plots on the RX data of transceivers in UltraScale and UltraScale+ devices. The core enables configuration and tuning of the GTH/GTY transceivers and is accessible through logic that communicates with the dynamic reconfiguration port (DRP) of the transceivers. You can use the core to change attribute settings as well as registers that control the values on the `rxrate`, `rxlpmen`, `txdiffctrl`, `txpostcursor`, and `txprecursor` ports.

The Vivado Serial I/O Analyzer in the Hardware Manager communicates with the core through JTAG when the design is programmed onto the device. There is only one instance of In-System IBERT required per design. In-System IBERT can work with all GTs used in the design. However, you must generate separate In-System IBERT cores according to the different GT types (for example, GTH, GTY).

Creating an In-System IBERT design with an internal system clock can prevent a scan from being performed. When creating an eye scan, the status changes from **In Progress** to **Incomplete**. Eye scan is incomplete when the internal system clock (MGTREFCLK) is connected to the `clk/drpcclk_i` input port of In-System IBERT IP.

Note: If needed, consider using an external clock, which does not exhibit this behavior. Alternatively, click any available link in the Vivado Serial I/O Analyzer. Go to the Properties window, and find the MB_RESET reg under the LOGIC field. Set it to 1 and then toggle back to 0. Rerun the eye scan or sweep.

For more information on this core, see the *In-System IBERT LogiCORE IP Product Guide* ([PG246](#)).

Running Debug-Related DRCs

The Vivado Design Suite provides debug-related DRCs, which are selected as part of the default rule deck when `report_drc` is run. The DRCs check for the following:

- Block RAM resources for the device are exceeded because of the current requirements of the debug core.
- Non-clock net is connected to the clock port on the debug core.
- Port on the debug core is unconnected.

Modifying the Implemented Netlist to Replace Existing Debug Probes

It is possible to replace debug nets connected to an ILA core in a placed and routed design checkpoint. You can do this by using the Engineering Change Order (ECO) flow. This is an advanced design flow used for designs that are nearing completion, where you need to swap nets connected to an existing ILA probe port. For information on using the ECO flow to modify nets on existing ILA cores, see this [link](#) in the *Vivado Design Suite User Guide: Implementation* ([UG904](#)).

Inserting, Deleting, or Editing ILA Cores on an Implemented Netlist

If you want to add, delete, or modify ILA cores (for example, resizing probe width, changing the data depth, etc.), Xilinx recommends that you use the Incremental Compile flow. The Incremental Compile flow for debug cores operates on a synthesized design or checkpoint (DCP) and uses a reference implemented checkpoint, ideally from a previous implementation run. This approach might save you time versus a complete re-implementation of the design.

For information on using the Incremental Compile flow to insert, delete, or edit ILA cores, see this [link](#) in the *Vivado Design Suite User Guide: Programming and Debugging* ([UG908](#)).

Connecting a Net to a Free External Pin Using Post-Route ECO

In some cases you might want to bring a net out to a free device pin for debug using external test equipment. This approach can be useful if you are debugging issues that require minimal changes to the design QoR or require measurements not possible to obtain through other means. You can do this by using the Engineering Change Order (ECO) flow as long as the device has an unused I/O that can be used for this purpose. For more information on using the ECO flow to modify a routed design, see this [link](#) in the *Vivado Design Suite User Guide: Implementation (UG904)*.

Using Remote Debugging

Xilinx provides the following ways to debug or upgrade your design remotely:

- Use the Xilinx Hardware Server product to connect to a remote computer in the lab.

Additional Resources and Legal Notices

Xilinx Resources

For support resources such as Answers, Documentation, Downloads, and Forums, see [Xilinx Support](#).

Solution Centers

See the [Xilinx Solution Centers](#) for support on devices, tools, and intellectual property at all stages of the design cycle. Topics include design assistance, advisories, and troubleshooting tips.

Documentation Navigator and Design Hubs

Xilinx® Documentation Navigator (DocNav) provides access to Xilinx documents, videos, and support resources, which you can filter and search to find information. To open DocNav:

- From the Vivado® IDE, select **Help** → **Documentation and Tutorials**.
- On Windows, select **Start** → **All Programs** → **Xilinx Design Tools** → **DocNav**.
- At the Linux command prompt, enter `docnav`.

Xilinx Design Hubs provide links to documentation organized by design tasks and other topics, which you can use to learn key concepts and address frequently asked questions. To access the Design Hubs:

- In DocNav, click the **Design Hubs View** tab.
- On the Xilinx website, see the [Design Hubs](#) page.

Note: For more information on DocNav, see the [Documentation Navigator](#) page on the Xilinx website.

References

These documents provide supplemental material useful with this guide.

1. [Vivado® Design Suite Documentation](#)
2. *UltraFast Design Methodology Quick Reference Guide* ([UG1231](#))
3. *UltraFast Design Methodology Timing Closure Quick Reference Guide* ([UG1292](#))
4. *UltraFast Design Methodology Checklist* ([XTP301](#))

Vivado Design Suite User and Reference Guides

1. *Xilinx Power Estimator User Guide* ([UG440](#))
2. *UltraFast Embedded Design Methodology Guide* ([UG1046](#))
3. *Vivado Design Suite Tcl Command Reference Guide* ([UG835](#))
4. *Vivado Design Suite User Guide: Design Flows Overview* ([UG892](#))
5. *Vivado Design Suite User Guide: Using the Vivado IDE* ([UG893](#))
6. *Vivado Design Suite User Guide: Using Tcl Scripting* ([UG894](#))
7. *Vivado Design Suite User Guide: System-Level Design Entry* ([UG895](#))
8. *Vivado Design Suite User Guide: Designing with IP* ([UG896](#))
9. *Vivado Design Suite User Guide: Embedded Processor Hardware Design* ([UG898](#))
10. *Vivado Design Suite User Guide: I/O and Clock Planning* ([UG899](#))
11. *Vivado Design Suite User Guide: Logic Simulation* ([UG900](#))
12. *Vivado Design Suite User Guide: Synthesis* ([UG901](#))
13. *Vivado Design Suite User Guide: Using Constraints* ([UG903](#))
14. *Vivado Design Suite User Guide: Implementation* ([UG904](#))
15. *Vivado Design Suite User Guide: Hierarchical Design* ([UG905](#))
16. *Vivado Design Suite User Guide: Design Analysis and Closure Techniques* ([UG906](#))
17. *Vivado Design Suite User Guide: Power Analysis and Optimization* ([UG907](#))
18. *Vivado Design Suite User Guide: Programming and Debugging* ([UG908](#))
19. *Vivado Design Suite User Guide: Dynamic Function eXchange* ([UG909](#))
20. *Vivado Design Suite User Guide: Getting Started* ([UG910](#))
21. *Vivado Design Suite Properties Reference Guide* ([UG912](#))
22. *Vivado Design Suite User Guide: Release Notes, Installation, and Licensing* ([UG973](#))

23. *Vivado Design Suite User Guide: Designing IP Subsystems Using IP Integrator* ([UG994](#))
24. *Vivado Design Suite User Guide: Creating and Packaging Custom IP* ([UG1118](#))
25. a. *Vivado Design Suite 7 Series FPGA and Zynq-7000 SoC Libraries Guide* ([UG953](#))
b. *UltraScale Architecture Libraries Guide* ([UG974](#))
26. *UltraScale+ Devices Integrated Block for PCI Express LogiCORE IP Product Guide* ([PG213](#))

Vivado Design Suite Tutorials

1. *Vivado Design Suite Tutorial: High-Level Synthesis* ([UG871](#))
2. *Vivado Design Suite Tutorial: Design Flows Overview* ([UG888](#))
3. *Vivado Design Suite Tutorial: Logic Simulation* ([UG937](#))
4. *Vivado Design Suite Tutorial: Embedded Processor Hardware Design* ([UG940](#))
5. *Vivado Design Suite Tutorial: Dynamic Function eXchange* ([UG947](#))

Other Xilinx Documentation

1. a. *7 Series FPGAs PCB Design Guide* ([UG483](#))
b. *UltraScale Architecture PCB Design User Guide* ([UG583](#))
c. *Zynq-7000 SoC PCB Design Guide* ([UG933](#))
2. a. *7 Series FPGAs SelectIO Resources User Guide* ([UG471](#))
b. *UltraScale Architecture SelectIO Resources User Guide* ([UG571](#))
3. a. *7 Series FPGAs Clocking Resources User Guide* ([UG472](#))
b. *UltraScale Architecture Clocking Resources User Guide* ([UG572](#))
4. a. *UltraScale Architecture GTH Transceivers User Guide* ([UG576](#))
b. *UltraScale Architecture GTY Transceivers User Guide* ([UG578](#))
5. *UltraScale Devices Gen3 Integrated Block for PCI Express LogiCORE IP Product Guide* ([PG156](#))
6. *Virtual Input/Output LogiCORE IP Product Guide* ([PG159](#))
7. *In-System IBERT LogiCORE IP Product Guide* ([PG246](#))
8. *7 Series FPGAs Memory Resources User Guide* ([UG473](#))
9. *7 Series DSP48E1 Slice User Guide* ([UG479](#))
10. *UltraScale Architecture DSP Slice User Guide* ([UG579](#))
11. *7 Series FPGAs and Zynq-7000 SoC XADC Dual 12-Bit 1 MSPS Analog-to-Digital Converter User Guide* ([UG480](#))
12. *Reference System: Kintex-7 MicroBlaze System Simulation Using IP Integrator* ([XAPP1180](#))

13. *Designing Using SelectIO Interface Component Primitives* ([XAPP1324](#))
 14. *Zynq-7000 SoC and 7 series Devices Memory Interface Solutions* ([UG586](#))
 15. *UltraScale Architecture-Based FPGAs Memory IP LogiCORE IP Product Guide* ([PG150](#))
 16. *Simulating FPGA Power Integrity Using S-Parameter Models* ([WP411](#))
 17. *Extending the Thermal Solution by Utilizing Excursion Temperatures* ([WP517](#))
 18. a. *7 Series Schematic Review Recommendations* ([XMP277](#))
 - b. *Kintex UltraScale and Virtex UltraScale FPGAs Schematic Review Checklist* ([XTP344](#))
 - c. *UltraScale+ FPGAs and Zynq Ultrascale+ Devices Schematic Review Checklist* ([XTP427](#))
 19. *UltraScale FPGA BPI Configuration and Flash Programming* ([XAPP1220](#))
 20. *BPI Fast Configuration and iMPACT Flash Programming with 7 Series FPGAs* ([XAPP587](#))
 21. *Using SPI Flash with 7 Series FPGAs* ([XAPP586](#))
 22. *SPI Configuration and Flash Programming in UltraScale FPGAs* ([XAPP1233](#))
 23. *Using Encryption to Secure a 7 Series FPGA Bitstream* ([XAPP1239](#))
 24. *Mechanical and Thermal Design Guidelines for Lidless Flip-Chip Packages* ([XAPP1301](#))
 25. *Vitis HLS User Guide* ([UG1399](#))
 26. *Vitis HLS Methodology in the Vitis HLS User Guide* ([UG1399](#))
-

Training Resources

1. [UltraFast Design Methodology Training Course](#)
2. [Vivado Design Suite QuickTake Video: UltraFast Vivado Design Methodology](#)
3. [Vivado Design Suite QuickTake Video: Vivado Design Flows Overview](#)
4. [Vivado Design Suite QuickTake Video: Targeting Zynq Using Vivado IP Integrator](#)
5. [Vivado Design Suite QuickTake Video: Partial Reconfiguration in Vivado Design Suite](#)
6. [Vivado Design Suite QuickTake Video: Creating Different Types of Projects](#)
7. [Vivado Design Suite QuickTake Video: Managing Sources With Projects](#)
8. [Vivado Design Suite QuickTake Video: Using Vivado Design Suite with Revision Control](#)
9. [Vivado Design Suite QuickTake Video: Managing Vivado IP Version Upgrades](#)
10. [Vivado Design Suite QuickTake Video: I/O Planning Overview](#)
11. [Vivado Design Suite QuickTake Video: Configuring and Managing Reusable IP in Vivado](#)
12. [Vivado Design Suite QuickTake Video: How To Use the "write_bitstream" Command in Vivado](#)

13. [Vivado Design Suite QuickTake Video: Design Analysis and Floorplanning](#)
14. [Vivado Design Suite QuickTake Video: Introducing the UltraFast Design Methodology Checklist](#)
15. [Vivado Design Suite Video Tutorials](#)

Please Read: Important Legal Notices

The information disclosed to you hereunder (the "Materials") is provided solely for the selection and use of Xilinx products. To the maximum extent permitted by applicable law: (1) Materials are made available "AS IS" and with all faults, Xilinx hereby DISCLAIMS ALL WARRANTIES AND CONDITIONS, EXPRESS, IMPLIED, OR STATUTORY, INCLUDING BUT NOT LIMITED TO WARRANTIES OF MERCHANTABILITY, NON-INFRINGEMENT, OR FITNESS FOR ANY PARTICULAR PURPOSE; and (2) Xilinx shall not be liable (whether in contract or tort, including negligence, or under any other theory of liability) for any loss or damage of any kind or nature related to, arising under, or in connection with, the Materials (including your use of the Materials), including for any direct, indirect, special, incidental, or consequential loss or damage (including loss of data, profits, goodwill, or any type of loss or damage suffered as a result of any action brought by a third party) even if such damage or loss was reasonably foreseeable or Xilinx had been advised of the possibility of the same. Xilinx assumes no obligation to correct any errors contained in the Materials or to notify you of updates to the Materials or to product specifications. You may not reproduce, modify, distribute, or publicly display the Materials without prior written consent. Certain products are subject to the terms and conditions of Xilinx's limited warranty, please refer to Xilinx's Terms of Sale which can be viewed at <https://www.xilinx.com/legal.htm#tos>; IP cores may be subject to warranty and support terms contained in a license issued to you by Xilinx. Xilinx products are not designed or intended to be fail-safe or for use in any application requiring fail-safe performance; you assume sole risk and liability for use of Xilinx products in such critical applications, please refer to Xilinx's Terms of Sale which can be viewed at <https://www.xilinx.com/legal.htm#tos>.

AUTOMOTIVE APPLICATIONS DISCLAIMER

AUTOMOTIVE PRODUCTS (IDENTIFIED AS "XA" IN THE PART NUMBER) ARE NOT WARRANTED FOR USE IN THE DEPLOYMENT OF AIRBAGS OR FOR USE IN APPLICATIONS THAT AFFECT CONTROL OF A VEHICLE ("SAFETY APPLICATION") UNLESS THERE IS A SAFETY CONCEPT OR REDUNDANCY FEATURE CONSISTENT WITH THE ISO 26262 AUTOMOTIVE SAFETY STANDARD ("SAFETY DESIGN"). CUSTOMER SHALL, PRIOR TO USING OR DISTRIBUTING ANY SYSTEMS THAT INCORPORATE PRODUCTS, THOROUGHLY TEST SUCH SYSTEMS FOR SAFETY PURPOSES. USE OF PRODUCTS IN A SAFETY APPLICATION WITHOUT A SAFETY DESIGN IS FULLY AT THE RISK OF CUSTOMER, SUBJECT ONLY TO APPLICABLE LAWS AND REGULATIONS GOVERNING LIMITATIONS ON PRODUCT LIABILITY.

Copyright

© Copyright 2013–2021 Xilinx, Inc. Xilinx, the Xilinx logo, Alveo, Artix, Kintex, Kria, Spartan, Versal, Vitis, Virtex, Vivado, Zynq, and other designated brands included herein are trademarks of Xilinx in the United States and other countries. AMBA, AMBA Designer, Arm, ARM1176JZ-S, CoreSight, Cortex, PrimeCell, Mali, and MPCore are trademarks of Arm Limited in the EU and other countries. OpenCL and the OpenCL logo are trademarks of Apple Inc. used by permission by Khronos. PCI, PCIe, and PCI Express are trademarks of PCI-SIG and used under license. MATLAB and Simulink are registered trademarks of The MathWorks, Inc. All other trademarks are the property of their respective owners.