

Vivado Design Suite Tutorial

Design Analysis and Closure Techniques

UG938 (v2013.2) June 19, 2013





Notice of Disclaimer

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 the Limited Warranties which can be viewed at <http://www.xilinx.com/warranty.htm>; 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 Critical Applications: <http://www.xilinx.com/warranty.htm#critapps>.

©Copyright 2012-2013 Xilinx, Inc. Xilinx, the Xilinx logo, Artix, Vivado, ISE, Kintex, Spartan, Virtex, Vivado, Zynq, and other designated brands included herein are trademarks of Xilinx in the United States and other countries. All other trademarks are the property of their respective owners...

Revision History

| Date | Version | Revision |
|------------|---------|-------------------------|
| 06/19/2013 | 2013.2 | Validated with Release |
| 03/20/2013 | 2013.1 | Validated with Release. |
| | | |

Table of Contents

| | |
|--|----|
| Revision History..... | 2 |
| Table of Contents | 3 |
| Vivado Design Analysis and Closure Techniques Tutorial | 5 |
| Overview | 5 |
| Tutorial Objectives..... | 5 |
| Tutorial Design Description | 6 |
| Software Requirements..... | 6 |
| Hardware Requirements | 6 |
| Preparing the Tutorial Design Files..... | 6 |
| Lab 1: Design Analysis | 7 |
| Step 1: Opening the Example Project..... | 7 |
| Step 2: Viewing the Device Resources and Clock Regions..... | 9 |
| Examining Sites and BELs..... | 13 |
| Examining Clock Regions..... | 14 |
| Step 3: Exploring the Logical Hierarchy | 17 |
| Cross probing RTL Source Files | 18 |
| Step 4: Displaying Design Resources Statistics..... | 19 |
| Viewing the Resource Estimates of the Entire Design | 19 |
| Examining Resources in More Detail..... | 20 |
| Step 5: Performing Pre-Implementation Timing Analysis..... | 21 |
| Analyzing Timing End Points with the Slack Histogram | 21 |
| Analyzing the Pre-Placed Timing | 24 |
| Step 6: Implementing the Design..... | 27 |
| Examining the Constraints File..... | 27 |
| Defining Implementation Runs..... | 28 |
| Launching Runs | 30 |
| Conclusion..... | 34 |

| | |
|---|----|
| Lab 2: Timing Closure Techniques..... | 35 |
| Step 1: Analyzing Post-Implementation Timing | 35 |
| Viewing Timing Paths | 35 |
| Viewing the Routed Path | 38 |
| Viewing Path Properties | 39 |
| Step 2: Exploring the Hierarchy of the Timing Paths | 40 |
| Step 3: Highlighting Module Placement and Connectivity..... | 43 |
| Visualizing the Connectivity | 44 |
| Step 4: Floorplanning the Design..... | 46 |
| Automatically Creating Pblocks..... | 46 |
| Placing Pblocks | 47 |
| Viewing Pblock Connections | 49 |
| Finding a Global Clock | 50 |
| Moving Pblocks | 51 |
| Clearing Placement LOCs..... | 53 |
| Resizing Pblocks | 55 |
| Step 5: Running Implementation with the Floorplan..... | 60 |
| Conclusion..... | 62 |

Vivado Design Analysis and Closure Techniques Tutorial

Overview

This tutorial introduces some of the capabilities and benefits of using the Xilinx[®] Vivado[®] Design Suite for designing high-end FPGAs. The document contains a series of step-by-step exercises that highlight methods to achieve and maintain better performing designs in less time. The steps detail the following:

- Pre-implementation design and analysis capabilities
- Implementation exploration features
- Floorplanning implementation results

Note: This tutorial covers a subset of the features of the Vivado Design Suite. Additional features are covered in detail in other tutorials.

Tutorial Objectives

This tutorial takes you through the various analysis, floorplanning, and implementation features of the Vivado Design Suite. When using this tutorial, you should focus on the demonstrated techniques and how they relate to your own designs. The exercises cover the following topics:

- Analyze device utilization statistics to target alternate devices and choose the optimal device.
- Run Design Rule Checks (DRC) to quickly resolve constraint conflicts that would otherwise cause implementation errors.
- Use the Netlist, Logic Hierarchy, and Schematic windows to explore logic.
- Perform a quick estimation of timing performance to assess design feasibility and identify potential problem areas.
- View, modify, or create constraints in the design.
- Analyze the design hierarchical connectivity and data flow, as well as identify critical logic connectivity and clock domains.
- Floorplan timing-critical logic to improve timing and data flow.
- View Routing Resources in the detailed Device window

Tutorial Design Description

The sample design used throughout this tutorial consists of a small design called `project_cpu_hdl`, which targets an xc7k70t device. The small sample design used in this tutorial includes:

- A RISC processor CPU core
- A pseudo FFT
- Four gigabit transceivers (GTs)
- Two USB interfaces

A small design is used to enable timely completion of the synthesis and implementation runs with minimum hardware requirements.

Software Requirements

This tutorial requires that the 2013.2 Vivado Design Suite software release or later is installed. For installation instructions and information, see the *Vivado Design Suite User Guide: Release Notes, Installation, and Licensing* ([UG973](#)).

Hardware Requirements

The supported Operating Systems include Redhat 5.6 Linux 64 and 32 bit, and Windows 7, 64 and 32 bit.

Xilinx recommends a minimum of 2 GB of RAM when using the Vivado tool.

Preparing the Tutorial Design Files

You can find the files for this tutorial in the Vivado Design Suite examples directory at the following location:

- **<Vivado_install_area>/Vivado/<version>/examples/Vivado_Tutorial**

You can also extract the provided zip file, at any time, to write the tutorial files to your local directory, or to restore the files to their starting condition.

Extract the zip file contents from the software installation into any write-accessible location.

- **<Vivado_install_area>/Vivado/<version>/examples/Vivado_Tutorial.zip**

The extracted Vivado_Tutorial directory is referred to as the *<Extract_Dir>* in this Tutorial.

Note: You will modify the tutorial design data while working through this tutorial. You should use a new copy of the original Vivado_Tutorial directory each time you start this tutorial.

Lab 1: Design Analysis

Step 1: Opening the Example Project

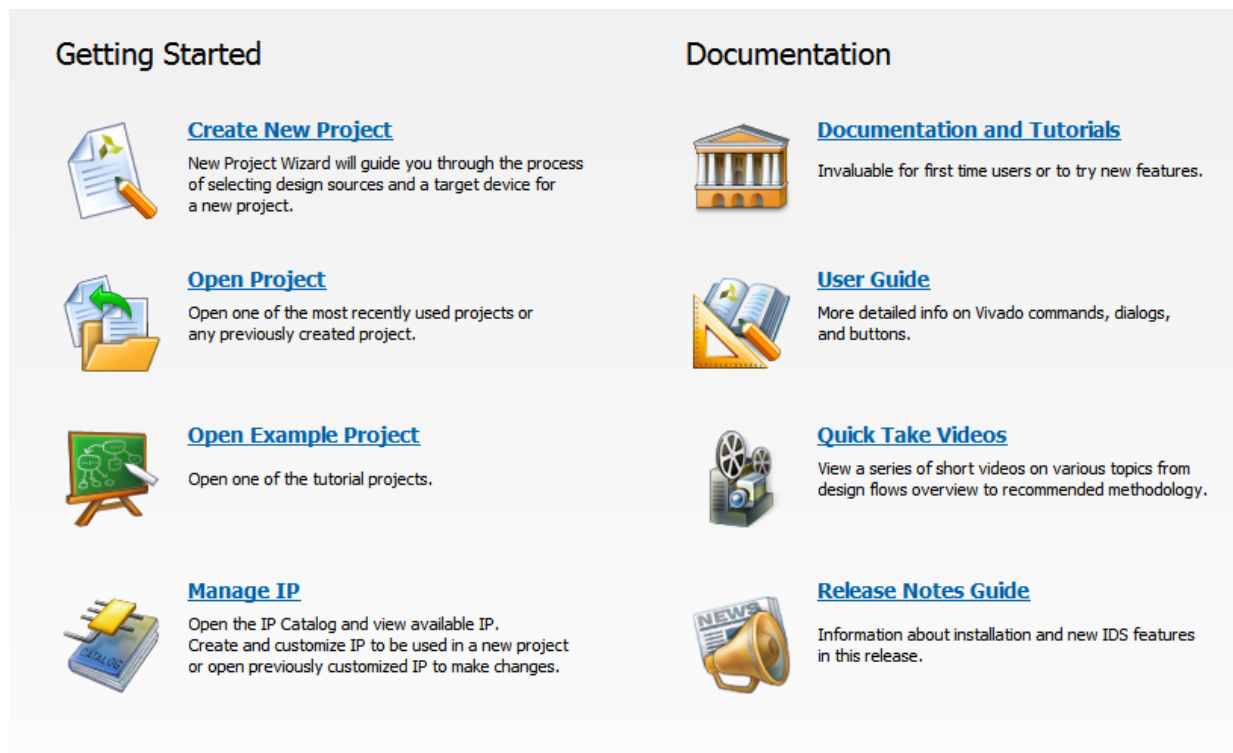


Figure 1: Vivado IDE Getting Started Page

- On Linux,
 1. Change to the directory where the lab materials are stored:
cd <Extract_Dir>/Vivado_Tutorial
 2. Launch the Vivado IDE: **vivado**
- On Windows,
 1. Launch the Vivado Design Suite IDE:
Start > All Programs > Xilinx Design Tools > Vivado 2013.2 > Vivado 2013.2¹
 2. As an alternative, click the **Vivado 2013.2** Desktop icon to start the Vivado IDE.

¹ Your Vivado Design Suite installation may be called something other than **Xilinx Design Tools** on the **Start** menu.

The Vivado IDE Getting Started page opens, as shown in [Figure 1](#), with links to open or create projects and to view documentation.

3. From the Getting Started page, click **Open Example Project** and select the **CPU (HDL)** design.

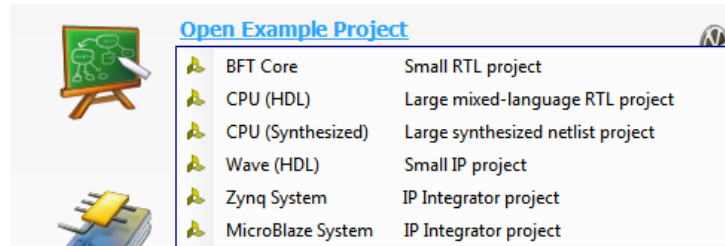


Figure 2: Open Example Design

The design project opens in the Vivado IDE, and a dialog box opens stating that the project is read-only and prompting you to save the project to a new location.

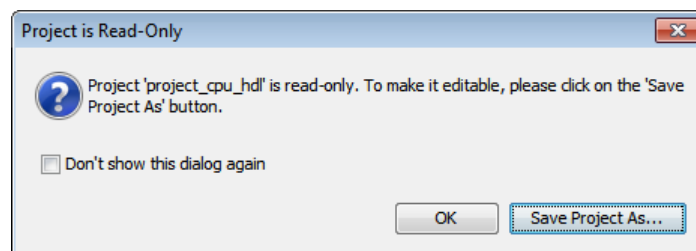


Figure 3: Read-Only Project

4. Select **Save Project As** and specify a project name and location.

The project is saved to the specified location. If the specified directory does not exist, the Vivado Design Suite will create the directory as needed.

Note: This design will also be used in Lab #2 for the second portion of this tutorial. You will need to remember where you have saved the tutorial project in this step.

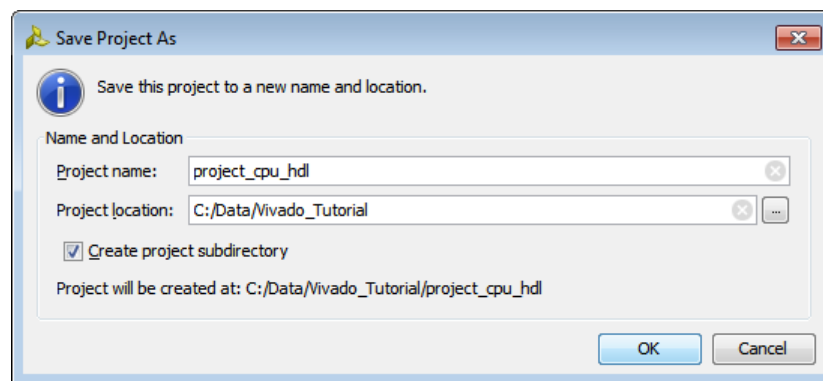


Figure 4: Save Project As...

The Vivado IDE displays project information in the Project Summary window. The Project Summary shows the project settings, and the run details for both the synthesis and implementation runs.

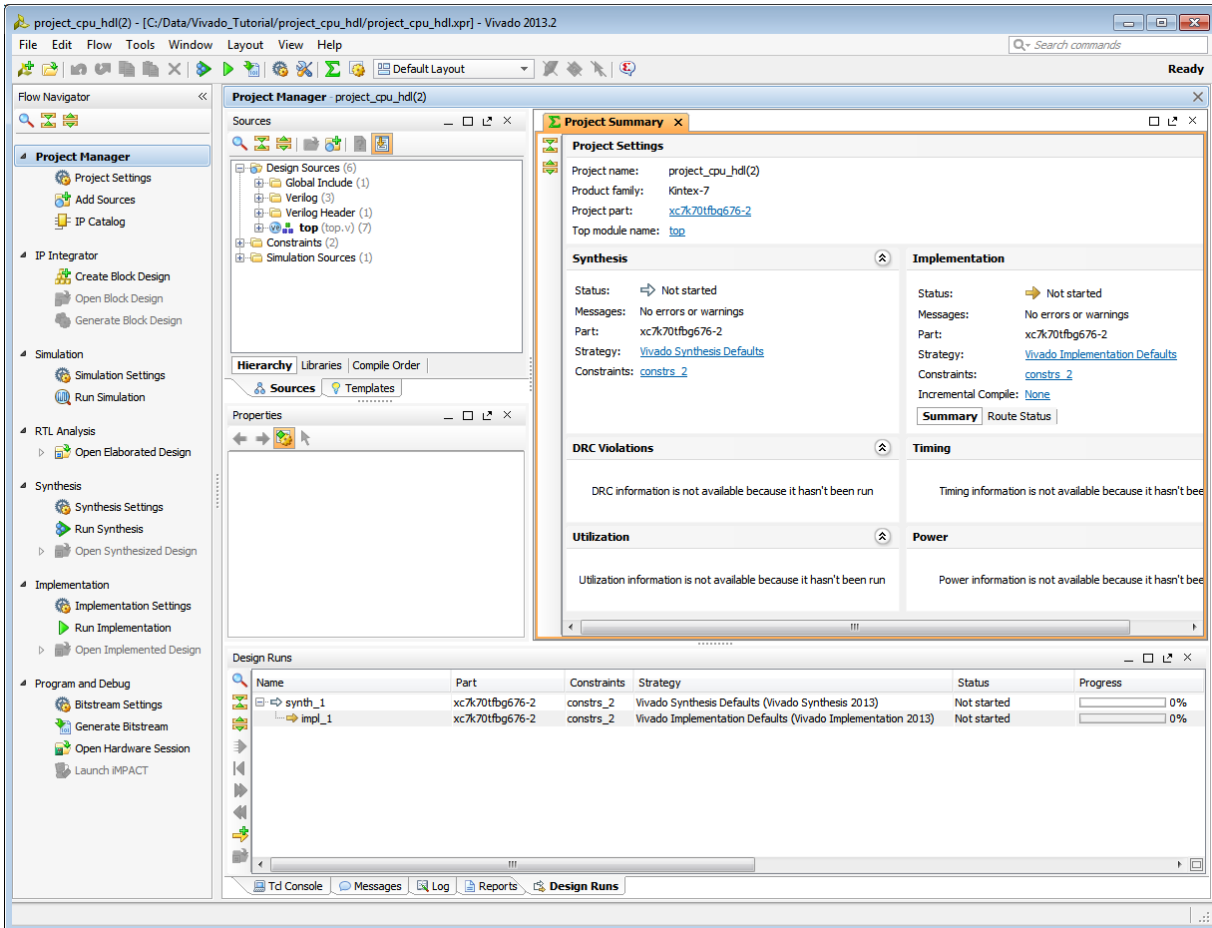


Figure 5: Vivado Environment

Step 2: Viewing the Device Resources and Clock Regions

1. In the Sources window, **expand** the **Constraints** folder to ensure constrs_2 is active, as shown in [Figure 6](#).
2. If needed, select **constrs_2**, right-click and select **Make Active** from the popup menu.

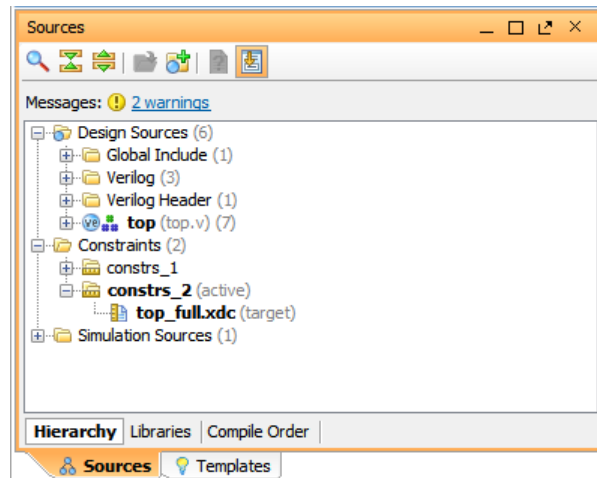


Figure 6: Sources window

3. Click the **Design Runs** window and examine the runs information.

The Design Runs window offers commands to manage, launch, and reset runs for both synthesis and implementation. You can manage multiple runs in the Design Runs window.

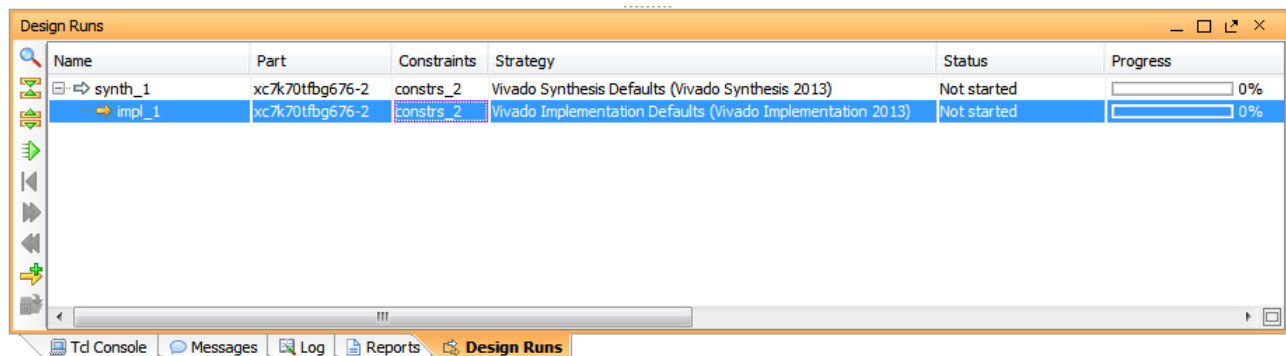


Figure 7: Design Runs window

4. In the Flow Navigator, select **Run Synthesis** to synthesize the design.

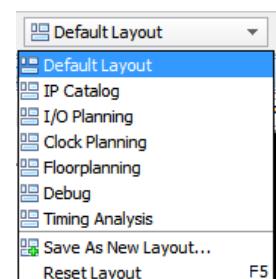
Watch the Log window for run status. It takes several minutes for the run to complete.

5. After synthesis completes, select **Open Synthesized Design** from the Synthesis Completed dialog box and click **OK**.

The Synthesized Design opens with the netlist window as seen in [Figure 8](#).

6. In the Layout Selector, in the main toolbar, change the window layout to the **Default Layout** if that is not the current window layout.

7. Click on the **Device** window to make it active.



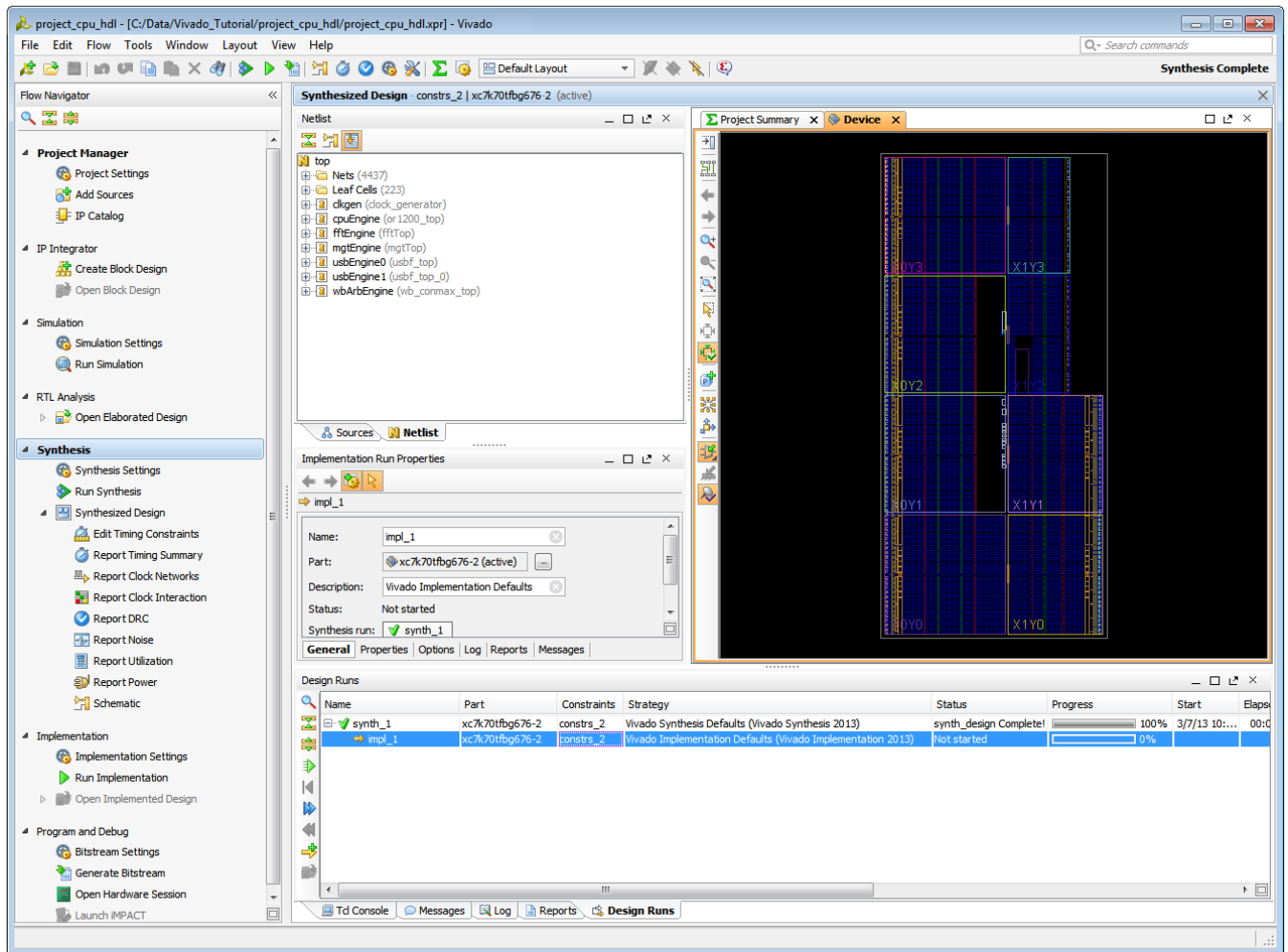


Figure 8: Open the Synthesized Design

- In the Device window, use **Zoom Area** by clicking and holding the left mouse button, while dragging down and to the right.

This will draw a Zoom Area rectangle. Repeat as needed to zoom into the device close enough to see the different device resources.

In the Device window, the Vivado IDE displays the FPGA device resources that can be used to implement the design, as a matrix of tiles. Sites are available within a tile for placement of netlist instances. The available sites include, for example, SLICES, RAMs, MULTs, and DSPs, and vary in shape and color in the Device window to differentiate the object types contained in the site. Slices are fundamental building blocks of Xilinx FPGAs, containing LUTs and registers, though the specific contents of a slice may vary with the device family. For more information on these terms, refer to the [Xilinx Glossary](#).

"LOC" and "BEL" placement constraints are used to assign logic to either a site or a slice. Leaf cells can be assigned specifically to a site, or generally to a slice with a "LOC" placement constraint, or can be assigned to specific logic resources within a slice using both "LOC" and "BEL" constraints.

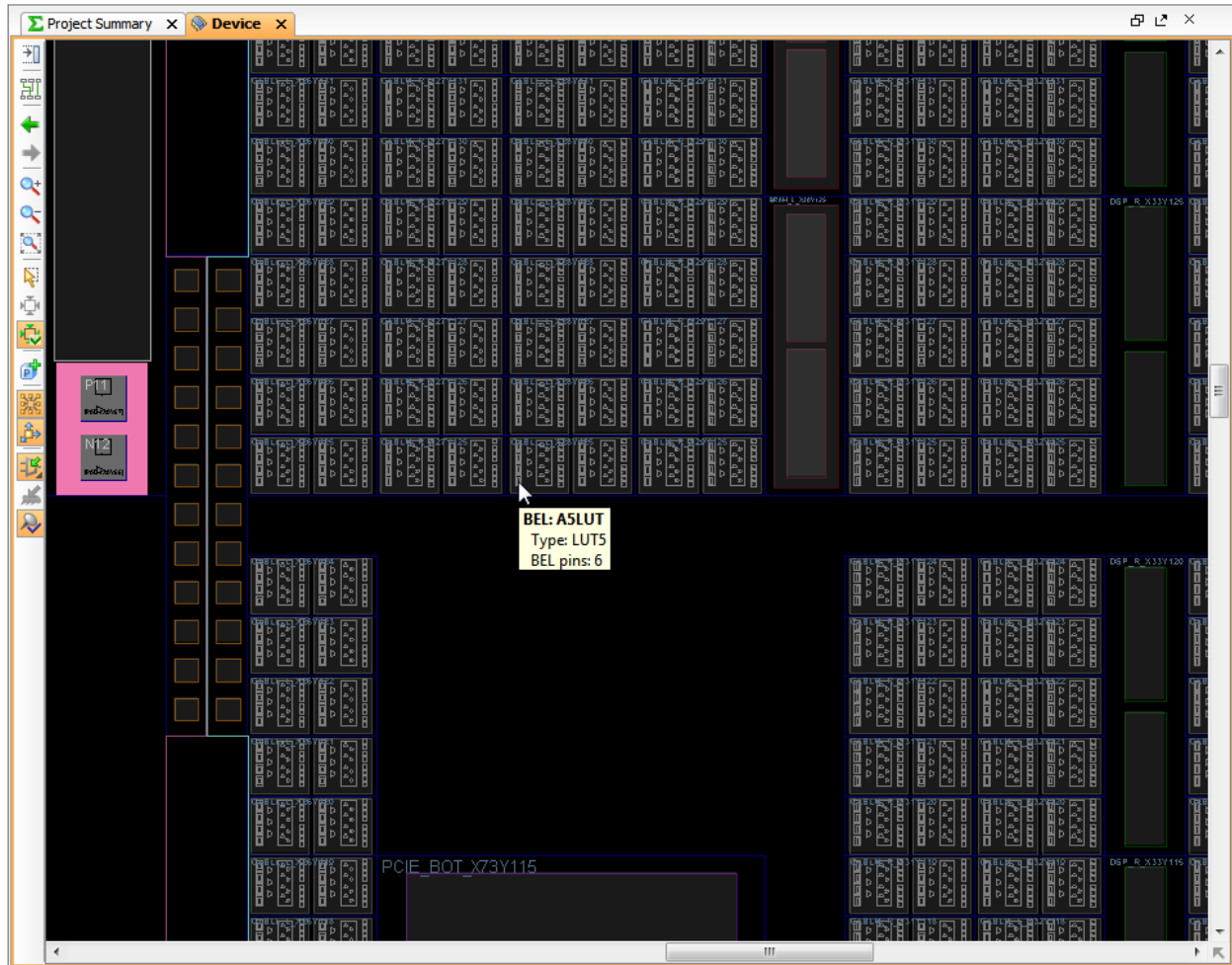


Figure 9: Viewing Sites, Slices and BELs in the Device window

9. Move the cursor slowly over the various device resources to view the tooltips, and take note of the following:
 - Hover the mouse over a site to show a tooltip.
 - I/O port locations and I/O buffer assignments appear inside I/O banks.
 - Tall dark red tiles indicate a RAMB36 site that can also hold two RAMB18s or a FIFO.²
 - Tall green tiles indicate sites for DSP48s.
 - Square blue tiles are a CLB containing SLICES.
 - SLICE coordinates appear in the status bar at the bottom left of the Vivado IDE main window.

BEL: F7AMUX Type: SELMUX2_1 BEL pins: 4 Site: SLICE_X6Y98

Examining Sites and BELs

Below the Netlist window, the Site Properties or BEL Properties window is displayed. These windows appear when a site or bel is selected within the Device window, or some other window.

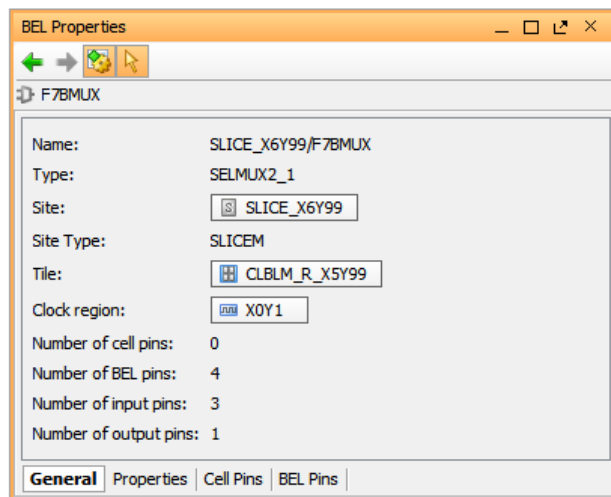


Figure 10 : BEL Properties window

1. Select several different types of resources within the Device window, and examine the Site and BEL Property information.
2. Use the **Zoom Fit** command from the Device window toolbar menu, or click and drag the cursor up and to the left in the Device window, to see the whole device.
3. Click the **Auto Fit Selection** button on the Device window tool bar menu, to zoom and center any objects selected in the next section.

² The colors mentioned here are for the Vivado IDE default color scheme. Your colors may be different.

Examining Clock Regions

The Device window shows the clock regions on the device, which is useful to help guide floorplanning and placement. The Clock Region window lists all the clock regions in the device. The Clock Region Properties window displays clock region information to indicate potential clock contentions prior to implementation. Viewing clock domains is covered later in the tutorial.

1. Select **Window > Clock Regions** to display the Clock Regions window.
2. **Select** Clock Region **X0Y2** listed in the Clock Regions window.

The clock region is highlighted in the Device window, and the Clock Regions Properties window is displayed.



TIP: The Clock Regions and the Clock Regions Properties windows are initially overlapping. To see them as shown in [Figure 11](#), you must select one of the windows and move it up onto the Sources window. Refer to the *Vivado Design Suite User Guide: Using the Vivado IDE* ([UG893](#)) for more information on moving windows.

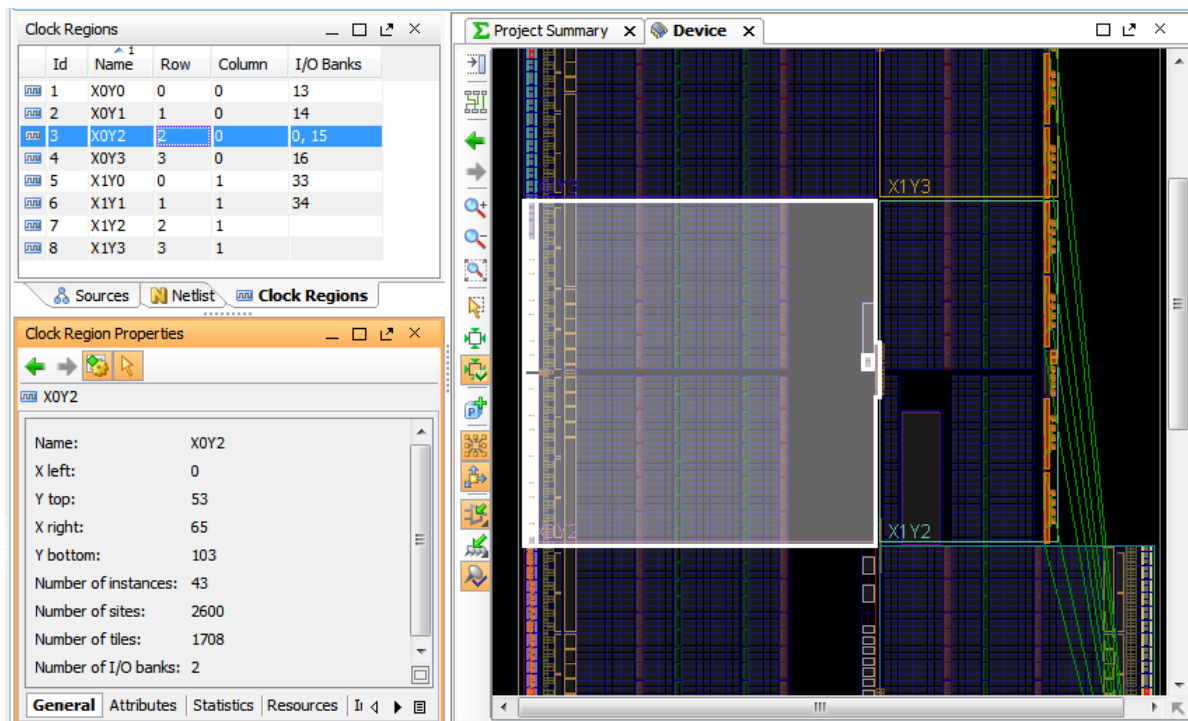




Figure 11: Viewing Clock Region Properties

3. Click the Properties tab to examine the **Clock Region Properties** window.
4. Click the **Statistics** tab of the Clock Region Properties, and scroll to examine the available device resources of the selected clock region.

5. Click the **Resources** tab of the Clock Region Properties, and select and examine the locations of the BUFR and IDELAYCTRL sites.
6. **Select** one of the **BUFRs**, and notice it is highlighted in the Device window.
Note: If Auto Fit Selection is enabled, the Device window is fit to the BUFR on selection.
7. In either the Device window or the Properties window, right-click and select **Mark** to mark the selected BUFR site.
8. In the Device window, use **Zoom Fit** to fit the whole device.
 Notice the mark on the BUFR allows you to quickly locate the site in the zoomed out Device window.
9. Select the **Unmark All** button on the main tool bar menu to remove the mark. 
10. In the Clock Region Properties window, click the **I/O Banks** tab to examine the I/O Banks in the selected clock region.
11. In the Clock Region Properties window, **select** one of the **I/O banks**.



TIP: To locate the I/O Banks tab, you may need to scroll the Clock Region Properties window using the displayed arrows. 

The selected I/O bank is highlighted in any open windows, such as the Device window, the I/O Ports window, and the Package and Package Pins windows. You can open and switch between these different windows to see the various representations of the selected I/O bank.

12. Select **Window > Clock Resources** to open the Clock Resources window.

In the Clock Resources window, the Vivado IDE displays a simplified view of the I/O and clocking resources in a spreadsheet-like interface. This window shows the relationship and interactions between regional and global clocking resources on the device: BUFRs, BUFIOs, BUFGs, MMCMs and GTs, while maintaining proper relative positioning between these resources. You can use the Clock Resources window to plan and place critical clocking and I/O logic prior to placing and routing the rest of the design.

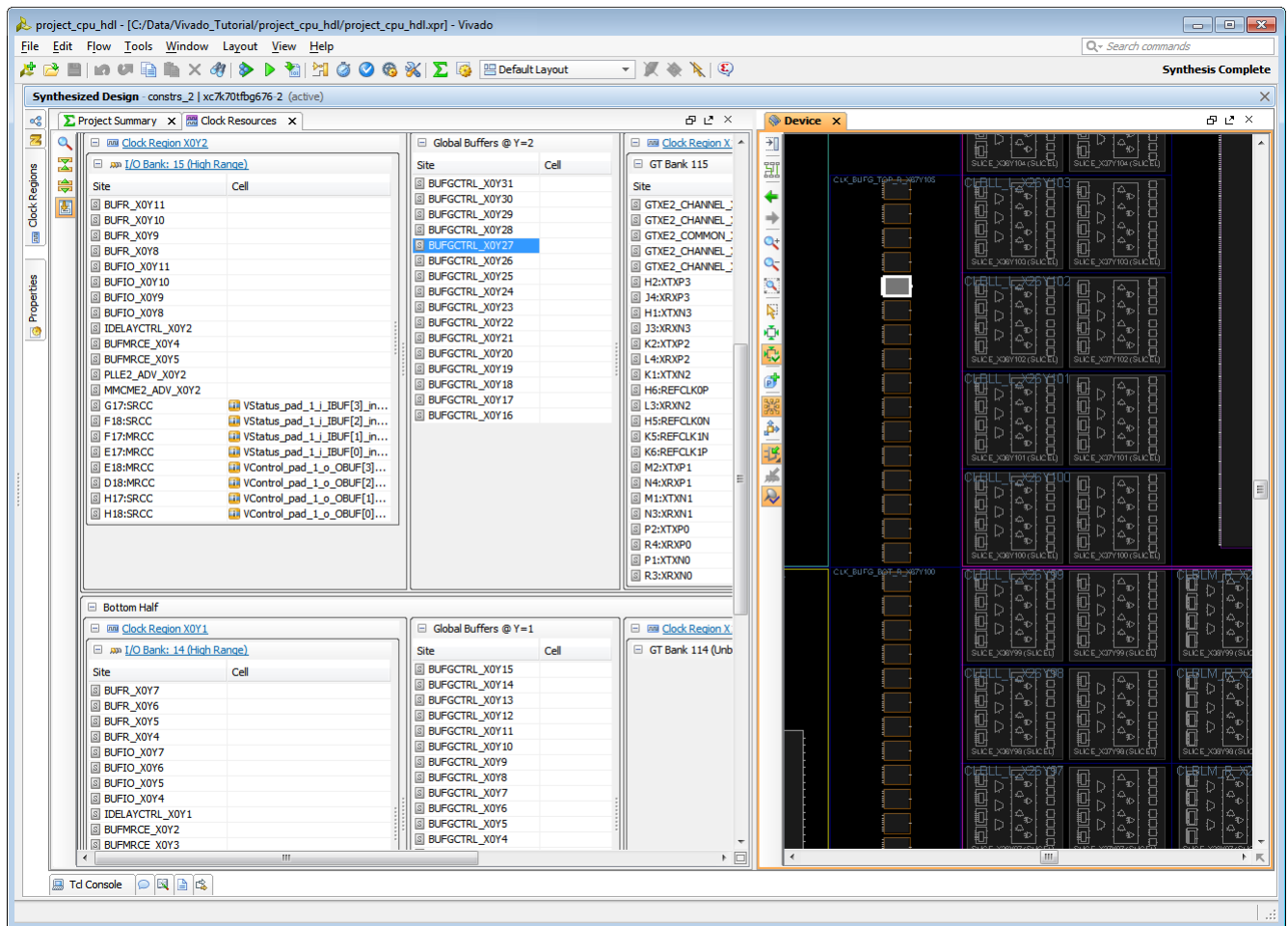
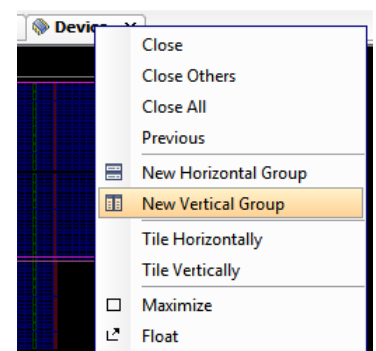


Figure 12: Clock Resources and Device windows



In the target part for this design, the Clock Resources window shows eight clock regions, in a 2x4 matrix, numbered from X0Y0 in the lower left to X1Y3 in the upper right. The device is divided into a top and a bottom half, which can be expanded and collapsed as needed to further simplify the display. Each clock region displays I/O banks containing clock-capable pins (CCIOs), BUFIOs, and BUFRs. In the center column of the device are global clock buffers (BUFGs) for managing global clocks on the device. The clock regions on the right side of the device have either an I/O bank or a column of gigabit transceivers (GTs).

In [Figure 12](#), the Clock Resources window and the Device window are sharing space in the graphical windows area of the Vivado IDE. You can arrange windows in this manner using the Tile Vertically, Tile Horizontally, or the New Vertical Group, New Horizontal Group commands. These commands can be found by right clicking, to open the popup menu, in the window tab of a graphical window, such as the Device window or Clock Resources window. See *Vivado Design Suite User Guide: Using the Vivado IDE* ([UG893](#)) for more information.



13. **Close** the **Clock Resource** window by clicking the **X** in the tab.

Step 3: Exploring the Logical Hierarchy

1. Click the Netlist window tab, and if needed, click the **Collapse All** button .
2. Click the plus sign next to the `usbEngine0` module to examine the hierarchy. .

The Netlist window should now look like [Figure 13](#).

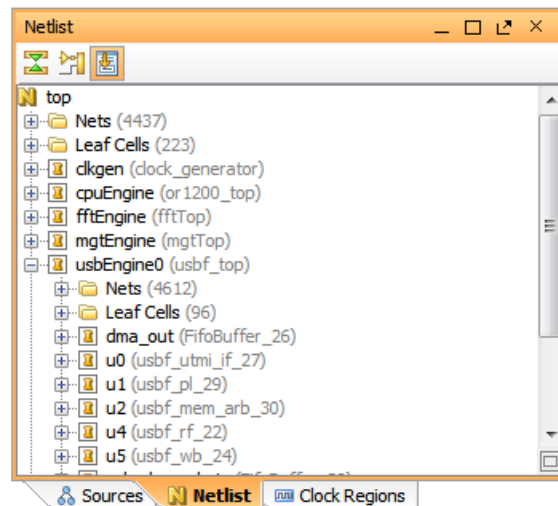



Figure 13: Expanding the Netlist Window

3. Click the plus sign next to the **Nets** folder of the `usbEngine0`. .
- These are the nets inside the `usbEngine0` module. You can collapse the Nets folder by clicking the '-' next to it.
4. Click the plus sign next to the **Leaf Cells** folder of `usbEngine0` to expand and examine the contents.
- These are the leaf cells inside the top-level of the `usbEngine0` module, not inside any of the sub-modules. There is also a Leaf Cells folder at the top level of the Netlist window that contains the leaf cells in the top-level of the design.
5. **Select** the **u4** module.
6. Right-click and select **Show Hierarchy**, or press **F6**.

The Hierarchy window opens in the Workspace. The Hierarchy window is used primarily during design analysis and floorplanning. The Hierarchy window displays the hierarchical relationship of the modules. The width of the blocks in the Hierarchy window are relative to the device resources consumed by each block. It lets you see how a timing path traverses the logic hierarchy, or gauge how big a module is before floorplanning the module.

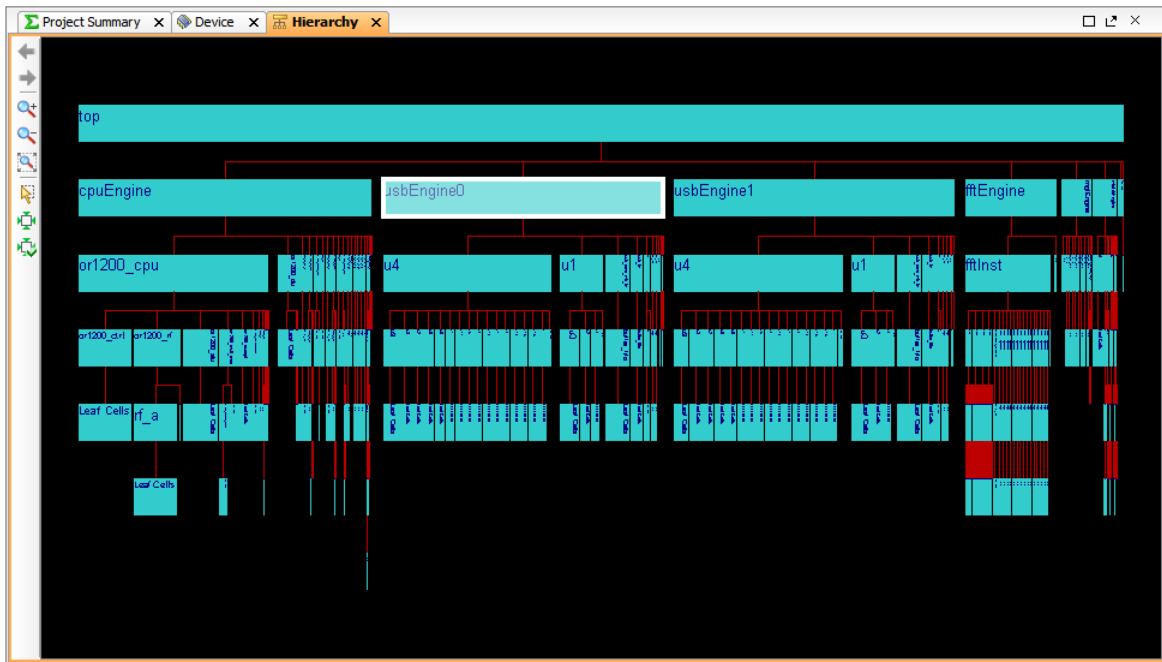



Figure 14: The Netlist and the Hierarchy Windows

You can select modules directly from the Hierarchy window for floorplanning. Logic selected in other windows is highlighted in the Hierarchy window.

7. **Select** any **module** in the **Hierarchy** window, and notice that the module is selected in the Netlist and other windows, as well. Selection works across windows.
8. Click the **Unselect All** main toolbar button, or press **F12**. 

Cross probing RTL Source Files


1. **Select** the **u4** block in the Hierarchy window, or in the Netlist window.
2. Right-click and select **Go to Source** from the popup menu, or press **F7**.

Notice the RTL source file is now open in the Text Editor, with the line number highlighted where the logic instance was derived.

You can use this feature, and the Go to Instatiation command, to quickly locate the definition of a module, or a specific instantiation of a module in the RTL source files.



TIP: The Text Editor displays some file formats with context-sensitive highlighting to make it easy to identify keywords and comment lines within the file.

3. **Select** several other **logic instances** in either the Netlist or Hierarchy window and view the RTL source files.
4. **Close** all open RTL source files in the **Text Editor**.
5. Click the **Unselect All** main toolbar button, or press **F12**. 

Step 4: Displaying Design Resources Statistics

The Vivado Design Suite utilization analysis provides design statistics to help you determine the optimal device to fit the design. The tool helps you see how the logic resources are split between modules. It is easy to explore multiple device types to determine the best overall utilization and performance estimations.

Viewing the Resource Estimates of the Entire Design

1. In the Flow Navigator, under Synthesized Design, select **Report Utilization**.

This opens the Report Utilization dialog box.

2. Click **OK** to accept the defaults.
3. Scroll down the **Utilization** window.

The Utilization window shows device resource utilization sorted by logic type on the left side, and displayed according to the design hierarchy on the right side.

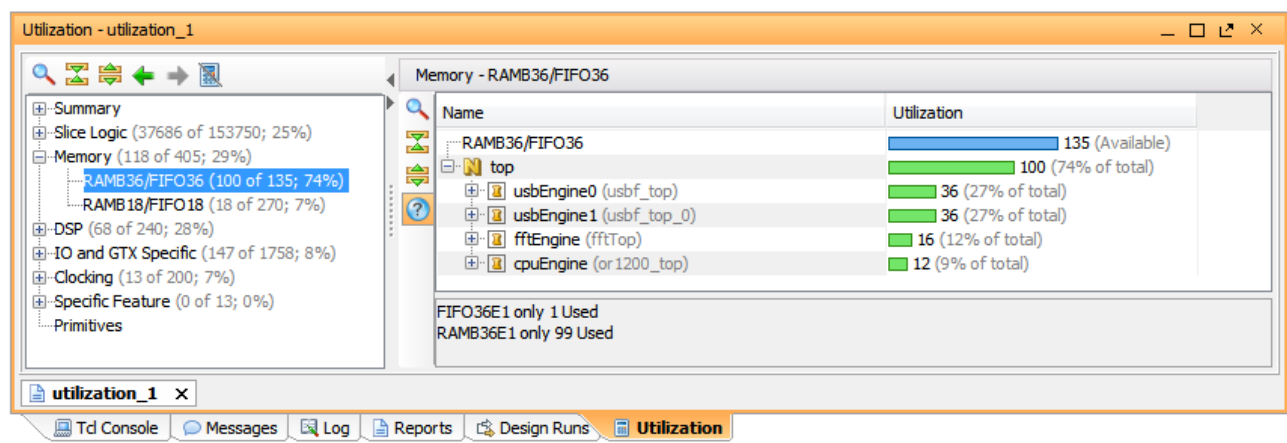


Figure 15: Resource Utilization window

4. In the Utilization window, click **Memory > RAMB36/FIFO36**

Notice that the usbEngine blocks are the two largest consumers of block RAM, with each consuming approximately 27% of the available resources on the target device.

5. Click the plus sign next to the `usbEngine1` to expand and examine the hierarchy.
6. Select **usbEngine0** from the Memory section of the Utilization window.

Notice that `usbEngine0` is also selected in other open windows, such as the Netlist window, and the Hierarchy window if it is still open.

7. **Close** the **Utilization** window by clicking the X button in the upper right corner of the window.

Examining Resources in More Detail

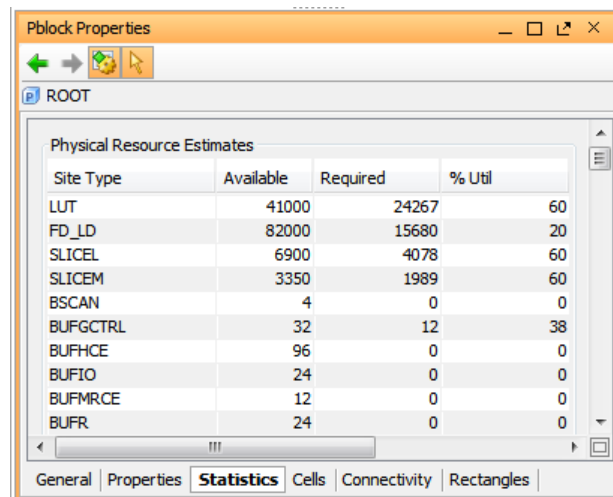
The Pblock Properties window offers you another way to view the device resources, and the resource utilization by the design.

1. Select **Window > Physical Constraints** to open the Physical Constraints window.
2. In the Physical Constraints window, **select** the **ROOT** design.

This opens the Pblock Properties window. The Vivado Design Suite provides the ability to hierarchically divide the design into smaller, more manageable physical blocks (Pblocks). Pblocks can include logic modules and cells from anywhere in the design hierarchy.

The Vivado tool also creates a single Pblock from the root design, or the top-level design, even when there are no other Pblocks defined. In this section you will examine the resources available in the root Pblock.

3. In the **Pblock Properties** window, select the **Statistics** tab.



| Site Type | Available | Required | % Util |
|-----------|-----------|----------|--------|
| LUT | 41000 | 24267 | 60 |
| FD_LD | 82000 | 15680 | 20 |
| SLICEL | 6900 | 4078 | 60 |
| SLICEM | 3350 | 1989 | 60 |
| BSCAN | 4 | 0 | 0 |
| BUFGCTRL | 32 | 12 | 38 |
| BUFHCE | 96 | 0 | 0 |
| BUFIO | 24 | 0 | 0 |
| BUFMRCE | 12 | 0 | 0 |
| BUFR | 24 | 0 | 0 |

Figure 16: Viewing Design Resource Statistics

4. View the **Physical Resource Estimates** displayed in the Pblock Properties window, as shown in [Figure 16](#).
 - **Available:** The available device sites covered by the Pblock, as drawn.
 - **Required:** The number of resources required by the logic assigned to the Pblock.
 - **% Util:** The estimated utilization of each site type. Red text indicates that utilization exceeds 100%. In this case the Pblock may need to be redrawn.

The ROOT Pblock includes the entire design placed onto the target Xilinx FPGA. In this case the Pblock Properties statistics reflect the logic requirements of the whole design, and reflect the resources available on the target part. If the Utilization % is exceeded for the ROOT, you will need to select a different target part.

5. **Scroll** through the **design statistics** displayed in the Pblock Properties window.

The Statistics tab displays the device resource utilization for each type of logic element, carry chain count and longest length, clock report, I/O utilization, and leaf cells and I/O statistics. If the design had any Relatively Placed Macros (RPMs), the RPM count and maximum size information would also be shown.

6. Close the Physical Constraints window.

Step 5: Performing Pre-Implementation Timing Analysis

In this step, you will investigate timing before implementing the design.

At this point, prior to implementation, the Vivado IDE timing engine provides an estimate of what the route delays will be, and does not report on whether the design has met timing. Before running implementation it can be helpful to perform an early Static Timing Analysis, including estimated route delays, to determine the achievability of the timing constraints in the design.

Analyzing Timing End Points with the Slack Histogram

The Slack Histogram shows the timing slack calculated at the endpoint. It provides a visual indication of the timing delays in the design, grouping endpoints into bins based on slack. The histogram lets you visualize how many endpoints have tight timing versus those that have a margin.

This can assist you in determining your next steps if the design is not meeting performance requirements; helping you focus on critical timing paths with negative or low slack values. Endpoints with negative slack are expected to have timing problems. Endpoints with low slack margins in preliminary timing analysis may not meet timing targets during implementation.



IMPORTANT: *The Slack Histogram does not show the complete timing path. You must use the Report Timing command to see the timing of a complete path.*

1. Select **Tools > Timing > Create Slack Histogram**.

This opens the Generate Slack Histogram dialog box, as seen in [Figure 17](#).

2. **Change** the Number of **bins** to 20.

The range of slack values found by the Vivado tool is divided into the number of bins specified, then the timing endpoints are grouped into the appropriate bin based on the slack value at the endpoint. A smaller number of bins can offer a quick summary view of the timing performance of the design. A larger number of bins are used to provide greater detail within a specific range of delays.

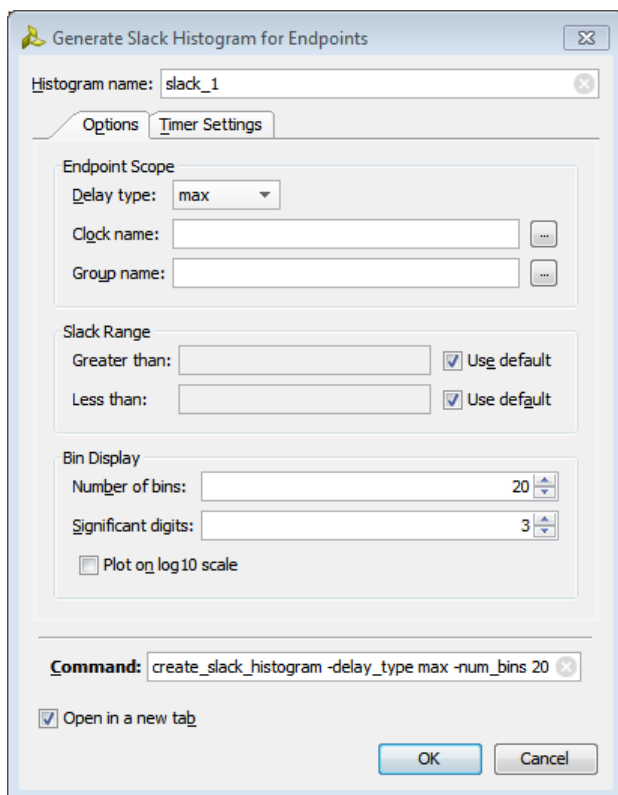


Figure 17: Generate Slack Histogram

3. Click the **Timer Settings** tab.

Examine the Timer Settings tab. You can specify various delay parameters used by the timing engine, to determine how the Vivado tool accounts for route delay when calculating slack. The timer can estimate route delay, or assume there is no interconnect routing delay.

4. Click **OK**.

A histogram displays the slack at the timing endpoints, grouped into bins. A table under the histogram lists the timing endpoints, and the calculated slack value.

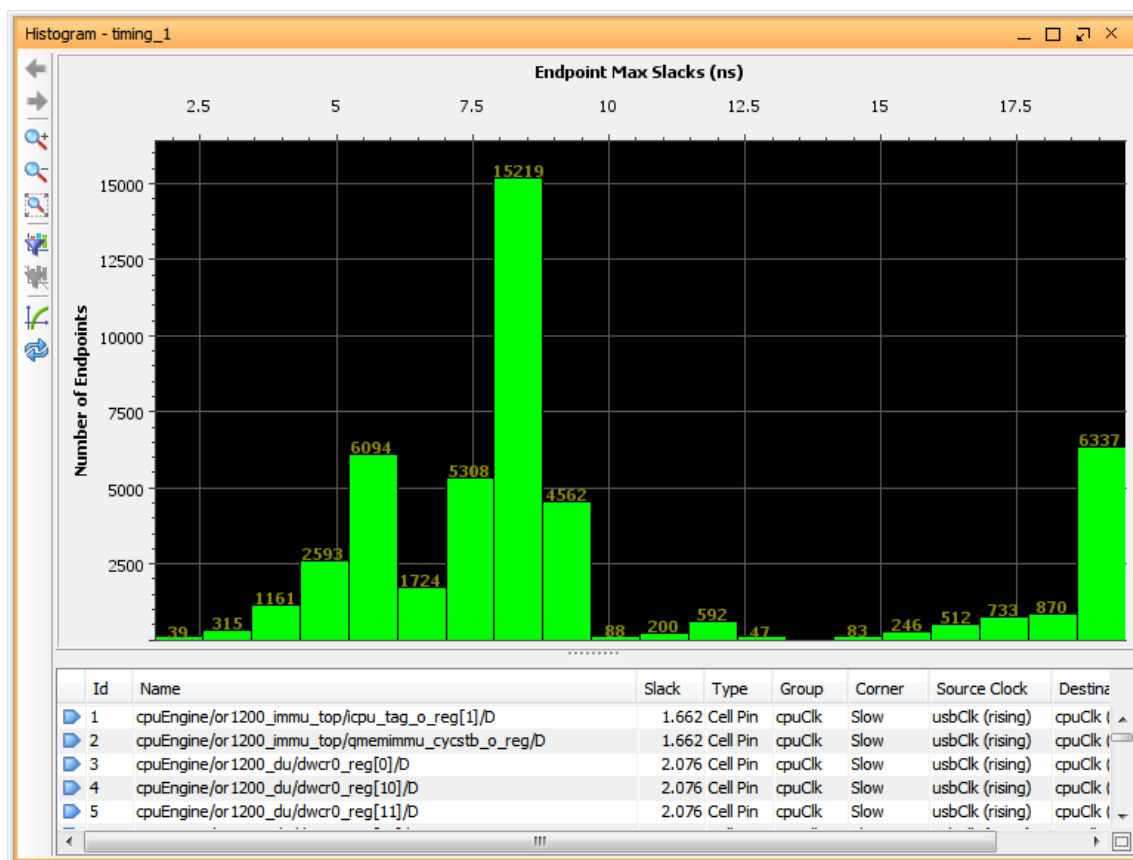


Figure 18: Slack Histogram

If there were endpoints with negative slack in this design, the failing endpoints would be displayed in a red bin, to the left of the 0 value in the X-axis. Endpoints with negative slack are expected to have timing problems.

5. **Click** on various **bins**.

The endpoints listed in the table, under the histogram, are filtered according to the selected bins. The slack range is also updated.

6. **Select** the left most **bin**, which is the bin with endpoints with the lowest slack margins, closest to failing timing.

Scroll down to inspect the endpoints and see the calculated slack values. Negative slack values would be displayed in red.

7. **Select** the first ten **timing paths** listed in the table below the slack histogram.

Use the Shift key to select multiple timing paths from the list.

8. Right click in the table, or in the histogram, and select the **Report Timing** command from the popup menu.

The Report Timing dialog box is opened, with the selected endpoints from the slack histogram listed as shown in [Figure 19](#).

Analyzing the Pre-Placed Timing

The Report Timing dialog box opens displaying the Targets tab. This lets you define the timing paths for the Vivado timing engine to analyze by specifying start points, through points, and endpoints.

The Report Timing command is a static timing path analyzer. You can use Report Timing to view specific timing paths at any point of the flow after synthesis when you need to investigate timing problems, or when you want to report the validity and the coverage of particular timing constraints.

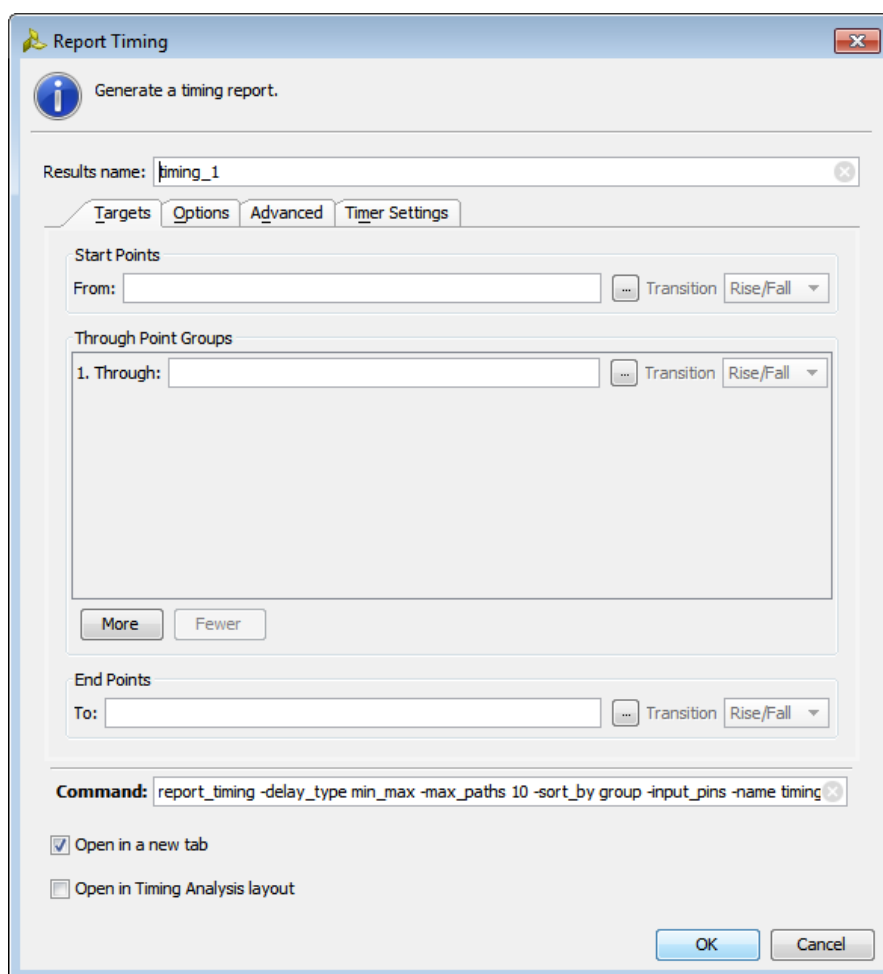


Figure 19: Report Timing – Targets

1. Select the **Options** tab.

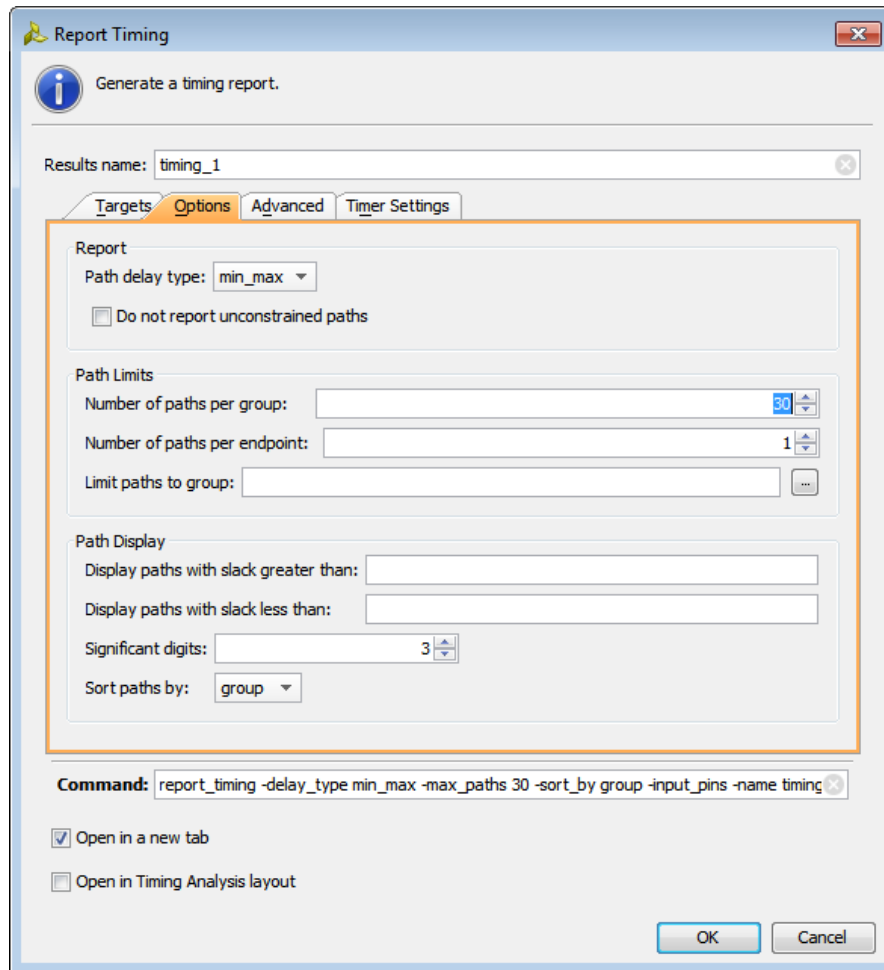


Figure 20: Report Timing - Options

2. In the **Number of paths per group** field, type 30.
3. Click **Advanced** and **Timer Settings**, and examine the dialog box to view the various options available.

Do not change anything on these tabs.

At this point in the design flow, prior to implementation, the Interconnect pull-down in the Timer Settings tab has two values: estimated, and none. Prior to running the Vivado implementation tools, the timer cannot give you actual interconnect delays. The routing delay can be estimated based on a routing delay model, or can be left out of the analysis.

4. Click **OK** to run timing analysis.

The Report Timing window opens.

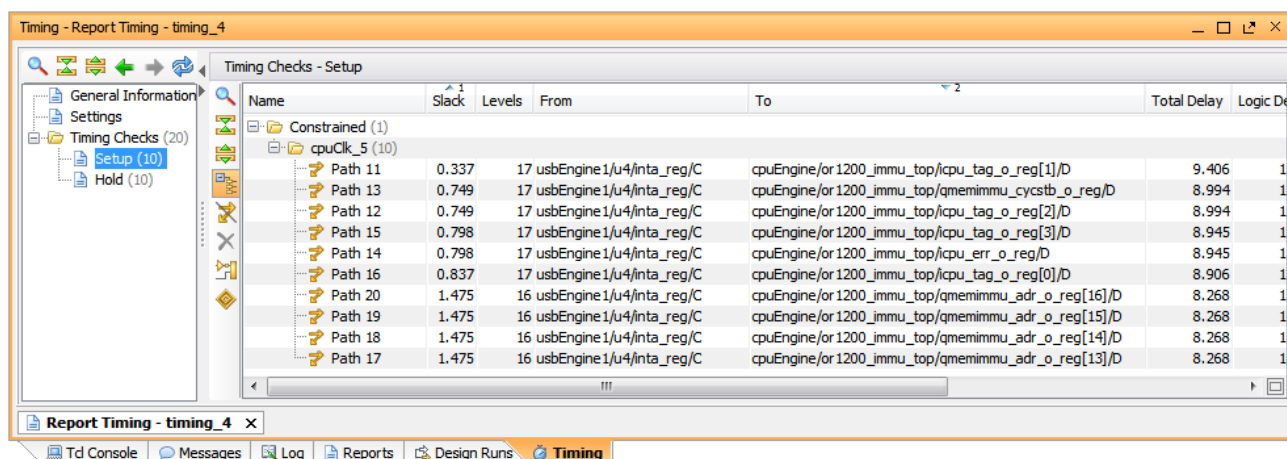


Figure 21: Timing Report

The tree view on the left displays the settings used to generate the timing report, and allows you to display either the timing results for setup or hold analysis. The list of paths in the right side of the timing report is initially grouped by constraint. This can be disabled, to simply list all paths without grouping, by disabling the Group by Constraint command on the timing report tool bar menu.

The report includes columns for the calculated timing slack, start point, endpoint, delay values, the number of logic stages, and the source and destination clocks. Paths that fail timing, appear in red in the report.

Scroll down the list of reported timing paths. Notice that most of the timing paths have start points in the `usbEngine1` module. The timing report can be sorted by clicking on the header of any of the columns in the report.

- Click the **To** column header twice to sort the list by destination.

The report is now reverse sorted by **To** column values. The timing report in Figure 21 is grouped by constraints, and sorted first by slack, in ascending order, and then by destination, in descending order.



TIP: You can sort all of the table style windows in the Vivado IDE in the same way. Click once to sort in ascending order, click twice to sort in descending order. To add a secondary sort, press **Ctrl** and select an additional column header.

- Click **Hold** on the left side of the Timing window to view the paths listed under the hold report.

Prior to implementation, you should focus on Setup violations, as Hold checks are dependent on routing.

- Click **Settings** on the left side of the Timing window to examine the timer settings used when the report was generated.

8. **Close** the Timing Report and the Slack Histogram windows by clicking the **X** button in the window.


Step 6: Implementing the Design

Vivado implementation is a timing-driven flow with native support of industry standard Synopsys Design Constraints (SDC) commands, and Xilinx Design Constraints format (XDC), to specify design requirements and restrictions. Vivado implementation encompasses all of the design steps required to place and route the netlist onto the FPGA device resources while meeting the logical, physical, and timing constraints of the design.



Important! The Vivado Design Suite does not support the UCF format. For information on migrating UCF constraints to XDC commands, refer to the Vivado Design Suite: Migration Methodology Guide ([UG911](#)).

Examining the Constraints File

1. If the Sources window is hidden from view, display it by selecting **Window > Sources**, or selecting the Sources tab as shown in [Figure 22](#).
2. In the Sources window, click the **Collapse All** button, , then expand `constrs_1` and `constrs_2`.

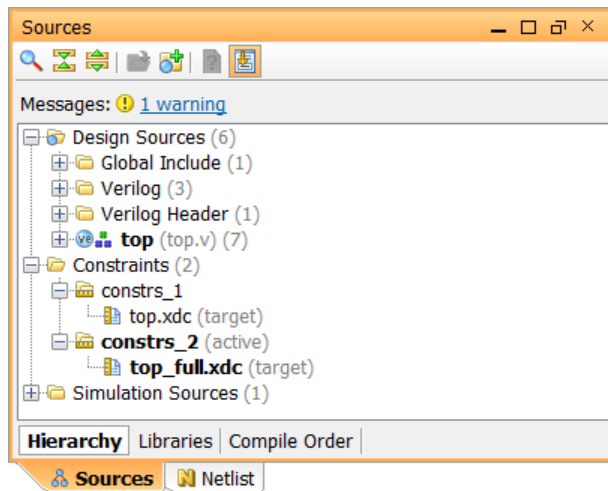


Figure 22: Sources Window

When running the implementation tools, the Vivado Design Suite uses the active constraint set by default.

Make sure that `constrs_2`, with `top_full.xdc`, is the active constraint set.

3. Double-click **top_full.xdc** to open the file, and examine the contents of the constraint file.

4. **Close** the XDC file.

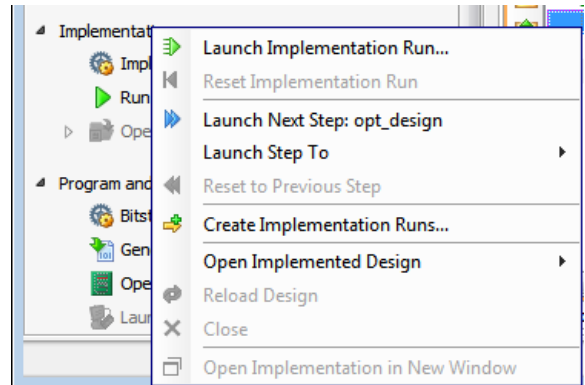
Do not save any changes if prompted.

Defining Implementation Runs

The Vivado IDE allows you to configure and run multiple synthesis and implementation runs simultaneously. This tutorial will demonstrate how to do that, but you will only launch one implementation run in order to conserve time.

1. In the Flow Navigator, right-click on Implementation, and click **Create Implementation Runs** in the popup menu.
2. Click **Next** in the Create New Runs confirmation dialog.

The Configure Implementation Runs dialog box opens, letting you create and configure one or more implementation runs, selecting strategies to use during the run.



The first implementation run listed should be `impl_2`. The strategy listed will depend on whether you have created any user-defined implementation strategies. See the *Vivado Design Suite User Guide: Implementation (UG904)* for more information on standard available strategies, and on defining new strategies.

3. Click the **Strategy** pull-down menu to select the **Flow_RuntimeOptimized** strategy listed under Vivado Strategies.
4. Click **More** to add a new run, called `impl_3`.

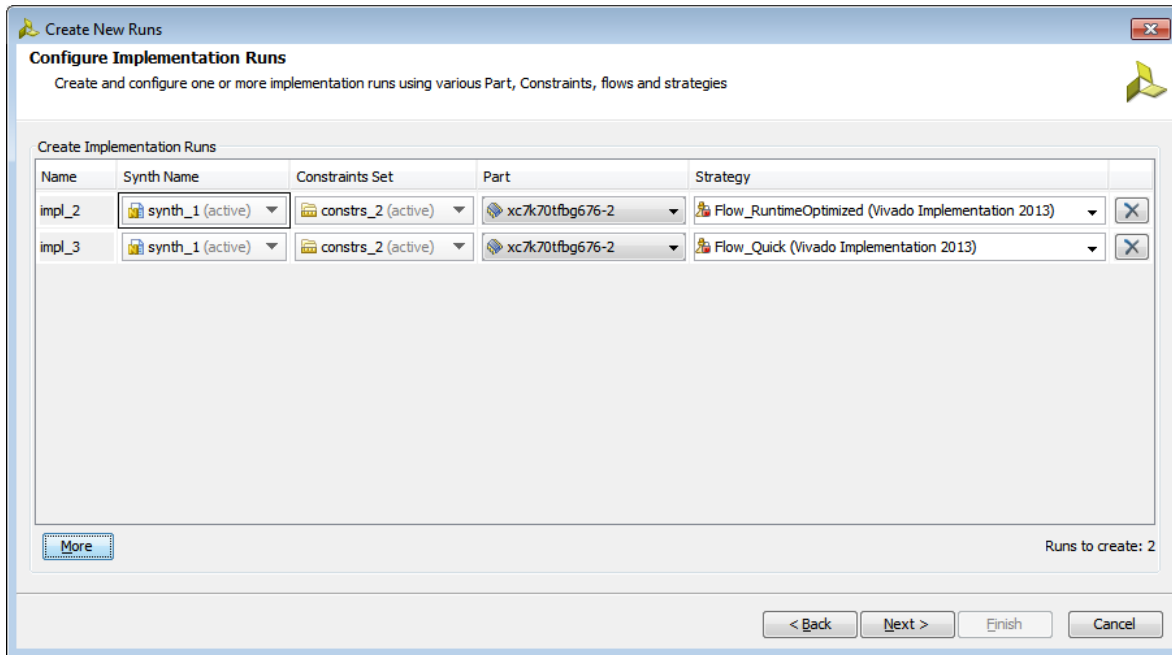


Figure 23: Choose Implementation Strategies

- Click the **Strategy** pull-down menu to select the **Flow_Quick** strategy.

This is the implementation run you will be using later in the tutorial. You can also configure Synth Name, Constraints Set, and Part when defining the run. However, at this time, you will use the default settings.

- Click **Next** to open the Launch Options dialog box.
- Review** the available **launch options**. You can:
 - Launch the runs locally to run implementation on the local computer.
 - Launch the runs remotely (on Linux only), to run implementation on remote hosts.
 - Compile run scripts for later use, to create batch scripts that can be run either locally or remotely.
 - Save the runs without launching implementation, which is what you will choose now.
- Select Do not launch now, and click Next.
- Review** the Create New Runs Summary, and click **Finish**.
- Select the `impl_3` run in the Design Runs window, and use the **Make Active** command from the popup menu to make it the active run.

Only one synthesis and one implementation run can be active in the Vivado tool. By default, the Vivado IDE directs commands to, and displays reports and messages from the active run.

Your Design Runs window should look as follows:

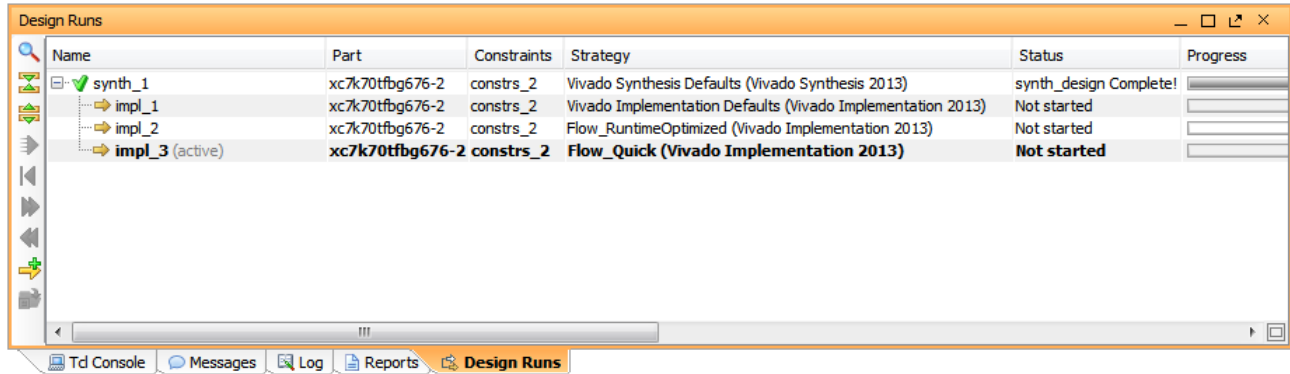


Figure 24: Design Runs window

Launching Runs

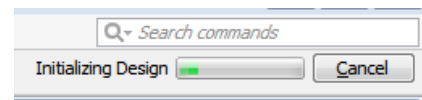
The next step in the flow is implementing the design through the Vivado IDE implementation tools. The Vivado implementation process includes logical as well as physical transformations of the design, and consists of the following sub-processes:

- **Opt Design:** Optimize the logical design to make it easier to fit into the target Xilinx FPGA.
- **Power Opt Design:** Optionally optimize elements of the design to reduce power consumption of the implemented FPGA.
- **Place Design:** Place the design onto the target Xilinx device.
- **Phys Opt Design:** Optionally optimize the timing of the design by replicating drivers of high-fanout nets to distribute the loads.
- **Route Design:** Route the design onto the target Xilinx device.

See the *Vivado Design Suite User Guide: Implementation* ([UG904](#)) for more information on these sub-processes of implementation, or the *Vivado Design Suite Tcl Command Reference Guide* ([UG835](#)) for information on the specific Tcl commands

1. Select **Run Implementation** in the Flow Navigator.

Implementation takes several minutes to complete. The status of the implementation run is displayed in the project status bar at the upper right of the Vivado IDE.



While you are waiting for implementation to complete, you can experiment with some of the analysis features available for use with the synthesized design. Commands such as the Report Clock Networks and Report Clock Interaction can be run from the Flow Navigator on the open synthesized design while implementation is running.

The Implementation Completed dialog box displays when the run is finished.

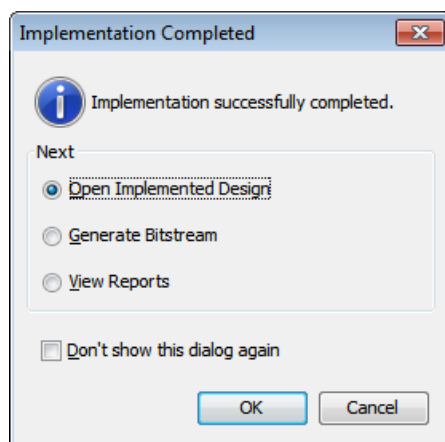


Figure 25: Implementation Completed

2. Select **Open Implemented Design** in the Implementation Completed dialog, and click **OK**.
3. Select **Yes** if prompted to close the synthesized design before opening the implemented design, and **Don't Save**, if prompted.



TIP: This prompt might have been turned off if you elected to never show this dialog again. To restore this prompt, go to **Tools > Options > Window Behavior** and check the **Show dialog before switching to a different design** checkbox.

The results from the implemented design are imported into the Vivado IDE as shown in [Figure 26](#). The Device window shows the placement of all logic instances onto the available device resources.

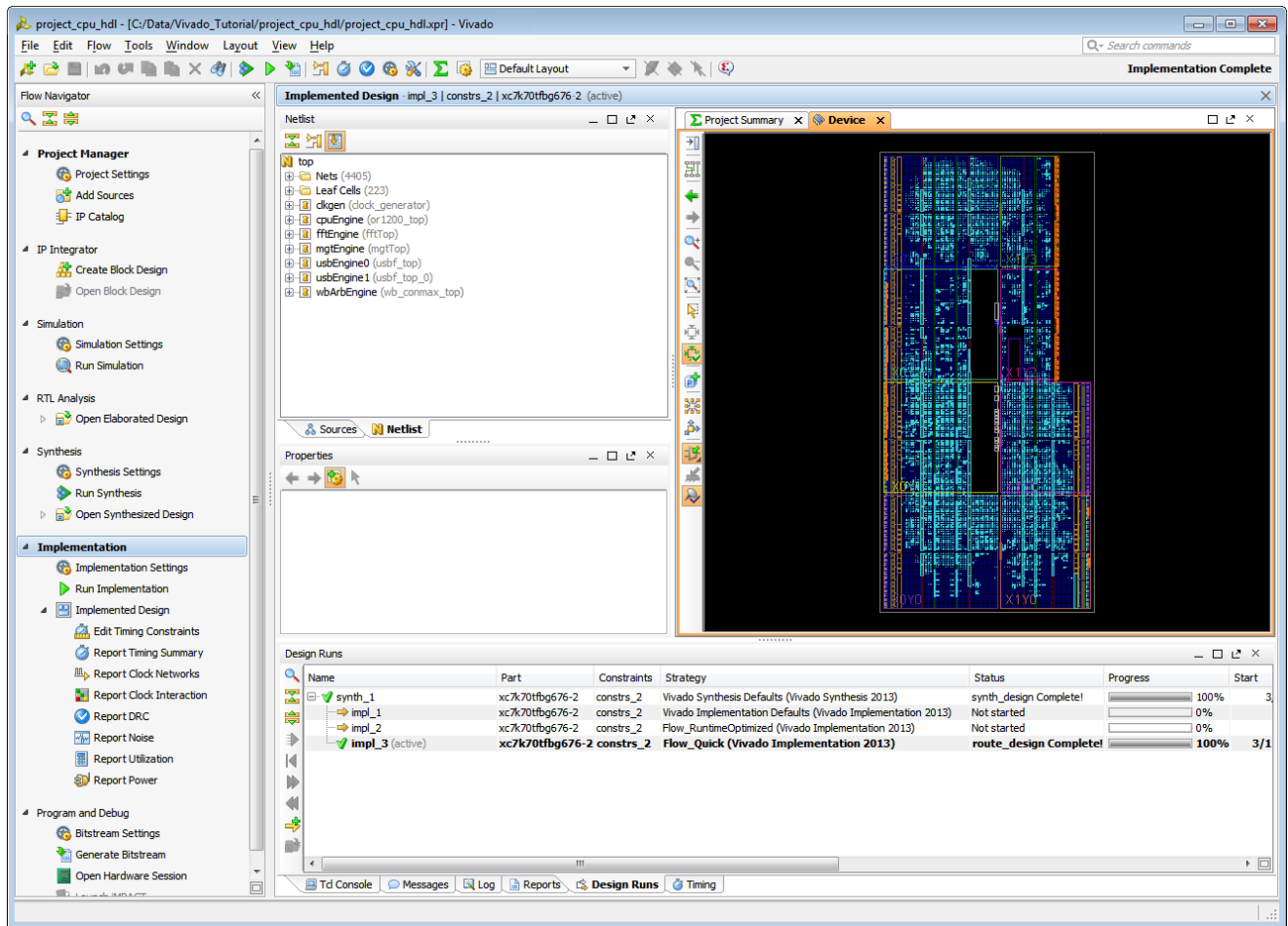



Figure 26: Implemented Design

- Click the **Messages** window tab at the bottom of the Vivado IDE, to display the Messages window.

Note: The Messages window collects all information, warning, and error messages from the various tools of the Vivado Design Suite.

- Click the **Collapse All** toolbar button to display the various message categories .
- Expand the **Implementation** messages.

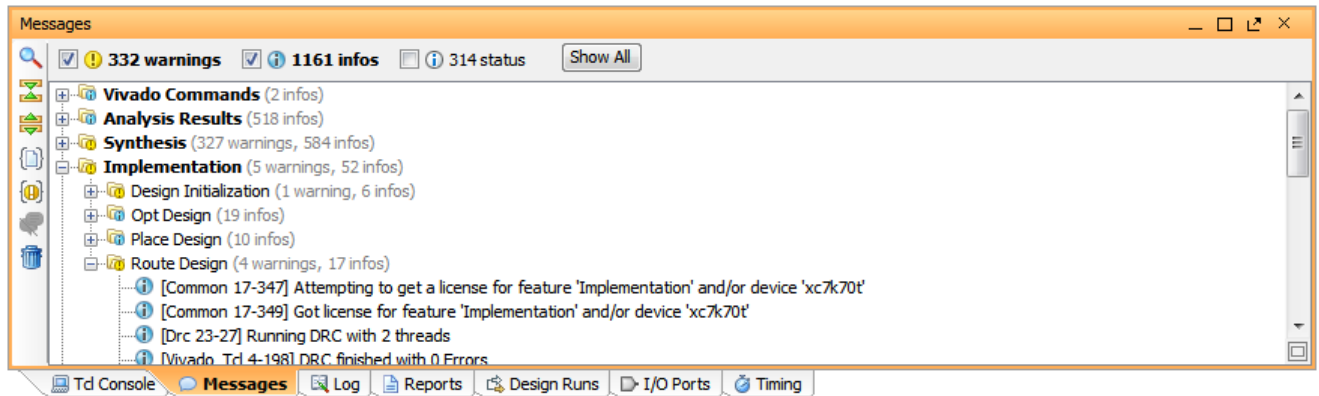


Figure 27: Implementation Messages

7. Expand **Route_Design** folder to see the messages generated at this step.
8. Click the **Reports** window tab to display the various reports generated by different tools in the Vivado Design Suite.
If the Reports window tab is not displayed, select **Window > Reports** to display the Reports window.
9. **Double-click** any **reports** listed in the Reports window to open those reports in the Vivado IDE text editor.
10. **Close** any open **report** files.

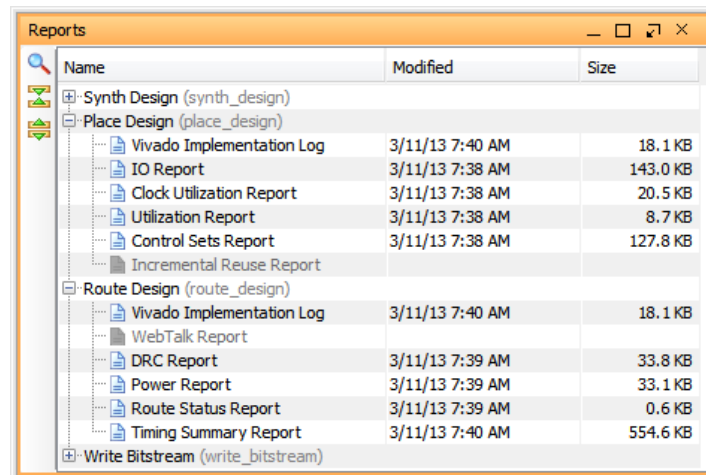


Figure 28: Implementation Reports

Conclusion

In this lab, you have opened the CPU HDL project, and examined various aspects of the design using tools available within the Vivado IDE. You have examined the hierarchy of the design, analyzed the timing slack histogram, and the timing report prior to implementation, and synthesized and implemented the design. Finally, you examined the messages and reports generated by the Vivado implementation tools.

This concludes Lab #1. You can proceed at this time to Lab #2, or use the following steps to exit the tool:

1. Click **File > Close Project** to close the CPU HDL design.
2. Click **File > Exit** to exit the Vivado Design Suite.

Lab 2: Timing Closure Techniques

This lab continues from the end of Lab #1 in this tutorial. You must complete Lab #1 prior to beginning Lab #2. If you closed the tool, or closed the tutorial project at the end of Lab #1, you will need to open them again.

1. Start by loading the Vivado Integrated Design Environment (IDE).
 - Launch Vivado IDE from the icon on the Windows desktop, or
 - Type **vivado** from a command terminal.

The Vivado IDE opens.

2. From the main menu, click **File > Open Recent Project** and select the project you saved in Lab #1.
3. After the project has finished opening, from the Flow Navigator click **Open Implemented Design**.

Step 1: Analyzing Post-Implementation Timing

With the design implemented onto the target device, you can begin to explore the design and constraints to resolve any challenges with meeting the timing requirements of the design. The Vivado Design Suite offers an array of tools and techniques to help you achieve timing closure.

The Report Timing Summary command creates a set of timing reports intended to provide sign-off level timing analysis for the post-implemented design. You can analyze timing results from the implementation process to drive any floorplanning that might be required to meet timing requirements.



IMPORTANT: Only the Vivado timing analysis Report Timing Summary command provides a sign-off level timing report.

The Vivado tool runs timing analysis after implementation, and generates a report file you can open in the Text Editor for review. However, to view the report interactively in the Vivado IDE, and cross-select and analyze the design, you must regenerate the report.

Viewing Timing Paths

1. From the Flow Navigator, under Implemented Design, select **Report Timing Summary** and click **OK** to run Vivado timing analysis with the default settings.

The Timing Summary window opens, as shown in [Figure 29](#). The left side of the window displays a tree view of different reports available in the Timing Summary. The right side of the window displays the details of the selected report. In the figure below, the Design Timing Summary is selected.

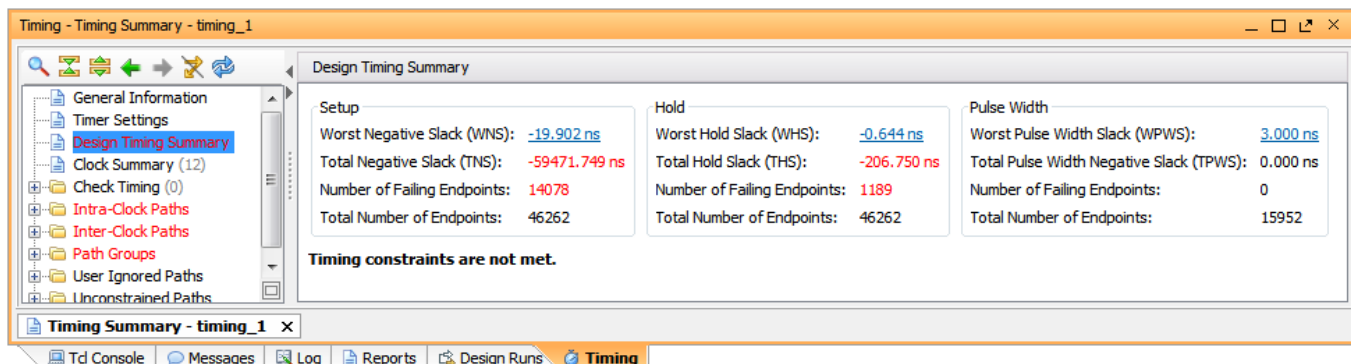


Figure 29: Report Timing Summary

The Design Timing Summary reports timing slack values, such as Worst Negative Slack (WNS), showing whether the design satisfied the timing constraints. Any negative slack values, or a number of failing endpoints, are an indication of timing problems.

The Flow_Quick strategy employed by the implementation run, `impl_3`, is intended to perform a quick placement and routing of the design to assess resource utilization and see if the design will fit onto the target part. This strategy ignores timing constraints, and makes no effort to meet timing, as is seen clearly in the results

In this design, the Design Timing Summary reports many paths which fail timing. You will investigate the timing paths with the worst slack.

2. **Click** the hyperlinked value for the **Worst Negative Slack** (WNS) to display the timing path with the worst slack.

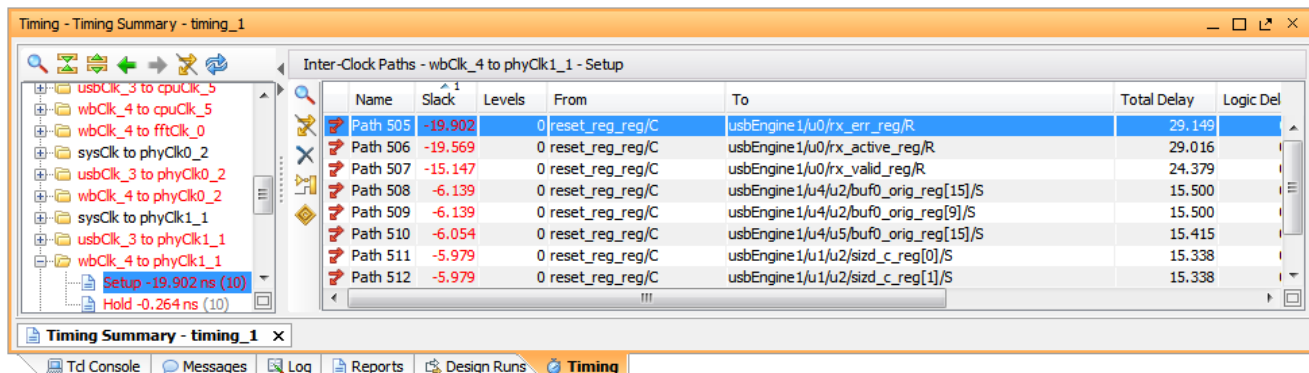


Figure 30: Worst Negative Slack

3. Select the first path in the list.

Notice that the worst timing paths in the report connect to the `usbEngine1` module. This information will guide your floorplanning strategy later in this tutorial.

4. Right-click and select **Mark** in the popup menu.

In an implemented design, selecting the timing path in the report also selects the path in the Device window. The Device window can make it easier to see timing challenges due to placement and routing, and help you visualize how floorplanning could improve timing.

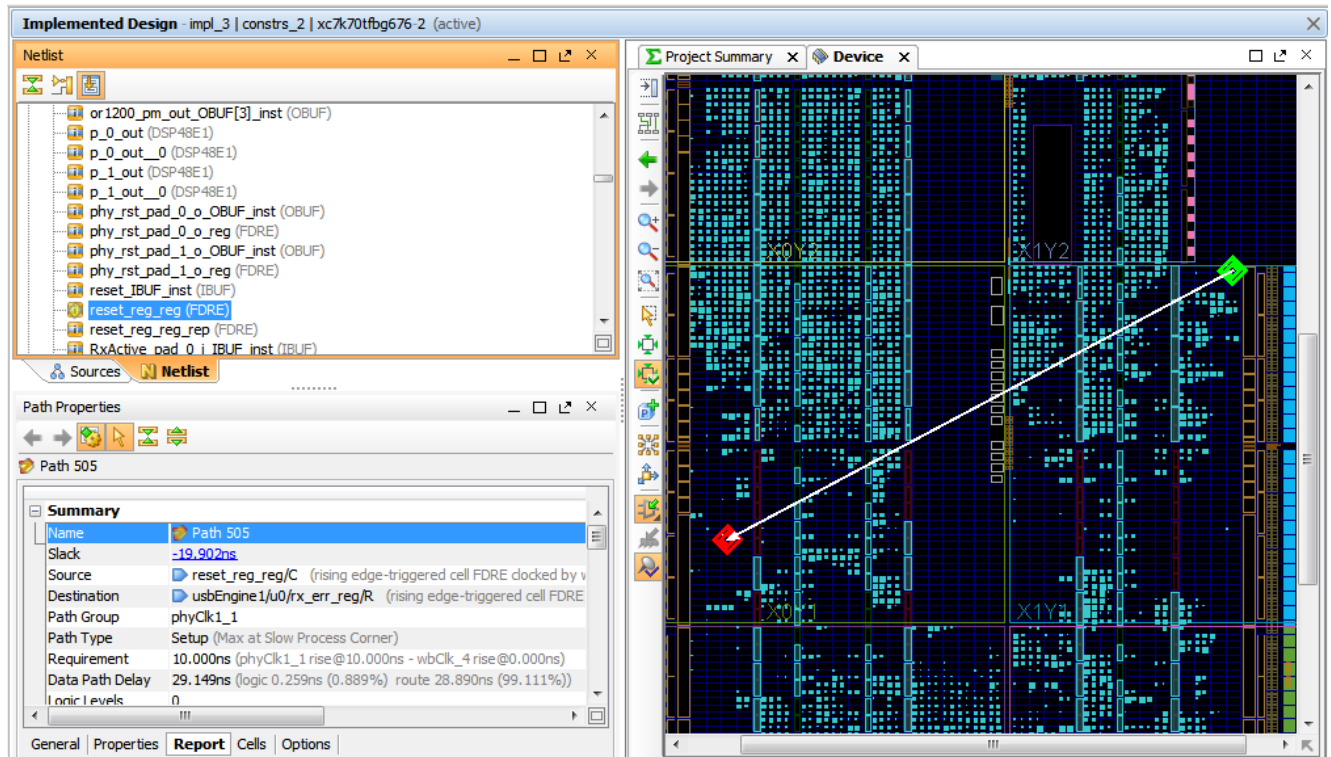


Figure 31: Marking the Timing Path

The selected path is marked with a green diamond showing the timing path *start* point, and a red diamond showing the timing path *end* point. Yellow diamonds would mark any *through* points in the path.

5. Right-click and, from the popup menu, select **Unmark**, or select the **Unmark All** button from the main toolbar to clear the marks.

Viewing the Routed Path

1. In the Device window, select the **Routing Resources** toolbar button to enable the display of routing resources. 

When enabled, the Routing Resources command displays a graphical representation of the device components, and the FPGA interconnections between the components. This provides a detailed view of the available routing resources on the device, and shows the specific resources used by a selected path, as seen in [Figure 32](#).

2. In the Device window, select the **Auto Fit Selection** toolbar button to enable automatic zoom for each newly selected object. 

Select various paths in the Timing Summary window, and notice the detailed routing displayed for each timing path in the Device window. This routing resources view can provide you a better understanding of the interconnect portion of the path delay, and help you resolve timing problems.

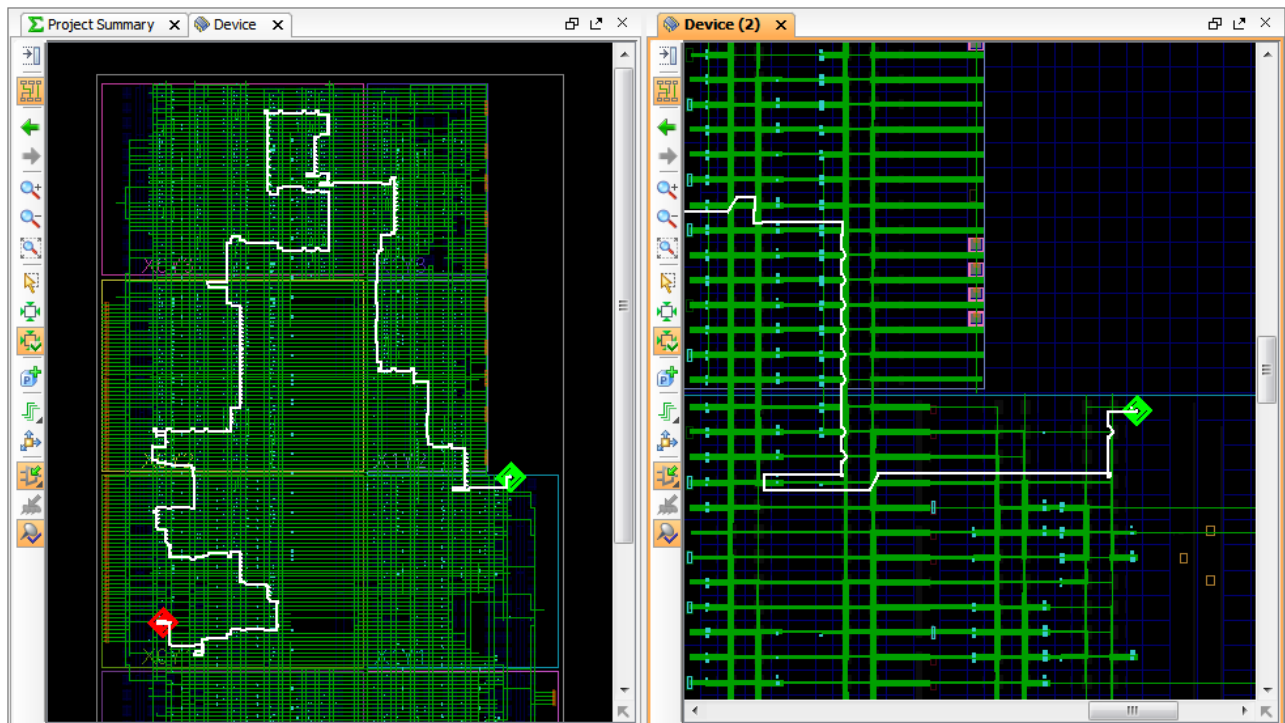


Figure 32: Routing Resources in the Device window

3. **Zoom** closely into a timing path displayed in the Device window to see the details of the routing resources.

When looking at the routed path, it may seem to take an indirect route between connection points. This is because the Vivado router considers many different aspects of a design. If the design is meeting timing, other considerations such as routing density and power analysis, or timing requirements on other paths may affect the route solution.

- Click the **Unselect All** button on the main toolbar menu, or press **F12**, to unselect all currently selected objects.
- In the Device window toolbar menu, select the **Routing Resources** toolbar button again to disable the detailed routing view.

Notice the placed cells are now displayed in the Device window again, without the added details of the routing resources.

Viewing Path Properties

When you select paths in the Timing Summary window, the Path Properties window opens to show a summary and details of the timing path.

- In the Timing Summary window, reselect the first path.
- Select the Path Properties window, and **Maximize**, or **Float**, the window for the selected path, and examine the information provided as shown in [Figure 33](#).

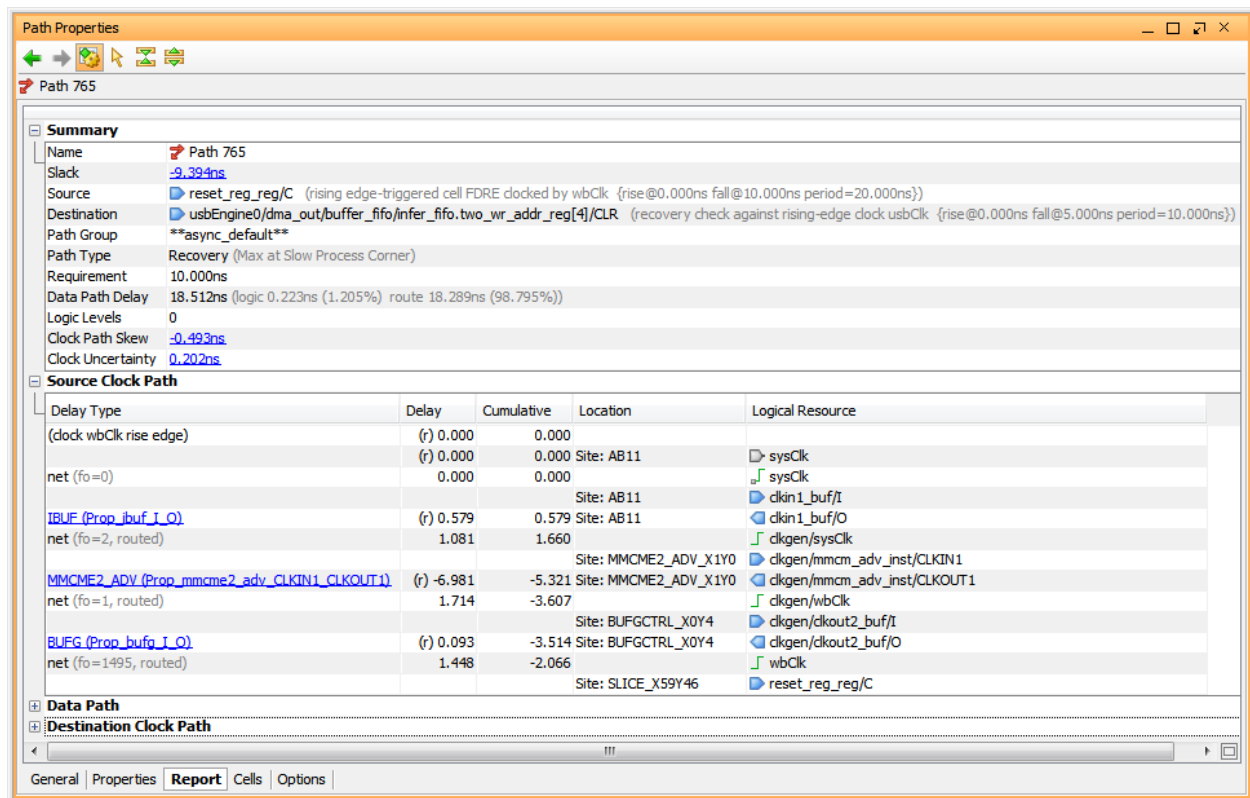


Figure 33: Path Properties window

- Restore** or **dock** the Path Properties window to its original size and location.



TIP: You can maximize or restore a window by double-clicking in the window banner.

Step 2: Exploring the Hierarchy of the Timing Paths

The Vivado IDE offers many tools to help you locate logic associated with the reported timing paths. In this step of the tutorial, you will use the Schematic and Hierarchy windows to visualize the logic associated with the timing paths with the worst delay.

1. On the left side of the Timing Summary window, expand the **Intra-Clock Paths**.
2. Select and expand the **usbClk_3** path by double-clicking the name, and select **Setup**.
3. On the right side of the Timing Summary window, **select** the first path listed.
4. Right click, and select **Schematic** to create a schematic for the selected path.

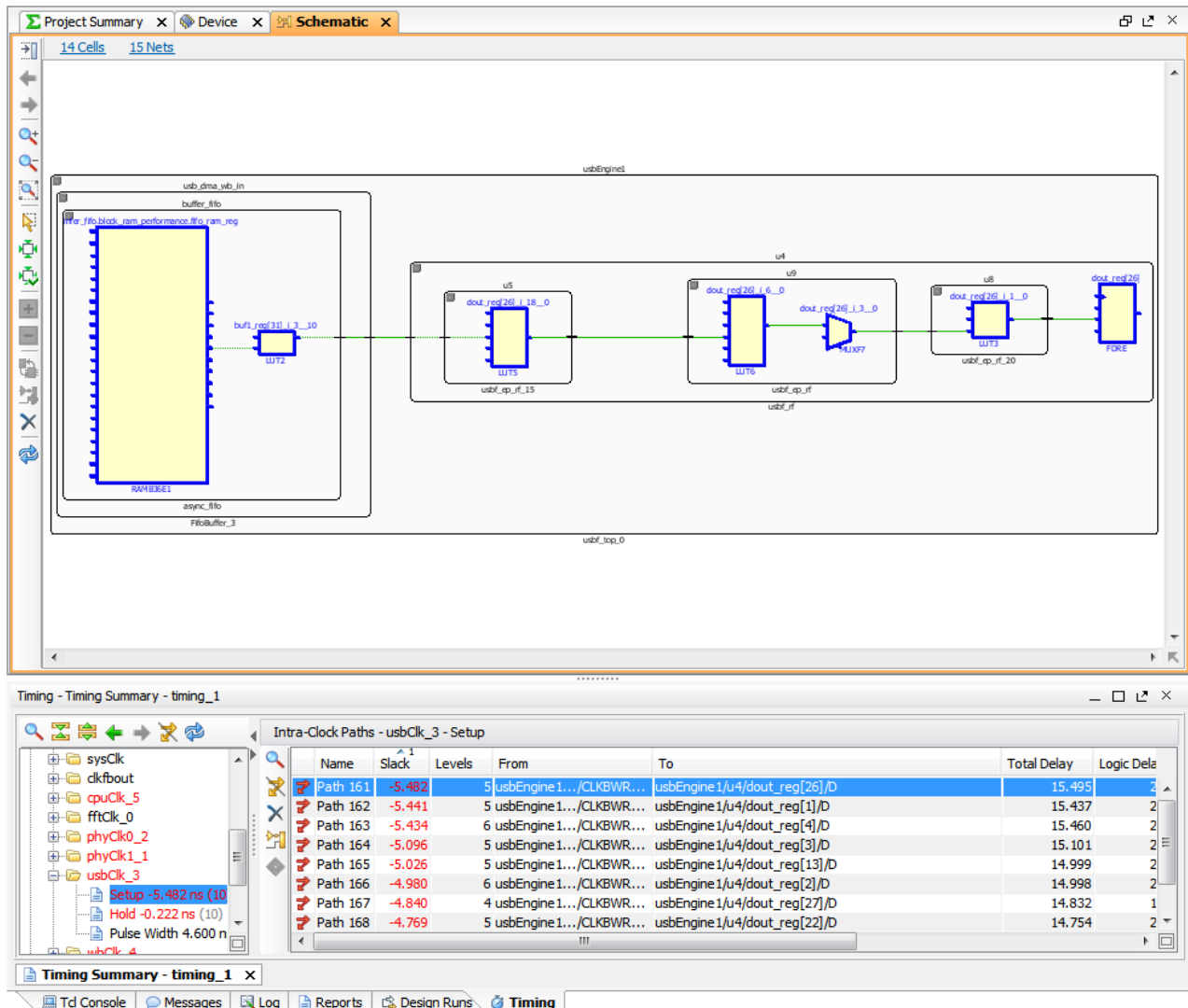


Figure 34: Timing Path Schematic

When you select a timing path in the Timing Summary window, the Vivado tool also selects the cells associated with the start points, end points, and through points on the timing paths. The Schematic window shows all of the logic cells on the selected timing paths.

5. In the Netlist window, click the **Collapse All** button. 

This simplifies the information displayed in the Netlist window, so that the items selected in the next task are more visible, and easier to see.



TIP: Sometimes, because of the volume of information presented in the Netlist window, it is beneficial to collapse the tree view, so you can more clearly see newly selected objects.

6. In the Schematic window, right-click and select **Select Leaf Cell Parents**.


The Select Leaf Cell Parents command selects the hierarchical parent of any currently selected leaf cells, or modules, working up through the hierarchy until it gets to the top-level of the design. The parent modules are also selected in the Netlist window.

7. In the Netlist window, right-click and select the **Show Hierarchy** command from the popup menu, or press **F6**.

The selected logic, which is associated with the timing paths with the worst delays, is all contained in the `usbEngine1` module. There is a duplicate module, `usbEngine0`, which with identical logic, will have many of the same problems. You will explore these two `usbEngine` blocks further for floorplanning.

8. **Close** the **Hierarchy and Schematic** windows, by clicking the X button on the window tabs.
9. Select the first path in the list in the Timing Summary window.
10. Right-click and select **Schematic** from the popup menu.

The timing path logic is shown in the Schematic window as seen in [Figure 34](#). The nested blocks shown in the Schematic window, represent the hierarchical structure of the design. With this view, it is easy to identify the various logic modules associated with the critical path logic. For the selected timing path, you can see the logic is contained within the `usbEngine0` block.

11. Click the **Unselect All** button on the main toolbar menu, or press **F12**. 
12. Select the **RAMB36E1** in the `buffer_fifo` block of the opened Schematic window.
13. Right-click and select **Expand/Collapse > Collapse Outside** from the popup menu.

This will collapse all of the modules outside of the selected cell, so that the schematic is only displaying the hierarchy of the selected block RAM.

14. Double-click the outside of the **ADDRARDADDR[15:0]** pin, in the top left of the block RAM, and notice that the logic connected to it is expanded outward.



TIP: Double-clicking outside the pin is the same as selecting the pin and using the **Expand Cone > To Leaf Cells** command from the popup menu.

15. Select the **infer_fifo.wr_addr_reg[0]** cell, displayed at the top of the newly added logic.
16. Right-click and select **Expand Cone > To Leaf Cells** from the popup menu, or double click the specified cell.

This expands the logic connected to the selected cell outward to the next connected leaf cell, as shown in [Figure 35](#). The logic associated with this expansion includes a reset net, a global clock buffer, and logic in the `usbEngine0` block. You can see the block ram is relatively well insulated from the other logic.

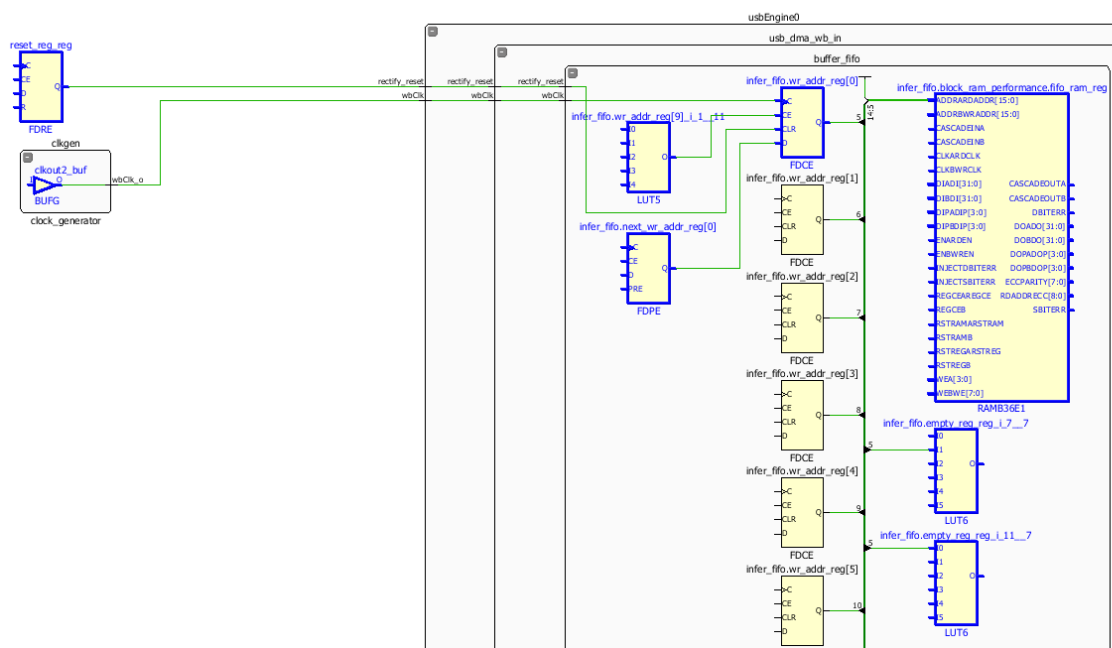



Figure 35: Interactively Expanding Logic

17. In the Schematic window, select the **Previous schematic** button 

You can use the **Previous schematic** and **Next schematic** toolbar buttons to switch between prior views displayed in the Schematic window.

18. In the Schematic window tab, click the **X** button to close the Schematic window.

The input to this block RAM is contained in the `usbEngine` block. You do not have to consider other hierarchy for floorplanning.

19. Click the **Unselect All** button, or press **F12**. 

20. In the Netlist window, click the **Collapse All** button. 

Step 3: Highlighting Module Placement and Connectivity

At this point, you have determined that the worst timing delays are found in the `usbEngine` blocks, and examined the logic in the schematic. You will now evaluate the current module placement to understand how the logic was implemented without floorplanning, to see how placement can be improved. This will lead you to a placement strategy you can implement through floorplanning.

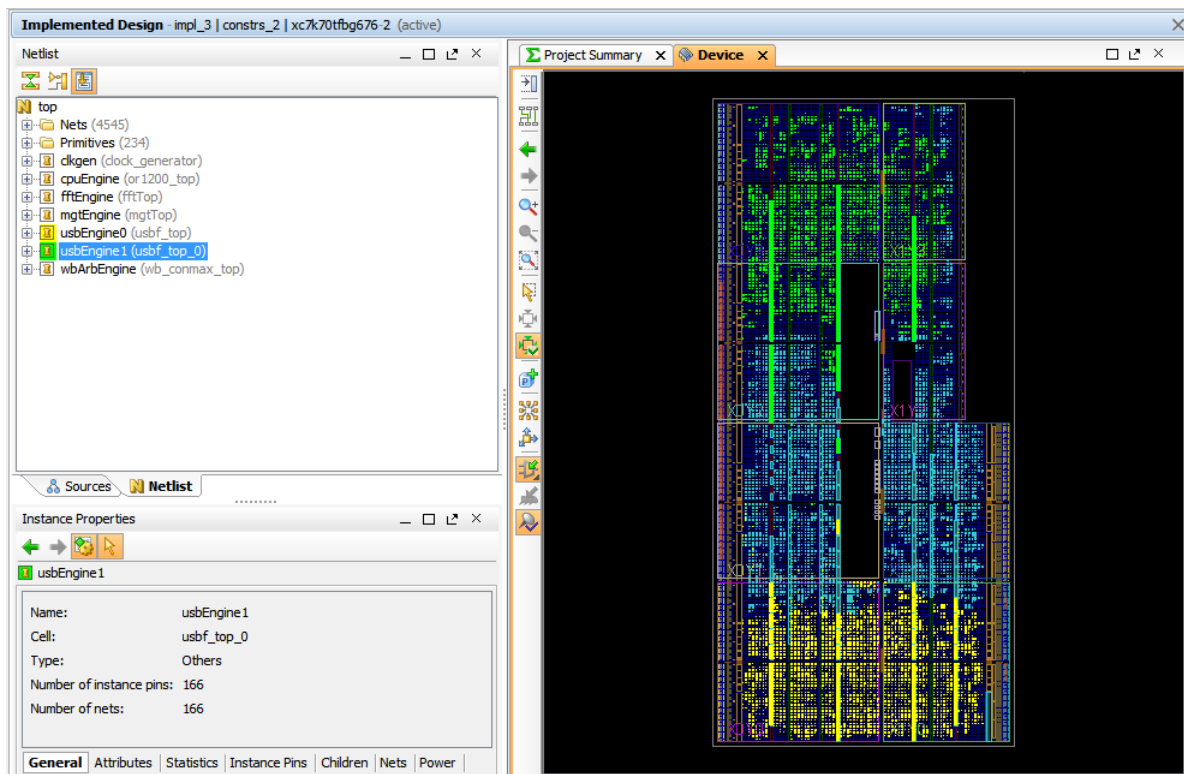





Figure 36: Highlight Module Placement

1. In the Netlist window, select **usbEngine0** and **usbEngine1**.
2. Right-click and select **Highlight Leaf Cells > Cycle Colors** from the popup menu.
The leaf cells in each module are highlighted in a different color as shown in [Figure 36](#). Notice the placement of the cells within each module, with `usbEngine0` highlighted in one color, and `usbEngine1` highlighted in a different color. Notice the logic is spread out across the device.
3. In the Device window, click the **Device View Options** button .
4. Uncheck the box next to **Cells** to disable the display of the placed cells in the Device window.

Various elements of the Device window can be hidden, to make it easier to see important details of the design or device.

Zoom into the Device window for a closer look, and pan around the window to examine the placement. With logic spread out across the device, the usbEngine blocks could benefit from floorplanning to improve timing.

5. Enable Cells in the **Device View Options** menu.
6. Click the **Device View Options** button  to hide the menu.
7. From the main toolbar, click the **Unhighlight All** button .

Visualizing the Connectivity

You can alleviate routing congestion and timing issues by floorplanning to prevent logic from being placed into critical or congested areas during implementation. Evaluating the design connectivity will help you determine which modules are suitable for floorplanning. For example, logic modules that connect to logic throughout the device may not be suitable for floorplanning, while tightly grouped and self-contained modules are suitable.

1. In the Device window, select the **Show I/O Nets** toolbar button .

The wires in the Device window in [Figure 37](#) show the connectivity between the I/O ports and placed instances on the device.

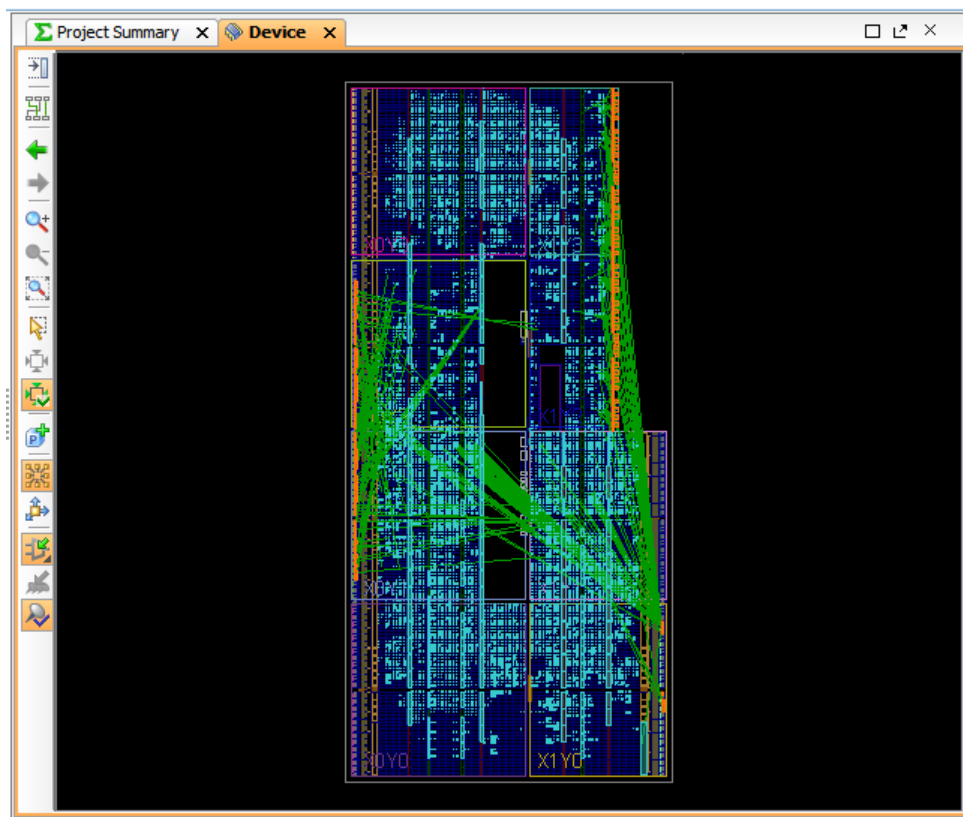



Figure 37: Show I/O Connectivity

2. Click to select one of the I/O Nets.
3. Inspect the net name in the Netlist window.
4. In the Netlist window, click the **Collapse All** button .
5. From the Netlist window, select **usbEngine0** and **usbEngine1**.
6. Right-click and select the **Show Connectivity** command from the popup menu, or press **Ctrl-T**.

The interface nets that connect `usbEngine0` and `usbEngine1` to the rest of the design are highlighted, as shown in the following figure. You can see that most of the connections go to the I/O Blocks on the left side of the device. This will help guide your placement during floorplanning.

You will floorplan the `usbEngine` blocks to keep the I/O logic near the I/O blocks, not specifically to improve timing. However, this could improve timing if the design was failing timing in some way.

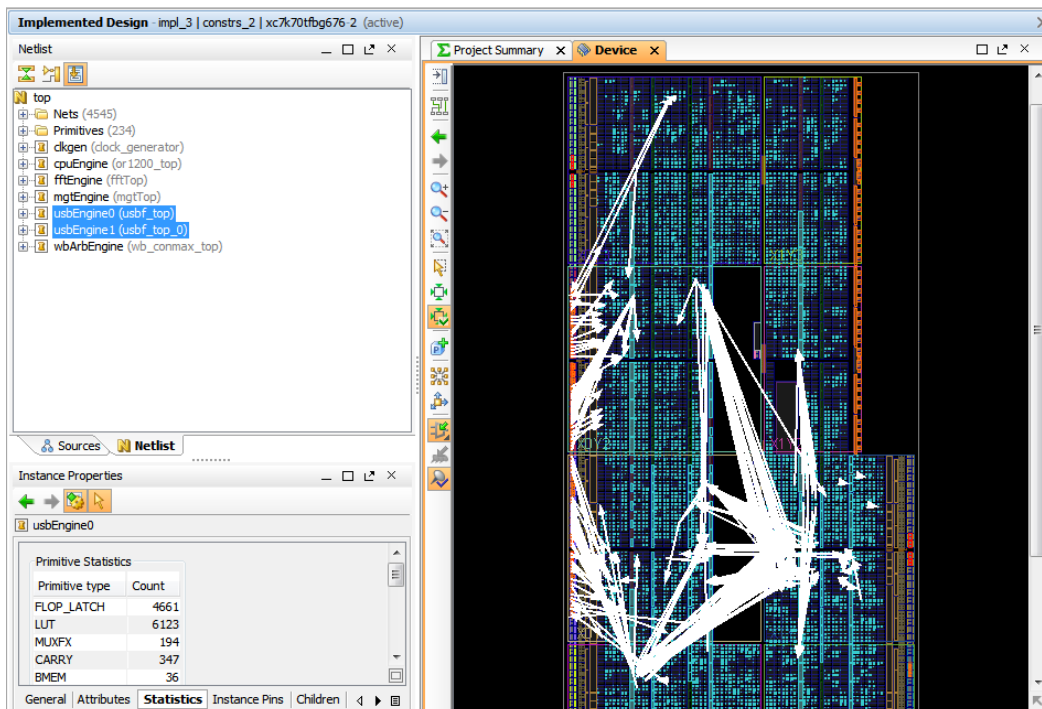


Figure 38: Show usbEngine Connectivity

7. Right-click and select **Show Connectivity** again to select all of the logic objects to which the interface nets connect.

You can use the Show Connectivity command to highlight or select a cone of logic from any source net or logic object.

8. Click the **Unselect All** button  or press **F12**.

Step 4: Floorplanning the Design

With an understanding of the design connectivity, and the problems of the prior placement, you can now begin floorplanning the design. The process of floorplanning begins by grouping some or all of the logic in the design into physical blocks, called Pblocks, and constraining that logic with common constraints. Grouping associated logic together into Pblocks helps you analyze the data flow of the design, limits interconnect lengths, prevents logic migration, and reduces timing delays.

Defining a Pblock results in an `create_pblock` Tcl command that is written to the target constraint file. The constraint defines the assigned logic, specified site ranges, and defined properties of the Pblock.

Automatically Creating Pblocks

1. Select Tools > Floorplanning > Auto-create Pblocks.

This opens the Auto-create Pblocks dialog box.

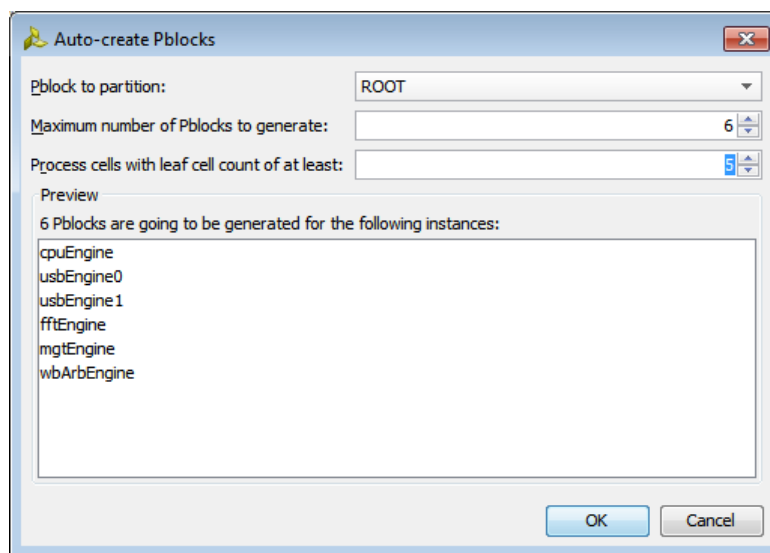


Figure 39: Auto-Create Pblocks



In the dialog box, review the options to define the maximum number of Pblocks to automatically create, and to limit the size of the modules to create Pblocks from.

2. Change the Maximum number of Pblocks to generate to 6.

If the design has more modules than the maximum number of Pblocks, the Vivado tool creates Pblocks starting with the largest modules first.

3. **Click OK** to create Pblocks for the listed modules.

4. **Select the Floorplanning layout** from the View Layout drop down menu to display the Physical Constraints window.

The newly created Pblocks, as well as the top-level ROOT, are listed in the Physical Constraints window, as shown in [Figure 40](#). In the Netlist view, the icon next to the six modules that have been defined as Pblocks has changed from  to , to illustrate this change.

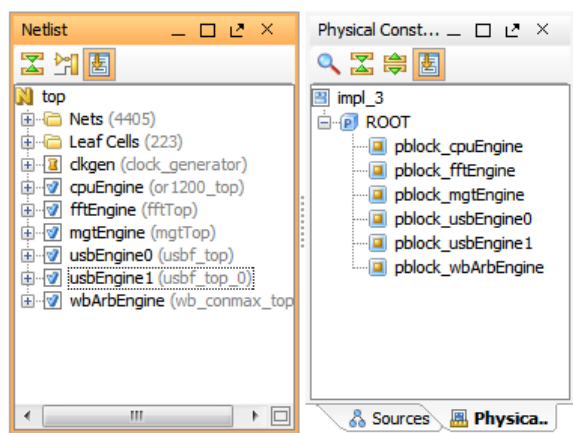


Figure 40: Physical Constraints window

Placing Pblocks

1. Select **Tools > Floorplanning > Place Pblocks**.

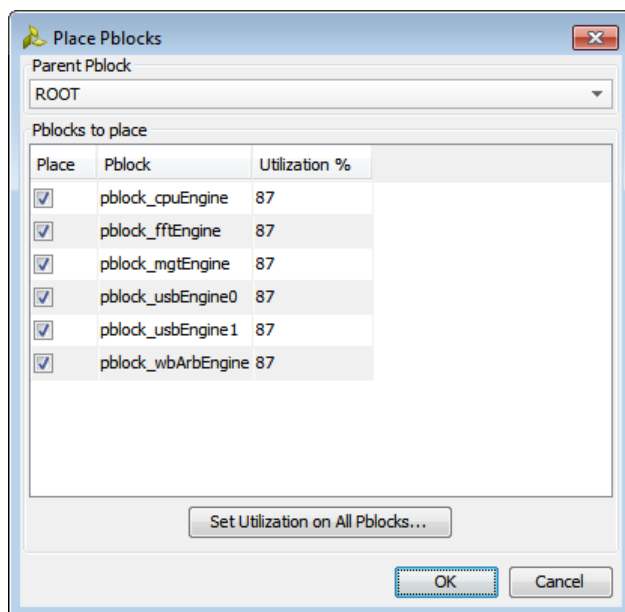


Figure 41: Place Pblocks

The Place Pblocks dialog box contains options to define which Pblocks to place, and what the target SLICE utilization for the Pblocks should be. The size of the Pblock created by the Vivado IDE depends on the Utilization % specified in the Place Pblocks dialog box.



TIP: The Utilization % is calculated using SLICE logic only. This provides a useful estimate for determining the size of a Pblock. However, Pblocks may need to be resized prior to implementation to insure sufficient resources are available for all logic types assigned to the Pblock.

2. Click **OK** to accept the **default values**, and place all the Pblocks.

The specific placement of the Pblocks is preliminary, and may be different each time the Place Pblocks command is run. You can arrange the Pblocks in the Device window to untangle the connectivity, as shown in [Figure 42](#), by selecting and moving them as described in [Moving Pblocks](#).

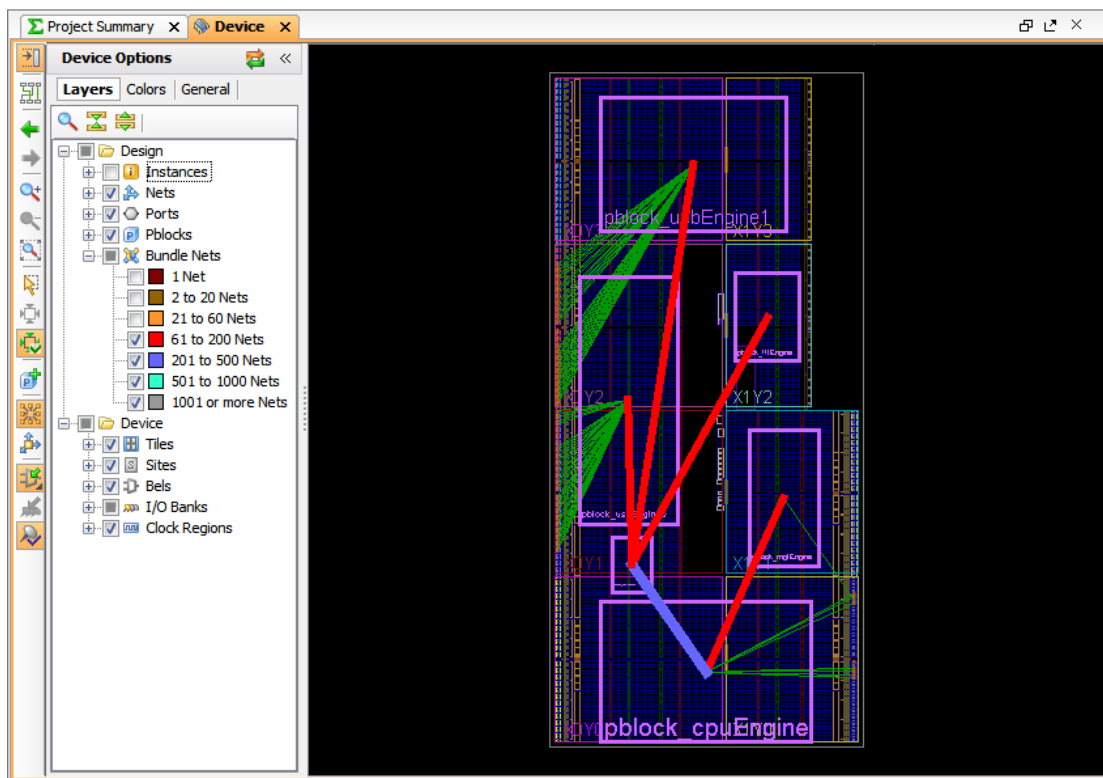




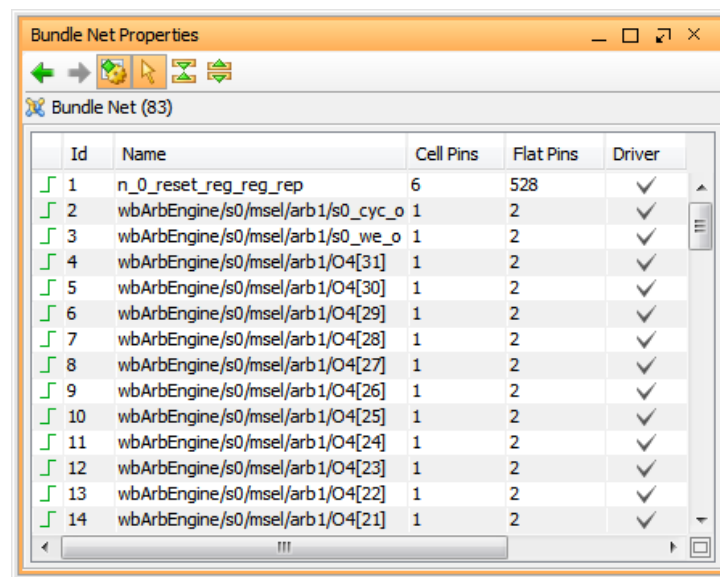
Figure 42: Pblock Data Flow with Bundled Nets

Viewing Pblock Connections

1. Select the **Show/Hide I/O Nets** toolbar button to enable the display of connections between the Pblocks and the I/O ports. 
2. Open the **Device View Options** menu from the toolbar, and enable the display of Bundle Nets under the Design heading. 

Bundle nets and I/O connections help you visualize the data flow of the design, as seen in [Figure 42](#). The preliminary floorplan provides early indications of data flow through the design, and highlights where potential routing congestion could occur. Bundles are color coded to show the number of nets connecting two Pblocks.

To see or change the current color definitions, use the **Tools > Options** command to open the Vivado Options dialog box. Select the Colors button on the left side of the dialog box, and select the Bundle Nets category on the right. You can also view the colors defined under the Device View Options menu in the Device window, but you cannot change the colors from that location.



| Id | Name | Cell Pins | Flat Pins | Driver |
|----|-------------------------------------|-----------|-----------|--------|
| 1 | n_0_reset_reg_reg_rep | 6 | 528 | ✓ |
| 2 | wbArbEngine/s0/msel/arb1/s0_cyc_o 1 | 2 | 2 | ✓ |
| 3 | wbArbEngine/s0/msel/arb1/s0_we_o 1 | 2 | 2 | ✓ |
| 4 | wbArbEngine/s0/msel/arb1/O4[31] | 1 | 2 | ✓ |
| 5 | wbArbEngine/s0/msel/arb1/O4[30] | 1 | 2 | ✓ |
| 6 | wbArbEngine/s0/msel/arb1/O4[29] | 1 | 2 | ✓ |
| 7 | wbArbEngine/s0/msel/arb1/O4[28] | 1 | 2 | ✓ |
| 8 | wbArbEngine/s0/msel/arb1/O4[27] | 1 | 2 | ✓ |
| 9 | wbArbEngine/s0/msel/arb1/O4[26] | 1 | 2 | ✓ |
| 10 | wbArbEngine/s0/msel/arb1/O4[25] | 1 | 2 | ✓ |
| 11 | wbArbEngine/s0/msel/arb1/O4[24] | 1 | 2 | ✓ |
| 12 | wbArbEngine/s0/msel/arb1/O4[23] | 1 | 2 | ✓ |
| 13 | wbArbEngine/s0/msel/arb1/O4[22] | 1 | 2 | ✓ |
| 14 | wbArbEngine/s0/msel/arb1/O4[21] | 1 | 2 | ✓ |

Figure 43: Bundle Net Properties

3. In the Device window, **select** a bundle net.

Examine the list of nets in the Bundle Net Properties window as shown in [Figure 43](#). These are the nets contained within the selected bundle, connecting two Pblocks.

Finding a Global Clock

In this step, you will use the Find command to locate and select the USB Global Clock (`usbClk`) to highlight the clocked Pblocks.

Effective floorplanning is often dependent on proper placement of the synchronous elements from different clock domains. You can highlight clock domains to visualize connectivity and ensure that Pblocks are properly placed to effectively utilize clock resources on the device. To learn more about clock resources and clock management, refer to the *Xilinx 7 Series FPGAs Clocking Resources User Guide* ([UG472](#)).

1. Select **Edit > Find**, or type Control-F.

This opens the Find dialog box to search through the design for specific types of objects.

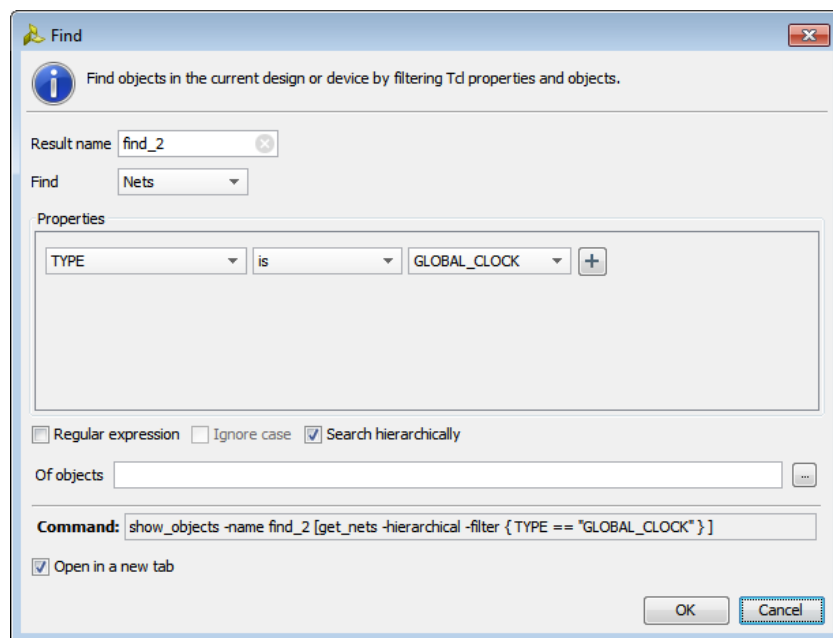


Figure 44: Find Global Clocks

2. In the Find dialog box, set the following options:

- Find: **Nets**
- Criteria: **Type**, **is**, **Global Clock**

3. Click **OK**.

The Find results window opens at the bottom of the Vivado IDE as shown in [Figure 45](#).

4. Click the **Name** column header once to sort the Find Results window by name.
5. In the Find Results window list, select **clkgen/usbClk**.

The selected net is highlighted in the Device window, connecting the two `usbEngine` Pblocks. This clock is limited to these two blocks.

| Id | Name | Cell Pins | Flat Pins | Driver | Route Status |
|----|---------------------|-----------|-----------|--------|-----------------------------|
| 4 | phyClk1 | 2 | 3,786 | | Partially routed (antennas) |
| 5 | usbClk | 19 | 1,481 | | Partially routed (antennas) |
| 6 | wbClk | 9 | 1,496 | | Partially routed (antennas) |
| 7 | clkgen/clkfbout_buf | 2 | 2 | ✓ | Fully routed |
| 8 | clkgen/cpuClk | 1 | 3,368 | ✓ | Partially routed (antennas) |
| 9 | clkgen/fftClk | 1 | 1,437 | ✓ | Partially routed (antennas) |
| 10 | clkgen/phyClk0 | 1 | 3,786 | ✓ | Partially routed (antennas) |
| 11 | clkgen/phyClk1 | 1 | 3,786 | ✓ | Partially routed (antennas) |
| 12 | clkgen/usbClk | 1 | 1,481 | ✓ | Partially routed (antennas) |
| 13 | clkgen/wbClk | 1 | 1,496 | ✓ | Partially routed (antennas) |

Figure 45: Find Global Clock

Moving Pblocks

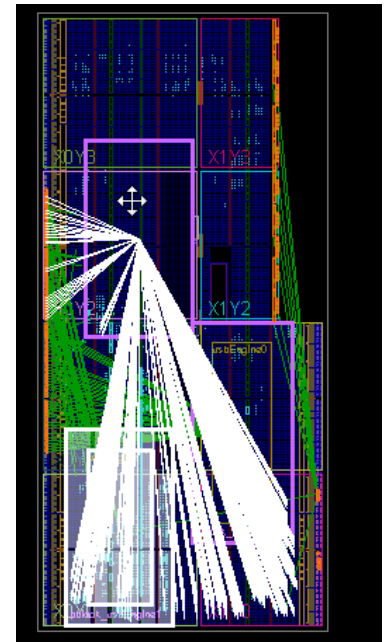
With the bundle nets and the I/O connections displayed, you may need to rearrange the preliminary Pblock placement to better visualize the data flow through the design as seen in [Figure 42](#). You can move the Pblocks in the Device window by clicking on a Pblock and dragging it to the desired location.

1. **Select a Pblock** in the Physical Constraints window, or select it directly in the Device window.
2. **Move** the Pblock by dragging it to a new location on the Device.

The placement you choose should seem appropriate based on the displayed connections between the Pblocks and to the I/O ports. To reduce delay, you should place the Pblock near the I/O ports it is connected to.

When you place the Pblock in its new location, the Move dialog box prompts you to select which resource types in the new location should be enabled for use by the Pblock.

3. Select **OK** to include all resource types.



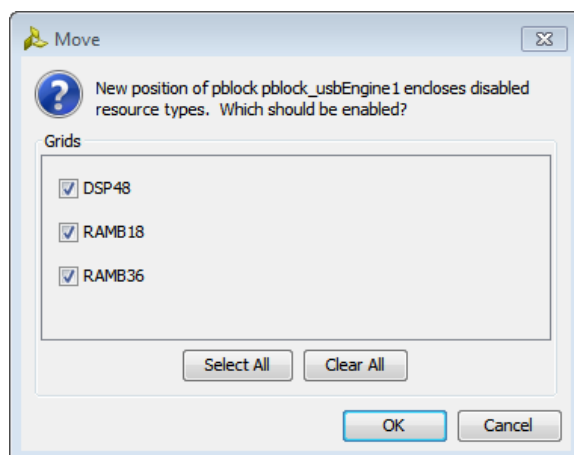


Figure 46: Move Pblock

When you move a Pblock, with logic elements that have been previously placed, the Vivado tool will open the Choose LOC Mode dialog box to prompt you to decide how to handle existing cell placement, or LOC constraints, as shown in [Figure 47](#).

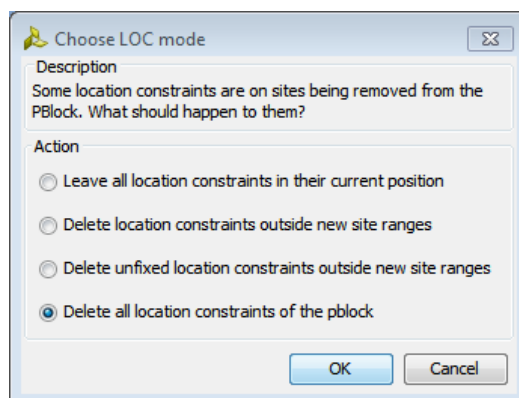


Figure 47: Choose LOC Mode

The choices are:³

- Leave cells where they are currently placed.
- Unplace cells that fall outside the new Pblock area.
- Unplace only unfixed cells that fall outside the new Pblock area.
- Unplace all cells, fixed and unfixed, in the Pblock.

³ The choices offered in the Choose LOC Mode dialog box depends on the contents of the Pblock being moved, and whether there are fixed and unfixed cells inside the block.

IMPORTANT: Both fixed and unfixed cells are placed.

Fixed and Unfixed are terms that apply to placed cells, and describe the way the Vivado tool views placed cells in the design.



- Fixed cells have been placed by you, or another designer, or have been imported from a constraints file. These placed cells are therefore considered fixed by the Vivado Design Suite, and will not be moved unless directed to do so.
- Unfixed cells have been placed by the Vivado tool in implementation, during the place_design command, or one of optimization commands. These placed cells are considered unfixed, or loosely placed, by the Vivado Design Suite, and can be moved as needed in design iterations.

4. Select Delete all location constraints of the pblock and click OK.

This will unplace all cells assigned to the Pblock, clearing the LOC constraint from the cells, allowing them to be relocated to the new area covered by the Pblock.

Clearing Placement LOCs

An alternative to clearing or keeping the current placement for each Pblock as you move it, is to clear the placement (LOCs) for the whole design. This lets you move Pblocks without having to respond to the Choose Loc Mode dialog box each time.

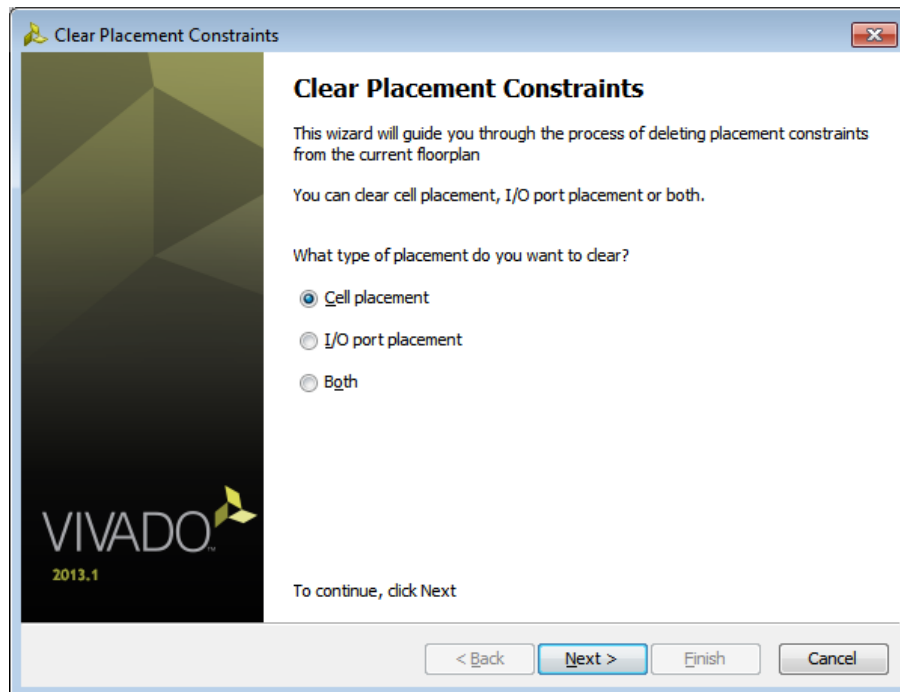


Figure 48: Clear Placement Constraints

1. Use the **Tools > Floorplanning > Clear Placement** command to clear the current placement for the whole design.

The Clear Placement Constraints dialog box lets you clear placement for either the cells or the I/O ports in the design. Or clear placement for both cells and ports.

2. Select **Cell placement** and click **Next**.

The Unplace Cells dialog box opens.

3. Select **Unplace all cells**, and click **Next**.

The Cell Types to Unplace dialog box opens as shown in [Figure 49](#).

4. **Click Next** to accept the default, and unplace everything but the I/O and the clock logic.

The Clear Placement Summary dialog box shows what instances will be unplaced.

5. **Click Finish** to clear the specified placement LOCs from the design.

With the placement cleared from the design, you can continue arranging the Pblocks as seen in [Figure 42](#), or as you prefer.

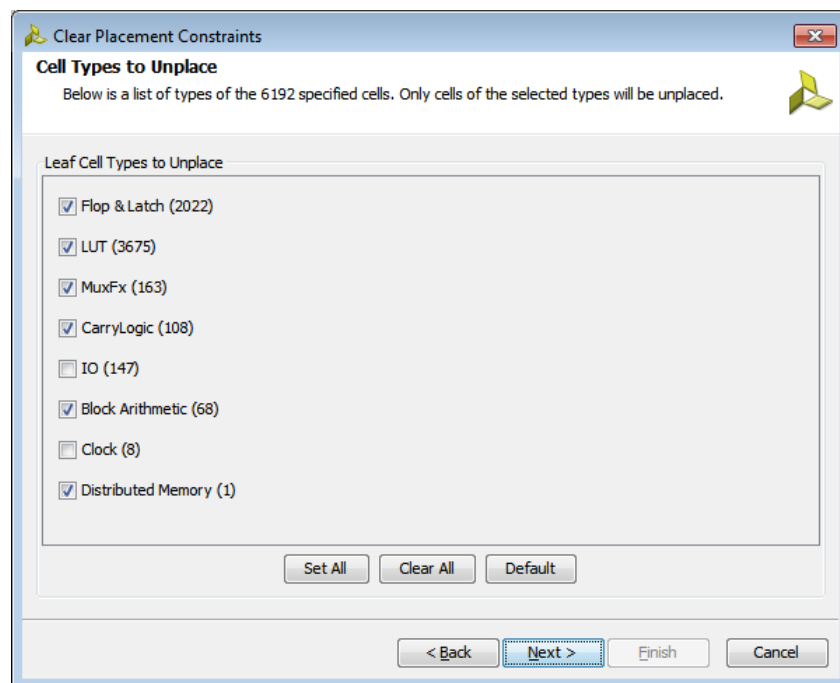


Figure 49: Clear Placement Types

Resizing Pblocks

At this point, to better focus on the two `usbEngine` blocks, select and clear the other Pblocks in the Physical Constraints window.

1. In the **Physical Constraints** window, **select** `pblock_cpuEngine`, `pblock_fftEngine`, `pblock_mgtEngine`, and `pblock_wbArbEngine`.
2. **Right click** and select the **Delete** command.

This deletes the selected Pblocks, leaving only the `usbEngine0` and `usbEngine1` Pblocks. The other Pblocks helped you understand the data flow through the different modules of the design, but are not needed for the final floorplan.

With the `usbEngine` blocks placed approximately where you would like them, you must now resize the Pblocks to include the required resources.

3. **Select a Pblock**, and click the **Statistics** tab in the Pblock Properties window.

The Statistics tab of the Pblock Properties window shows the device resources available in the area covered by the Pblock; shows the amount of resources required by the logic assigned to the Pblock; and shows the current utilization %.

The red values, as shown in [Figure 50](#) indicate insufficient resources available in the Pblock area to meet the requirements of the design logic assigned to the Pblock. In this case, the Pblock will need to be resized to increase its resources.

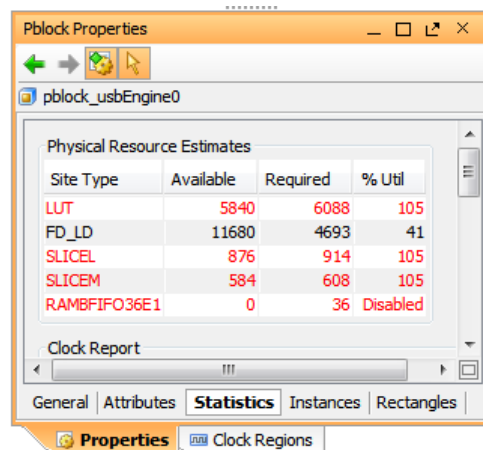


Figure 50: Pblock Properties - Statistics

4. Select **pblock_usbEngine0** in the Device window.
5. Right click and select the **Set Pblock Size** command from the popup menu.
6. **Draw a rectangle** to cover Clock Region X0Y1, on the left side the Device window.

If you size the Pblock to completely cover the clock region borders, you'll be prompted to confirm the clock region as the Pblock range. If the Pblock does not encompass the whole clock region, you'll be prompted to accept the logic types contained in the Pblock area.

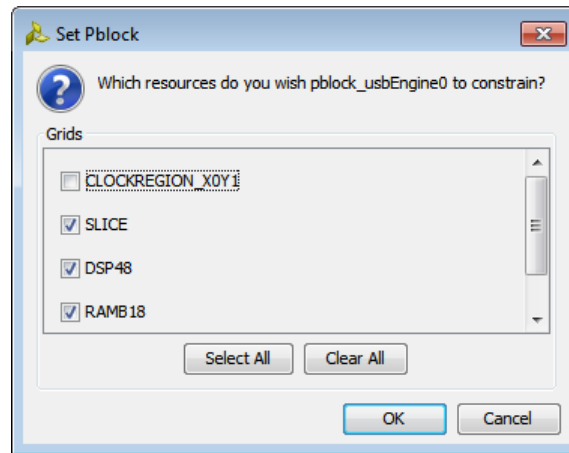


Figure 51: Set Pblock Size

7. **Click OK** to accept either the clock region, or the included logic.

At this time you can review the Statistics of the Pblock Properties window, and see that with only a single clock region `pblock_usbEngine0` is not large enough to include the required logic resources, as shown in [Figure 52](#).

8. **Scroll through** the **Pblock Properties** window to examine the different types of information presented.

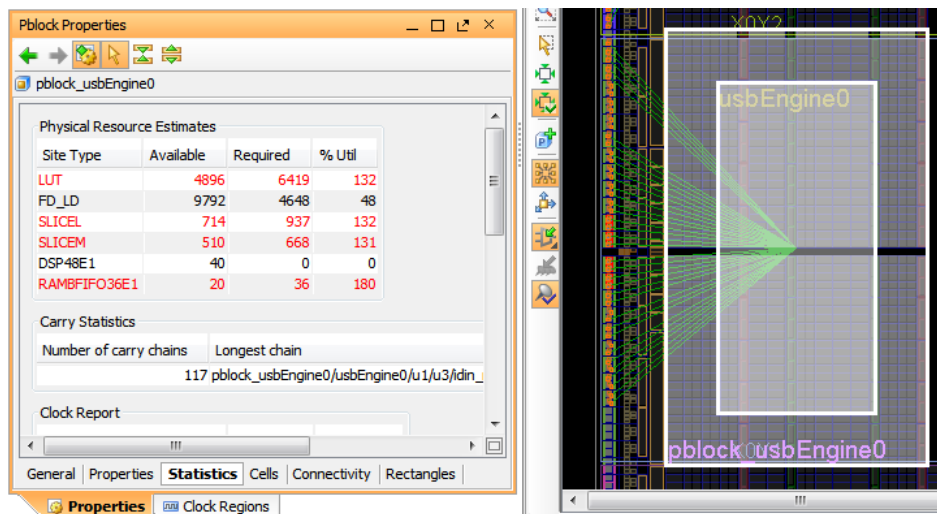


Figure 52: usbEngine0 Pblock Properties

9. **Click on** the bottom of the Pblock **and stretch** it all way to the bottom edge of the device and click **OK** to accept the Clock Region as the Pblock range.

Expand the Pblock rectangle to cover Clock Region X0Y0 and X0Y1 in the lower left of the Device window, as shown in [Figure 53](#). You will be prompted to accept the Clock Regions and not the individual logic site types.

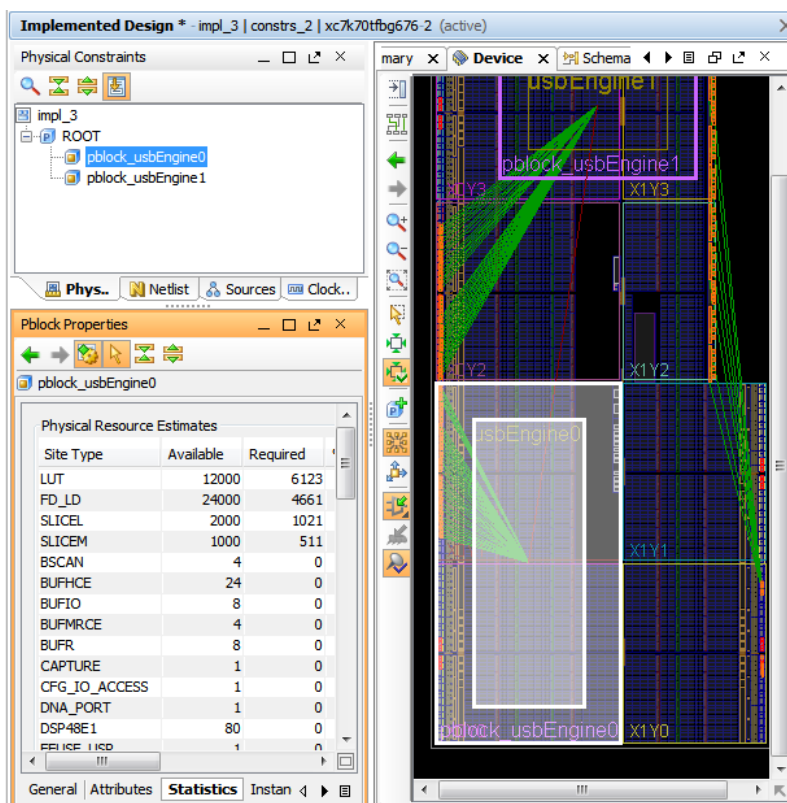


Figure 53: usbEngine0 Expanded

10. Select **pblock_usbEngine1** and resize the Pblock to cover both Clock Region X0Y2 and X0Y3 in the upper left of the Device window.

The final Pblock placement is shown in [Figure 54](#).

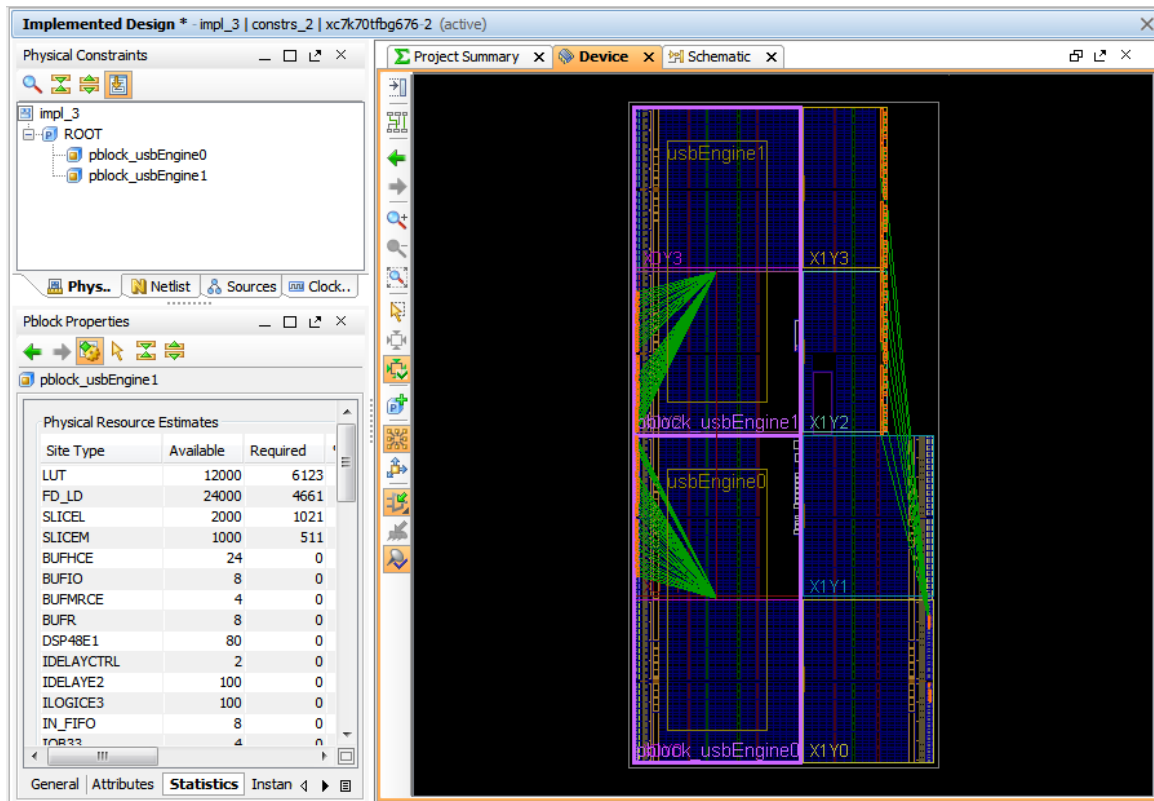


Figure 54: usbEngine0 and usbEngine1 Pblocks Placed

With new constraints created for the `usbEngine0` and `usbEngine1` Pblock placement, you must save the design constraints. It is a good idea to save the constraints into a new constraints set so that you can keep track of different design strategies. This allows you to revert to an older design strategy if needed, or develop and implement different strategies for a single design.

11. Select the **File > Save Constraints As** command.

This opens the Save Constraints As dialog box as shown in [Figure 55](#).

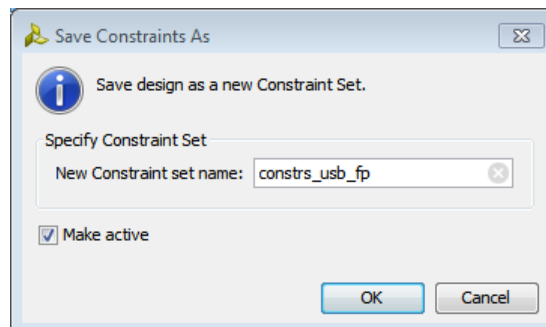


Figure 55: Saving a new Constraint Set

12. **Save** the new constraint set as **constrs_usb_fp**, for the usbEngine floorplan.

13. Make sure the **Make active** checkbox is selected, and click **OK**.

A new constraint set, called `constrs_usb_fp`, is created to store the floorplan constraints you have created here. The `top_full.xdc` file is copied to the new constraint set, and the constraints are updated as needed to reflect the current state of the design.

14. In the Sources window, expand the **constrs_usb_fp** constraint set, and double-click the **top_full.xdc** file to open it in the Text Editor.

Scroll through the open constraint file to examine the constraints. Toward the bottom of the file you will see the new Pblock constraints.

When you save the constraint set, the Synthesized Netlist and Implemented Design may go out of date, as reflected in the project status bar. In some cases this may be the result of small changes that you know do not actually affect synthesis or implementation. When this happens, you can force the design into an up-to-date state, as shown in [Figure 56](#).

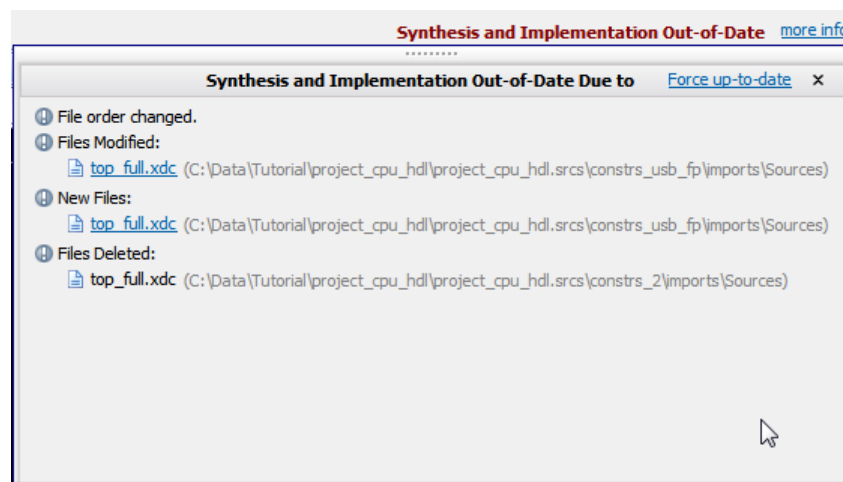


Figure 56: Force Up-to-date

In this case, the changes you made do not affect the Synthesized Netlist, so you will not need to rerun synthesis. However, clearing the placement and floorplanning the usbEngine blocks means that the Implemented Design is out-of-date, and implementation will need to be run again. You can force the design into an up-to-date state for synthesis, and then rerun implementation.

15. Click the **more info** link in the project status bar.

This opens the Synthesis and Implementation Out-of-Date dialog box which displays some possible reasons why the design may be out-of-date.

16. Click the **Force up-to-date** link in the upper right of the dialog box.

The project status bar is updated to reflect the up-to-date condition for the design.

Step 5: Running Implementation with the Floorplan

With the floorplan of the `usbEngine` blocks completed, you are now ready to rerun implementation.

1. Select the **Design Runs** window to make it active.
2. Right click and select the **Create Runs** command from the popup menu.
3. Select **Implementation** as the type of runs to create, and click **Next**.

This opens the Configure Implementation Runs dialog box.

4. Select the following options, and click Next:
 - **impl_usb_fp** for the Name,
 - **synth_1** for the Synth Name,
 - **constrs_usb_fp** for Constraints Set,
 - **xc7k70tfbg676-2** for Part,
 - **Flow_RuntimeOptimized** for Strategy.
 - **Make Active**

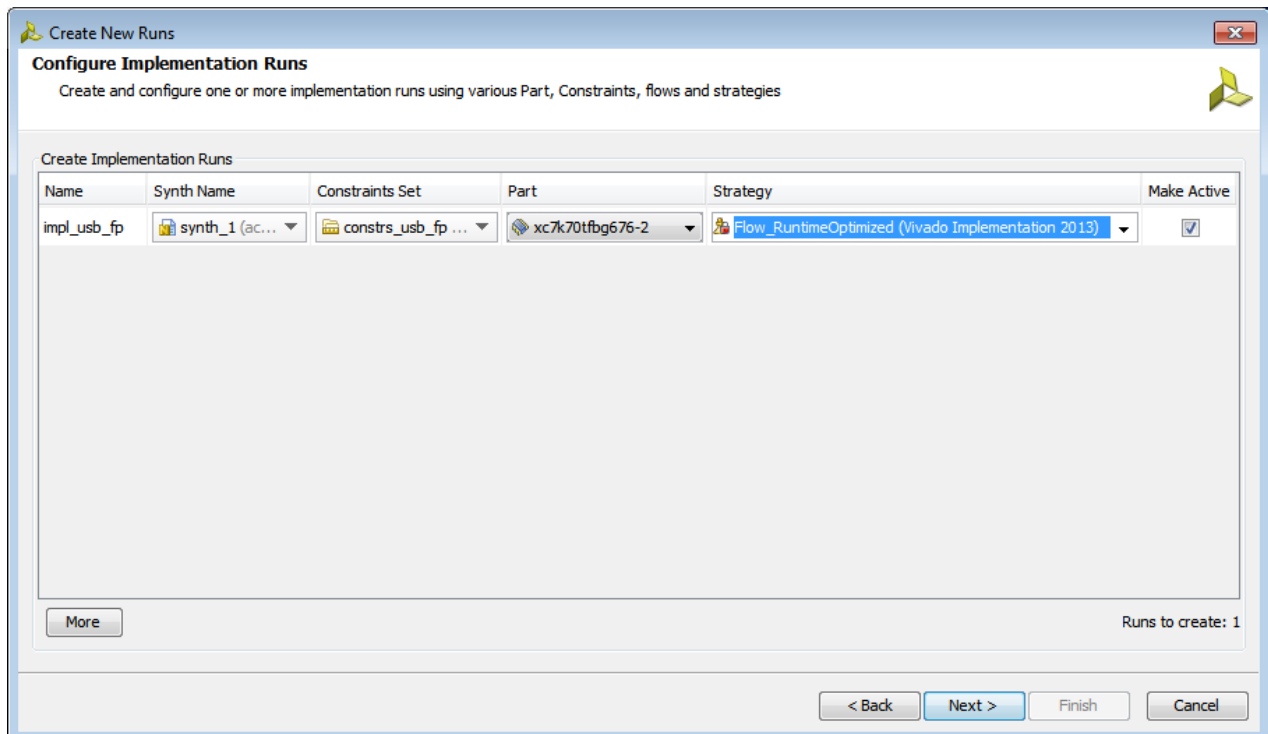


Figure 57: Floorplan Implementation Run

5. In the Launch Options dialog box, select **Do not launch now** and click **Next**.

6. On the Create New Runs Summary, check that the settings are correct, and click **Finish**.

A new implementation run is created with the specified properties.

7. From the Flow Navigator menu, select **Run Implementation**.

The implementation run is launched with the usbEngine blocks placed according to your floorplan. The Implementation Completed dialog box displays when the run is finished.

8. Select Open **Implemented Design**, and click OK.

The design has been implemented with the floorplan you created for the usbEngine blocks. You can see this by highlighting the blocks as you did earlier in this tutorial.

9. In the Netlist window, select **usbEngine0** and **usbEngine1**.

10. Right click and select **Highlight Leaf Cells > Cycle Colors** from the popup menu.

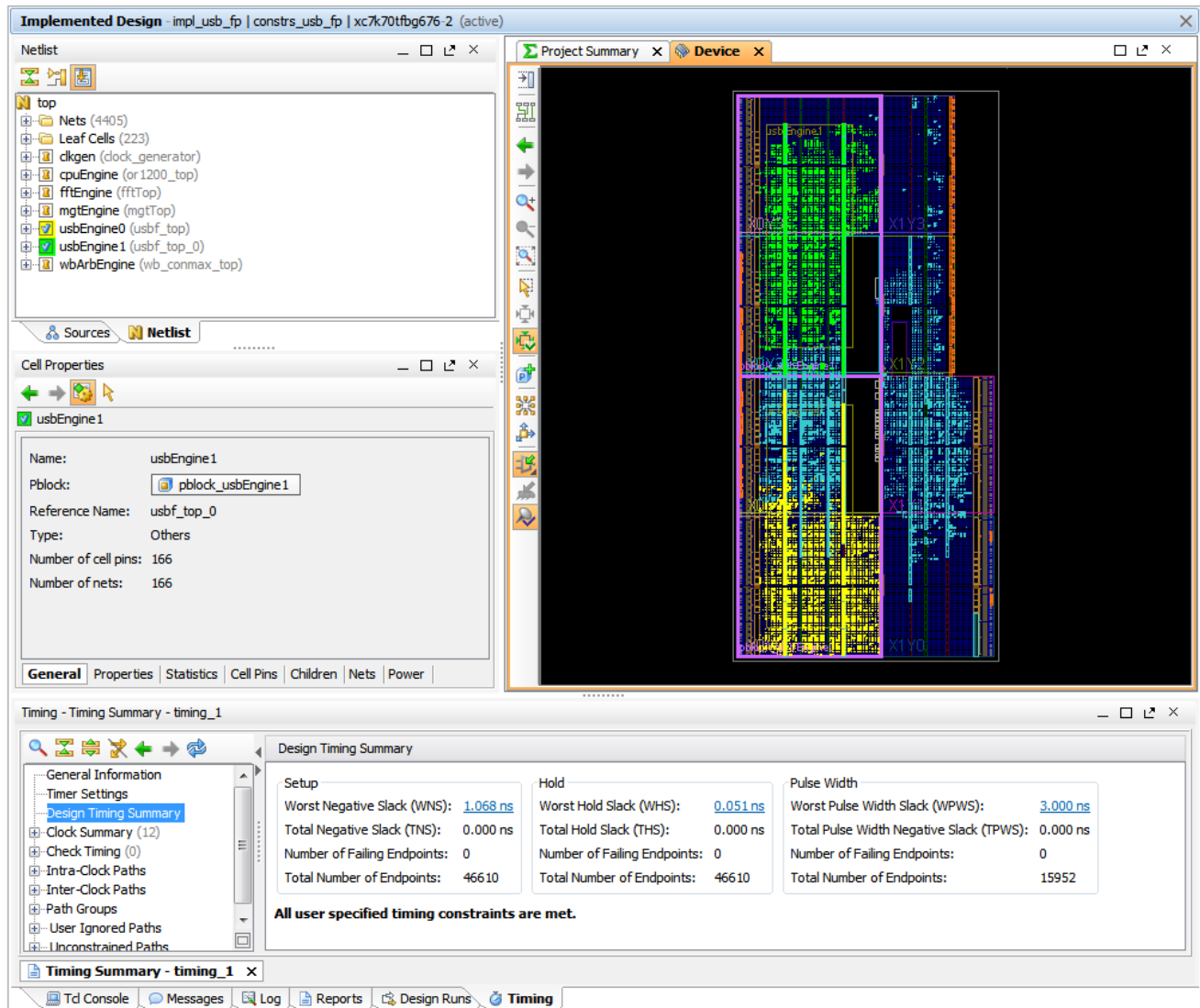


Figure 58: Implemented Floorplan

11. From the Flow Navigator select **Report Timing Summary** and click **OK** to run Vivado timing analysis with the default settings.

The Timing Summary Report now shows that all timing has been met as a result of the improved placement of your floorplan, and the improved routing results.

Conclusion

In this tutorial, you used the Vivado IDE to explore and analyze the synthesized design and targeted device prior to running the implementation tools. This enabled you to find potential design issues and errors early in the design cycle, rather than discovering issues during implementation. In addition, you used the graphical presentation of design resource estimates, design rule violations, timing estimation, constraints, and connectivity to help you understand your design and any areas with potential issues.

After running the design through the synthesis and implementation tools, you:

- Viewed implementation results and examined timing results.
- Analyzed critical path objects in the schematic, and selected the parent modules of those path objects.
- Highlighted module placement and displayed the connectivity of the modules using the Show Connectivity command.
- Analyzed placement and routing of critical timing paths.
- Floorplanned possible placements based on the hierarchy of the design.
- Implemented the floorplanned design