Vivado Design Suite Tutorial

Design Flows Overview

UG888 (v2015.4) November 18, 2015

XILINX
ALL PROGRAMMABLE™

Revision History

The following table shows the revision history for this document.

Date	Version	Changes
11/18/2015	2015.4	Validated with 2015.4 release. Added cross-reference links to additional documents where relevant.
09/30/2015	2015.3	Validated with 2015.3 release and updated relevant figures. Updated tcl command `-min_fanout` from code snippet in Step 3: Synthesizing the Design to `-fanout_greater_than`.
06/24/2015	2015.2	Validated with 2015.2 release. Edited Tcl commands in Step 3: Synthesizing the Design to allow for copying commands into the Tcl console.
04/01/2015	2015.1	Initial 2015.1, validated with release.

Table of Contents

Revision History 2

Vivado Design Flows Overview 4
Introduction 4

Lab 1: Using the Non-Project Design Flow 8
Introduction 8
Step 1: Examine the Example Script 8
Step 2: Starting Vivado with the Example Design 8
Step 3: Synthesizing the Design 10
Step 4: Launching the Vivado IDE 10
Step 5: Defining Timing Constraints and I/O Planning 12
Step 6: Exporting the Modified Constraints 15
Step 7: Implementing the Design 16
Step 8: Opening a Design Checkpoint 17
Step 9: Analyzing Implementation Results 17
Step 10: Exiting the Vivado Tool 20

Lab 2: Using the Project Design Flow 21
Introduction 21
Step 1: Creating a Project 21
Step 2: Using the Sources Window and Text Editor 27
Step 3: Elaborating the RTL Design 31
Step 4: Using the IP Catalog 32
Step 5: Running Behavioral Simulation 33
Step 6: Reviewing Design Run Settings 34
Step 7: Synthesizing and Implementing the Design 37
Step 8: Analyzing the Synthesized Design 38
Step 9: Analyzing the Implemented Design 41
Step 10: Generating a Bitstream file 43
Conclusion 45

Legal Notices 46
Please Read: Important Legal Notices 46

IMPORTANT: *This tutorial requires the use of the Kintex®-7 family of devices. You will need to update your Vivado tools installation if you do not have this device family installed. Refer to the Vivado Design Suite User Guide: Release Notes, Installation, and Licensing (UG973) for more information on Adding Design Tools or Devices.*

Introduction

This tutorial introduces the use models and design flows recommended for use with the Xilinx® Vivado® Integrated Design Environment (IDE). This tutorial describes the basic steps involved in taking a small example design from RTL to bitstream, using two different design flows as explained below. Both flows can take advantage of the Vivado IDE, or be run through batch Tcl scripts. The Vivado Tcl API provides considerable flexibility and power to help set up and run your designs, as well as perform analysis and debug.

VIDEO: *You can also learn more about the Vivado Design Suite design flows by viewing the quick take video at Vivado Design Flows.*

TRAINING: *Xilinx provides training courses that can help you learn more about the concepts presented in this document. Use these links to explore related courses:*

- *Vivado Introductory Workshop*
- *Vivado Design Suite Tool Flow*
- *Essentials of FPGA Design*

Working in Project Mode and Non-Project Mode

Some users prefer the design tool for automatically managing their design flow process and design data, while others prefer to manage sources and process themselves. The Vivado Design Suite uses a project file (`.xpr`) and directory structure to manage the design source files, store the results of different synthesis and implementation runs, and track the project status through the design flow. This automated management of the design data, process and status requires a project infrastructure. For this reason, Xilinx refers to this flow as the Project Mode.

Other users prefer to run the FPGA design process more like a source file compilation, to simply compile the sources, implement the design, and report the results. This compilation style flow is

referred to as the Non-Project mode. The Vivado Design Suite easily accommodates both of these use models.

Both of these flows utilize a project structure to compile and manage the design. The main distinctions are that Non-Project mode processes the entire design in memory. No files are written to disk. While Project mode creates and maintains a project directory structure on disk to manage design sources, results, and project settings and status.

The following provides a brief overview of Project mode and Non-Project mode. For a more complete description of these design modes, and the features and benefits of each, refer to this link in the *Vivado Design Suite User Guide: Design Flows Overview* (UG892).

Non-Project Mode

This use model is for script-based users who do not want Vivado tools to manage their design data or track their design state. The Vivado tools simply read the various source files and compile the design through the entire flow in-memory. At any stage of the implementation process, you can generate a variety of reports, run design rule checks (DRCs), and write design checkpoints. Throughout the entire flow, you can open the design in-memory, or any saved design checkpoint, in the Vivado IDE for design analysis or netlist/constraint modification. Source files, however, are not available for modification in the IDE when running the Non-Project mode. It is also important to note that this mode does not enable project-based features such as source file and run management, cross-probing back to source files, design state reporting, etc. Essentially, each time a source file is updated on the disk; you must know about it and reload the design.

There are no default reports or intermediate files created within the Non-Project mode. You must direct the creation of reports or design checkpoints with Tcl commands.

Project Mode

This use model is for users who want the Vivado tools to manage the entire design process, including features like source file, constraint and results management, integrated IP design, and cross probing back to sources. In Project mode, the Vivado tools create a directory to manage the design source files, IP data, synthesis and implementation run results and related reports. The Vivado Design Suite manages and reports the status of the source files, configuration, and the state of the design. You can create and configure multiple runs to explore constraint or command options. In the Vivado IDE, you can cross-probe implementation results back to the RTL source files. You can also script the entire flow with Tcl commands, and open Vivado IDE as needed.

Using Tcl Commands

The Tcl commands and scripting approach vary depending on the design flow used. When using the Non-Project mode, the source files are loaded using `read_verilog`, `read_vhdl`, `read_edif`, `read_ip`, and `read_xdc` commands. The Vivado Design Suite creates an in-memory design database to pass to synthesis, simulation, and implementation. When using Project mode, you can use the `create_project`, `add_files`, `import_files`, and `add_directories` commands to create the

project infrastructure needed to manage source files and track design status. Replace the individual "atomic" commands, `synth_design`, `opt_design`, `place_design`, `route_design`, and `write_bitstream` in the Batch flow, with an all-inclusive command called `launch_runs`. The `launch_runs` command groups the atomic commands together with other commands to generate default reports and track the run status. The resulting Tcl run scripts for the Project mode are different from the Non-Project mode. This tutorial covers the Project mode and Non-Project mode, as well as the Vivado IDE.

Many of the analysis features discussed in this tutorial are covered in more detail in other tutorials. Not every command or command option is represented here. To view the entire list of Tcl commands provided in the tools, consult the *Vivado Design Suite Tcl Command Reference Guide* (UG835).

This tutorial contains two labs that can be performed independently.

Lab 1: Using the Non-Project Design Flow

- Walk through a sample run script to implement the `bft` design.
- View various reports at each step.
- Review the `vivado.log` file.
- Write design checkpoints.
- Open the Vivado IDE after synthesis to review timing constraint definition and I/O planning and demonstrate methods to update constraints.
- Open the implemented Design Checkpoint to analyze timing, power, utilization and routing.

Lab 2: Using the Project Based Design Flow

- Create a new project.
- Walk through implementing the bft design using the Vivado IDE.
- View various reports at each step.
- Open the synthesized design and review timing constraint definition, I/O planning and design analysis.
- Open the implemented design to analyze timing, power, resource utilization, routing, and cross-probing.

Tutorial Design Description

The sample design used throughout this tutorial consists of a small design called `bft`. There are several VHDL and Verilog source files in the `bft` design, as well as a XDC constraints file.

The design targets an xc7k70T device. A small design is used to allow the tutorial to be run with minimal hardware requirements and to enable timely completion of the tutorial, as well as to minimize the data size.

Hardware and Software Requirements

This tutorial requires that the 2015.3 Vivado Design Suite software release or later is installed. The following partial list describes the operating systems that the Vivado Design Suite supports on x86 and x86-64 processor architectures:

See the *Vivado Design Suite User Guide: Release Notes, Installation, and Licensing* (UG973) for a complete list and description of the system and software requirements.

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.

Introduction

This lab focuses on Non-Project mode and the associated Tcl commands.

Step 1: Examine the Example Script

1. Open the example script:
   `<Extract_Dir>/Vivado_Tutorial/create_bft_kintex7_batch.tcl`, in a text editor and review the different steps.

   ```
   STEP#0: Define output directory location.
   STEP#1: Setup design sources and constraints.
   STEP#2: Run synthesis, report utilization and timing estimates, write checkpoint
   design.
   STEP#3: Run placement and logic optimization, report utilization and timing
   estimates, write checkpoint design.
   STEP#4: Run router, report actual utilization and timing, write checkpoint
   design, run drc, write verilog and xdc out.
   STEP#5: Generate a bitstream.
   ```

   Notice that many of the Tcl commands are commented out. You will run them manually, one at a time.

2. Leave the example script open, as you will copy and paste commands from it later in this tutorial.

Step 2: Starting Vivado with the Example Design

On Linux

1. Change to the directory where the lab materials are stored:

   `cd <Extract_Dir>/Vivado_Tutorial`

2. Launch the Vivado Design Suite Tcl shell, and source a Tcl script to create the tutorial design:

   `vivado -mode tcl -source create_bft_kintex7_batch.tcl`

On Windows

1. Launch the Vivado Design Suite Tcl shell:

   Start > All Programs > Xilinx Design Tools > Vivado 2015.x > Vivado 2015.x Tcl Shell

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

2. In the Tcl shell, change to the directory where the lab materials are stored:

   ```
   Vivado% cd <Extract_Dir>/Vivado_Tutorial
   ```

3. Source a Tcl script to create the tutorial design:

   ```
   Vivado% source create_bft_kintex7_batch.tcl
   ```

   After the sourced script has completed, the Vivado Design Suite Tcl shell, hereafter called the Tcl shell, displays the Tcl prompt: **`Vivado%`**

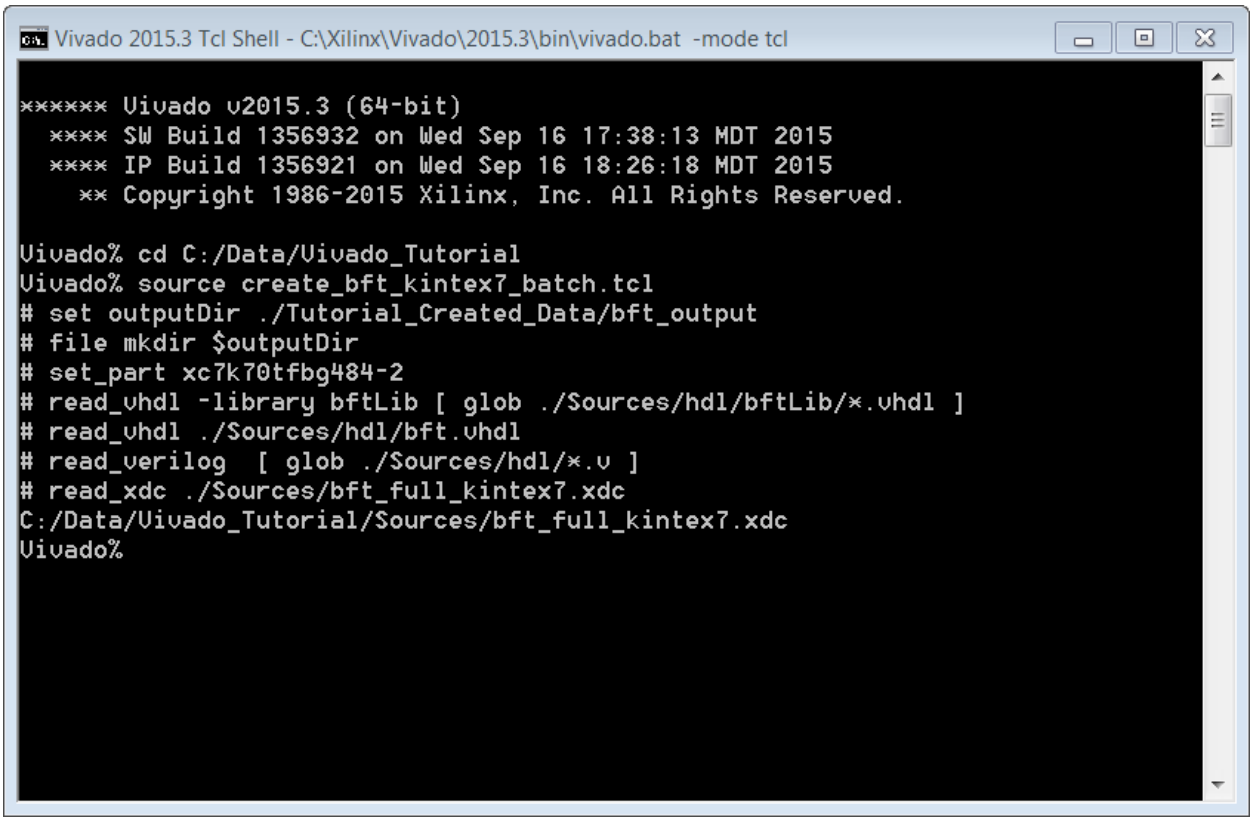

Figure 1: Start Vivado and Source Tcl Script

You can enter additional Tcl commands from the Tcl prompt.

Step 3: Synthesizing the Design

1. Copy and paste the `synth_design` command from the `create_bft_kintex7_batch.tcl` script into the Tcl shell and wait for synthesis to complete. You can paste into the Tcl shell using the popup menu, by clicking the right mouse button.

   ```
   synth_design -top bft
   ```

   Note: *The command in the example script is a comment. Do not copy the leading '**#**' character, or your command will also be interpreted as a comment.*

2. Examine the synthesis report as it scrolls by.

3. When the Vivado Tcl prompt has returned, copy and paste the `write_checkpoint`, `report_timing_summary`, `report_power`, `report_clock_interaction`, and `report_high_fanout_nets` commands that follow synthesis.

   ```
   write_checkpoint -force $outputDir/post_synth
   report_timing_summary -file $outputDir/post_synth_timing_summary.rpt
   report_power -file $outputDir/post_synth_power.rpt
   report_clock_interaction -delay_type min_max -file \
   $outputDir/post_synth_clock_interaction.rpt
   report_high_fanout_nets -fanout_greater_than 200 -max_nets 50 -file \
   $outputDir/post_synth_high_fanout_nets.rpt
   ```

4. Open another window to look at the files created in the output directory. On Windows, it may be easier to use the file browser.

   `<Extract_Dir>/Vivado_Tutorial/Tutorial_Created_Data/bft_output`

5. Use a text editor to open the various report (`*.rpt`) files that were created.

Step 4: Launching the Vivado IDE

Even though a Vivado project has not been created on disk, the in memory design is available in the tool, so from the Tcl shell you can open the Vivado IDE to view the design.

Non-Project mode enables the use of the Vivado IDE at various stages of the design process. The current netlist and constraints are loaded into memory in the IDE, enabling analysis and modification. Any changes to the logic or the constraints are live in memory and are passed to the downstream tools. This is quite a different concept than with the ISE tools that require saving and reloading files.

Open the IDE using the `start_gui` command.

```
Vivado% start_gui
```

The Vivado IDE provides design visualization and exploration capabilities for your use. From the Vivado IDE, you can perform further analysis and constraint manipulation on the design.

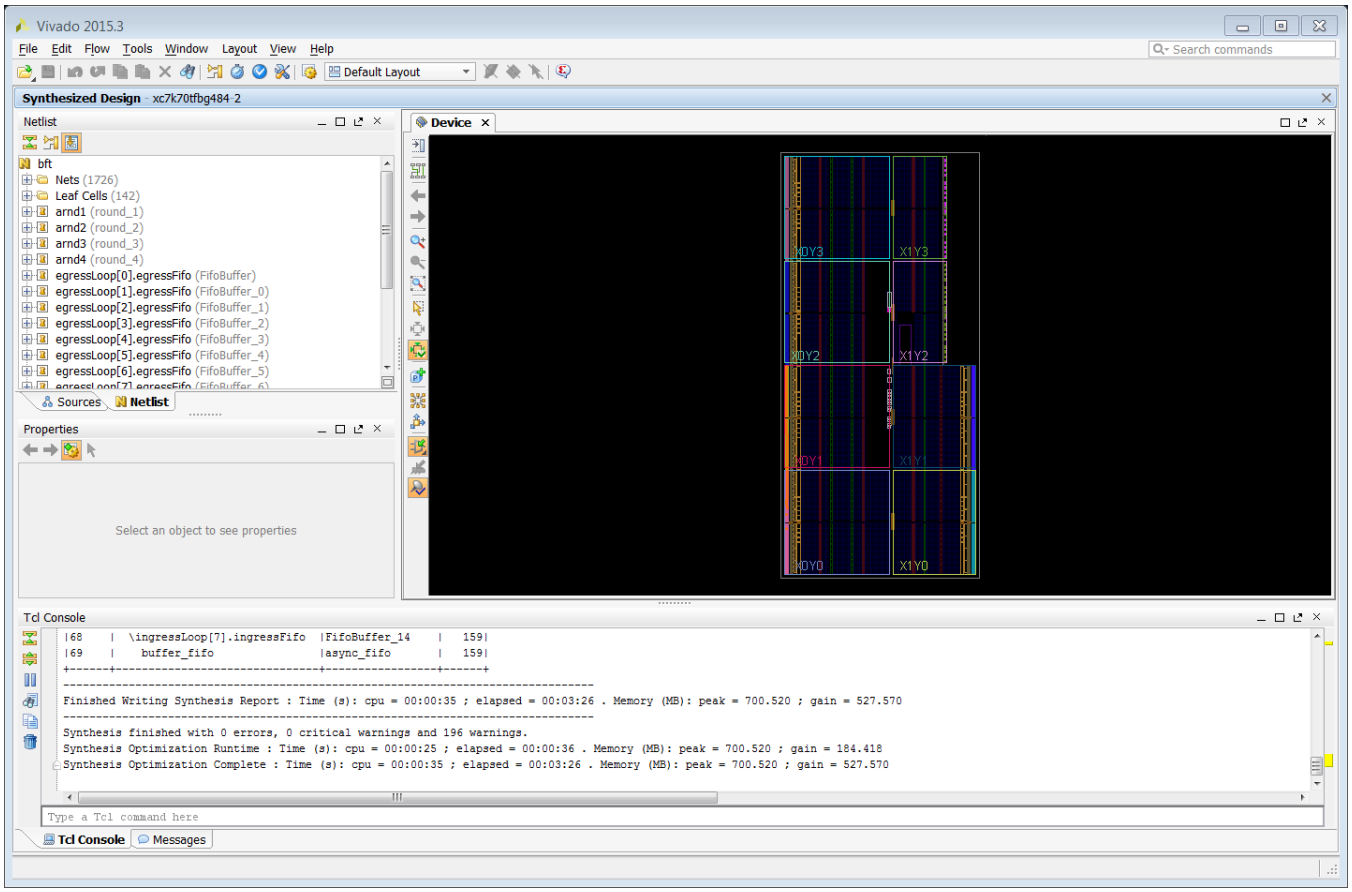

Figure 2: Vivado IDE - Non-Project Mode

TIP: *To stop the GUI and return to the Vivado Design Suite Tcl shell, use the* `stop_gui` *command. If you use the* ***File > Exit*** *command from the Vivado IDE, you will completely exit the Vivado tool.*

Since the design does not have a project in Non-Project mode, the Vivado IDE does not enable source file or run management. You are effectively analyzing the current in memory design. The Vivado Flow Navigator and other project based commands are also not available in Non-Project mode.

Step 5: Defining Timing Constraints and I/O Planning

You must often define timing and physical constraints for the design prior to implementation. The Vivado tools let you load constraints from constraints file(s), or enter constraints interactively using the IDE.

Defining Timing Constraints

1. Open the Timing Constraints window: **Window > Timing Constraints**, as shown in the following figure:

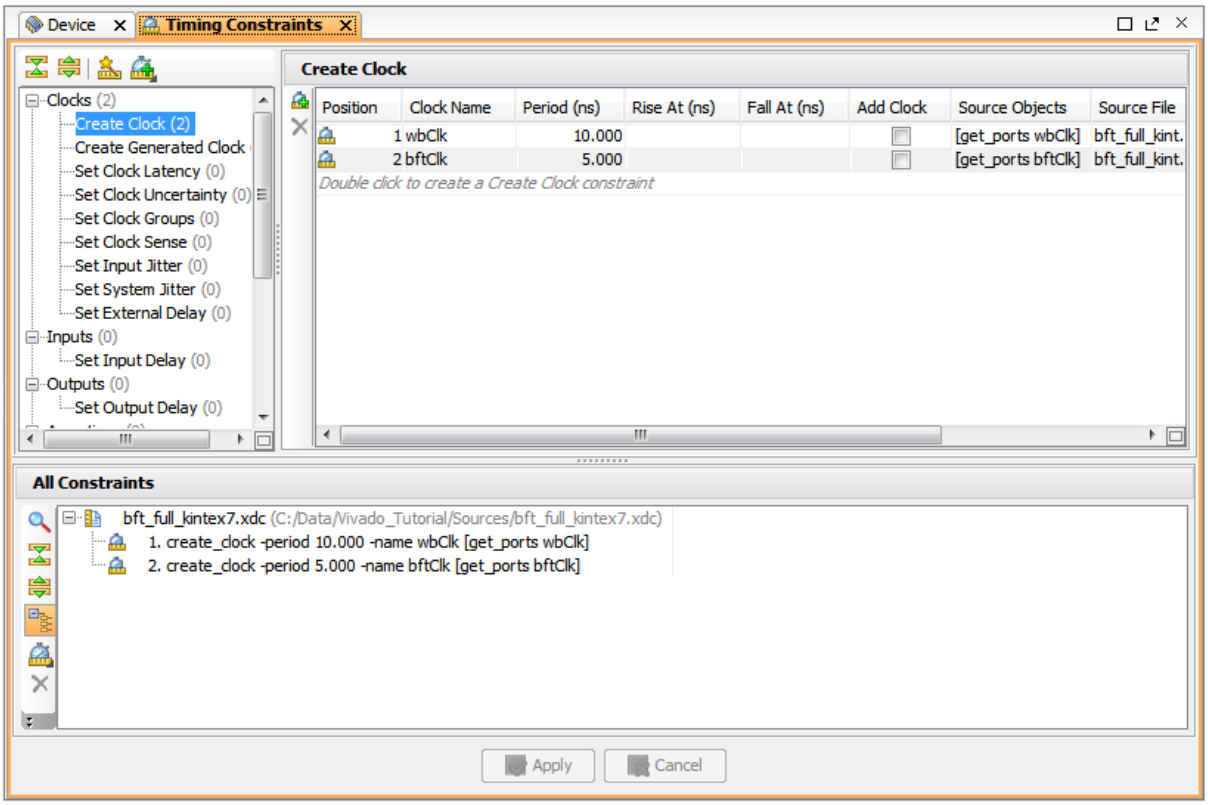

Figure 3: Define Timing Constraints

A tree view of the different types of constraints displays on the left side of the Timing Constraints window. This is a menu of timing constraints that can be quickly defined.

Notice the two clock constraints, `wbClk` and `bftClk`, displayed in the Timing Constraint spreadsheet on the right side of the Timing Constraints window. The values of currently defined constraints can be modified by directly editing them in the spreadsheet.

2. In the left hand tree view of the Timing Constraints window, double-click **Create Clock** under the Clocks category, as shown in Figure 3.

Note: *Expand the Clocks category if needed by clicking the* **+.**

The Create Clock wizard opens, as shown in the following figure, to help you define clock constraints. Notice the Tcl Command line on the bottom displays the XDC command that will be executed.

Do not create or modify any timing constraints at this time.

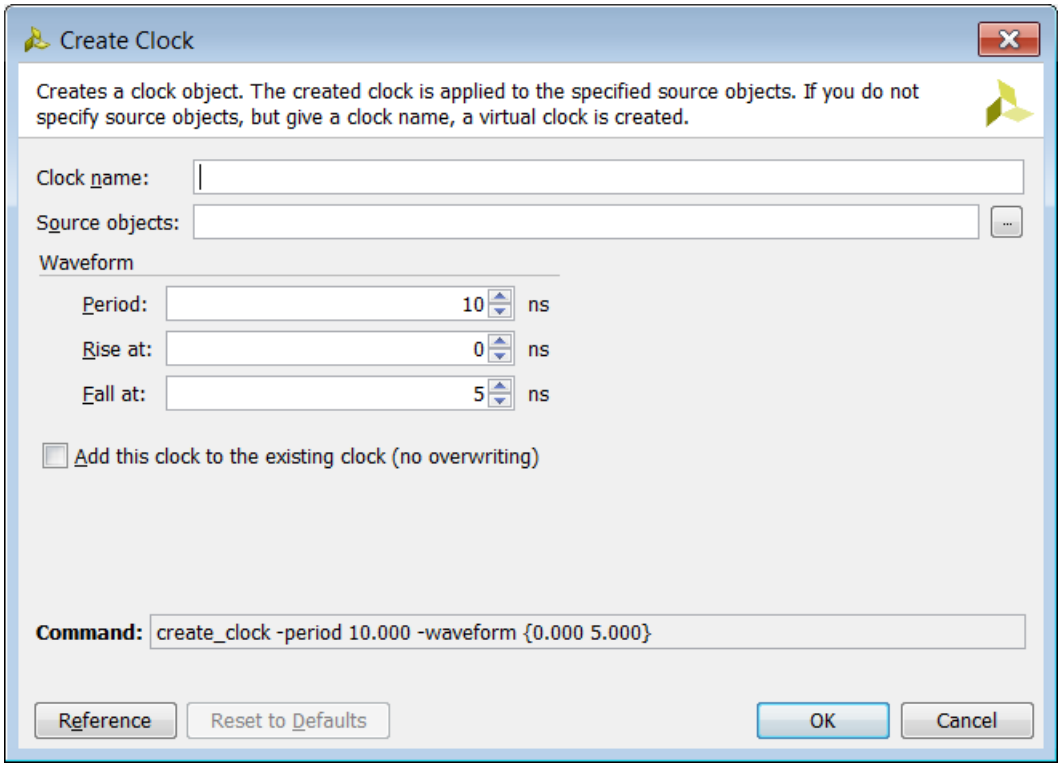

Figure 4: Create Clock Dialog Box

3. Click **Cancel**.
4. Close the **Timing Constraints** window by clicking the **X** in the window tab.

   The Vivado Design Suite offers a variety of features for design analysis and constraint assignment. Other tutorials cover these features in detail, and they are only mentioned here. Feel free to examine some of the features under the Tools menu.matl

I/O Planning

Vivado has a comprehensive set of capabilities for performing and validating I/O pin assignments. These are covered in greater detail in the I/O Planning Tutorial.

1. Open the I/O Planning view layout by selecting **I/O Planning** from the **Layout Selector** pull down, as shown in Figure 5.

2. Make the Package window the active view if it is not active.

   Note: *If the Package window is not open, you can open it using the* ***Windows > Package*** *command from the main menu.*

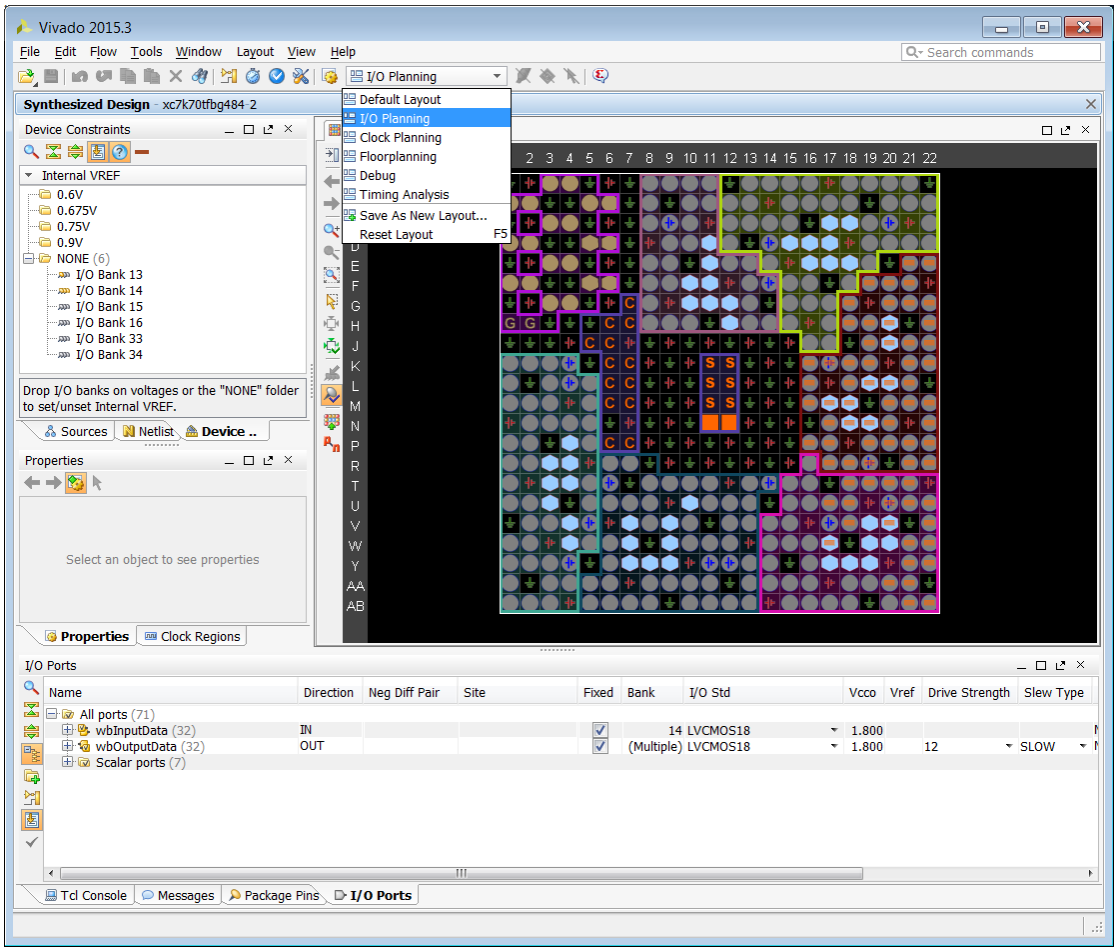

Figure 5: Open I/O Planning View Layout

3. In the Package window, double-click to select a placed I/O Port, shown as an orange block inside a package pin.
4. Drag the selected I/O Port onto another pin site in the same I/O bank.
5. Examine the I/O Ports window, look at the port name and package pin site columns.
6. Examine the data displayed in the I/O Port Properties window. Click each of the tabs at the bottom of the window.
7. Remember the port name and site of the port you moved.

   If necessary, write them down. You will look for the LOC constraint of the placed port in the XDC file after implementation.

Step 6: Exporting the Modified Constraints

Modified constraints can be output for later use. You can also save design checkpoints that include the latest changes. You will explore design checkpoints later in this tutorial.

IMPORTANT: *The Vivado Design Suite does not support NCF/UCF constraints. You should migrate existing UCF constraints to XDC format. Refer to the ISE to Vivado Design Suite Migration Guide (UG911) for more information.*

1. Use the **Export Constraints** command to output a modified XDC constraints file with the new I/O LOC constraint value.

   File > Export > Export Constraints

   The Export Constraints dialog box opens to let you specify a file name to create, as shown in the following figure.

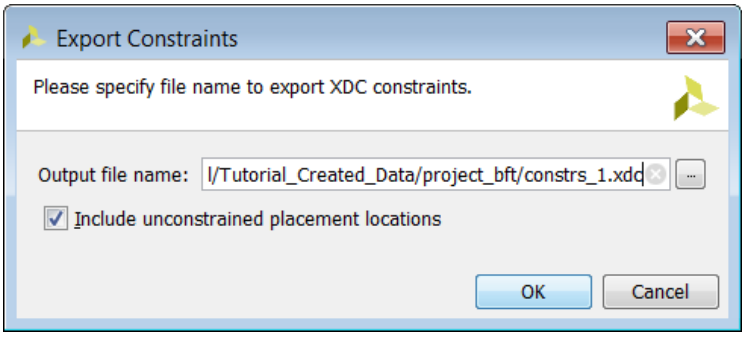

Figure 6: Export Constraints

2. Enter a name and location for the file and click **OK**.

   Notice the checkbox for **Include unconstrained placement locations**. When this is enabled, LOC constraints of all placed cells are exported, rather than of only fixed cells. For a more detailed description of fixed versus unfixed cells, refer to this link in the *Vivado Design Suite User Guide: Using Constraints* (UG903).

3. Use the **File > Open File** command to open the constraints file in the Text Editor.
4. Browse to select the newly exported constraints file and click **OK**.
5. Notice the file reflects the I/O Port placement change you made earlier.

   You can open any ASCII file in the Text Editor. This is helpful for editing Tcl scripts and constraints files, and viewing reports. The Text Editor is context sensitive, and highlights keywords and comments when displaying file types such as Verilog, VHDL, XDC, and Tcl.

6. Select the **Tcl Console** tab at the bottom of the IDE, and enter the `stop_gui` command.

   The Vivado IDE closes, and you are returned to the Tcl prompt in the Tcl shell.

Step 7: Implementing the Design

1. Open the `create_bft_kintex7_batch.tcl` script, or bring the script window to the front.
2. Individually copy and paste the Tcl commands in the script, in order from `opt_design` to `write_bitstream`:

```
opt_design
place_design
phys_opt_design
write_checkpoint -force $outputDir/post_place
report_timing_summary -file $outputDir/post_place_timing_summary.rpt

route_design
write_checkpoint -force $outputDir/post_route
report_timing_summary -file $outputDir/post_route_timing_summary.rpt
report_timing -sort_by group -max_paths 100 -path_type summary -file \
$outputDir/post_route_timing.rpt

report_clock_utilization -file $outputDir/clock_util.rpt
report_utilization -file $outputDir/post_route_util.rpt
report_power -file $outputDir/post_route_power.rpt
report_drc -file $outputDir/post_imp_drc.rpt

write_verilog -force $outputDir/bft_impl_netlist.v
write_xdc -no_fixed_only -force $outputDir/bft_impl.xdc
write_bitstream -force $outputDir/bft.bit
```

3. Examine each command and notice the various messages produced as the commands are run.
4. Close the text editor displaying the `create_bft_kintex7_batch.tcl` script.
5. Examine the files created in the output directory.

   `<Extract_Dir>/Vivado_Tutorial/Tutorial_Created_Data/bft_output`
6. Use a text editor to open the various report (`*.rpt`) files that were created.
7. Open the `bft_impl.xdc` file.
8. Validate that the design has been implemented with the I/O Port constraint that you modified earlier.

Step 8: Opening a Design Checkpoint

The Vivado IDE can open any saved design checkpoint. This snapshot of the design can be opened in the Vivado IDE or Tcl shell for synthesis, implementation, and analysis.

1. Open the Vivado IDE again: **`start_gui`**

   This loads the active design in-memory into the IDE.

   You will now load the implemented design checkpoint, closing the current in-memory design.

2. Open the implemented checkpoint.

   Use **File > Open Checkpoint** and browse to select the checkpoint file:

   `<Extract_Dir>/Vivado_Tutorial/Tutorial_Created_Data/bft_output/post_route.dcp`

3. If prompted, select **Close Without Saving** to close the current in-memory design.

   Now you can use the visualization and analysis capabilities of the IDE, working from a placed and routed design checkpoint.

Step 9: Analyzing Implementation Results

Vivado has an extensive set of features to examine the design and device data from a number of perspectives. You can generate standard reports for power, timing, utilization, clocks, etc. With the Tcl API, the custom reporting capabilities in the Vivado tools are extensive.

1. Click on the Device window tab to bring it front to the screen.
2. Run the `report_timing_summary` command to analyze timing data.

   Tools > Timing > Report Timing Summary

3. In the Report Timing Summary dialog, click **OK** to accept the default run options.

   Examine the information available in the Timing Summary window. Select the various categories from the tree on the left side of the Timing Summary window and examine the data displayed.

4. Now run the `report_timing` command to perform timing analysis

   Tools > Timing > Report Timing

5. In the Report Timing dialog, click **OK** to accept the default run options.
6. Collapse the **bftClk** tree in the Timing Checks – Setup window.
7. Select the first path listed under the wbClk in the Setup area.
8. Maximize or float the Path Properties window to look at the path details.

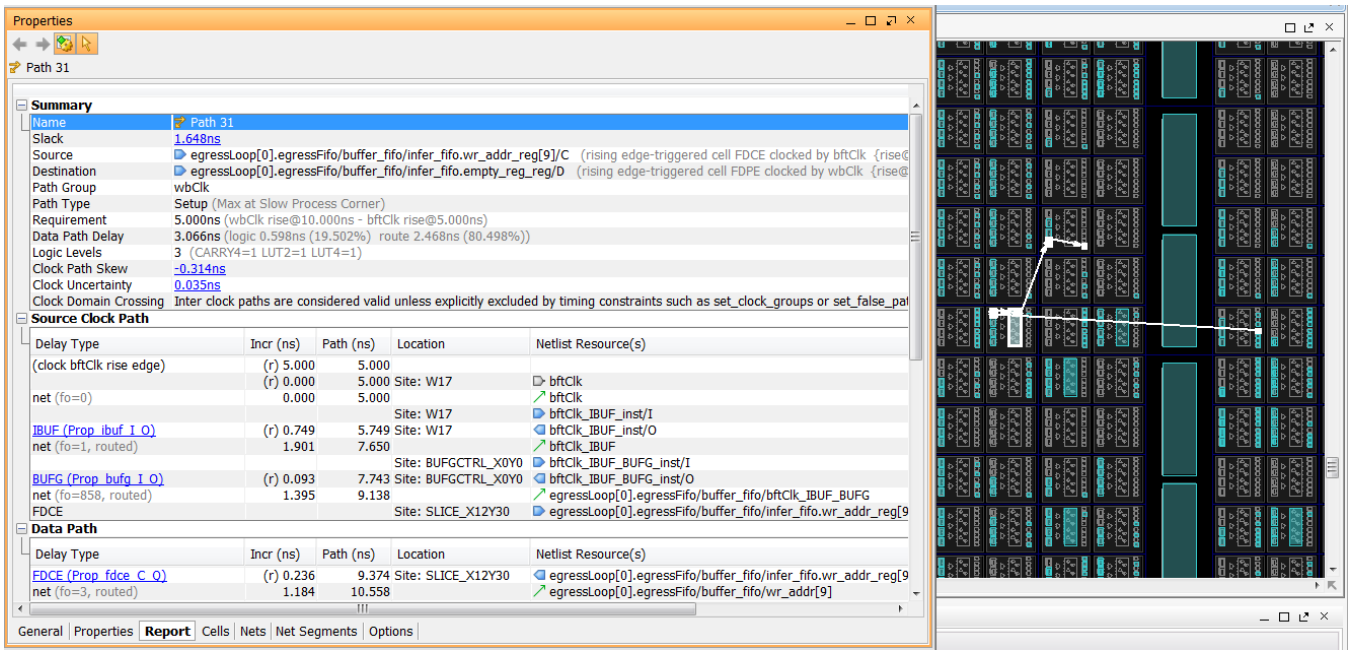

Figure 7: Float the Path Properties Window

9. Restore the Path Properties window by clicking the **Restore** button, or the **Dock** button, in the window banner.
10. In the Timing – Report Timing window, right-click to open the popup menu and select the **Schematic** command to open a Schematic window for the selected path.

    Note: *Alternatively, you can press the **F4** function key to open the Schematic window.*

11. Double-click on a schematic object, such as on a cell, pin, or wire, to expand the schematic connections and traverse the design hierarchy.
12. Close the Schematic window, or click the Device window tab to bring it to the front.
13. In the Device window, check to ensure that the Routing Resources button is enabled to display the detailed device routing.

    Notice the Device window displays and highlights the routing for the selected path.

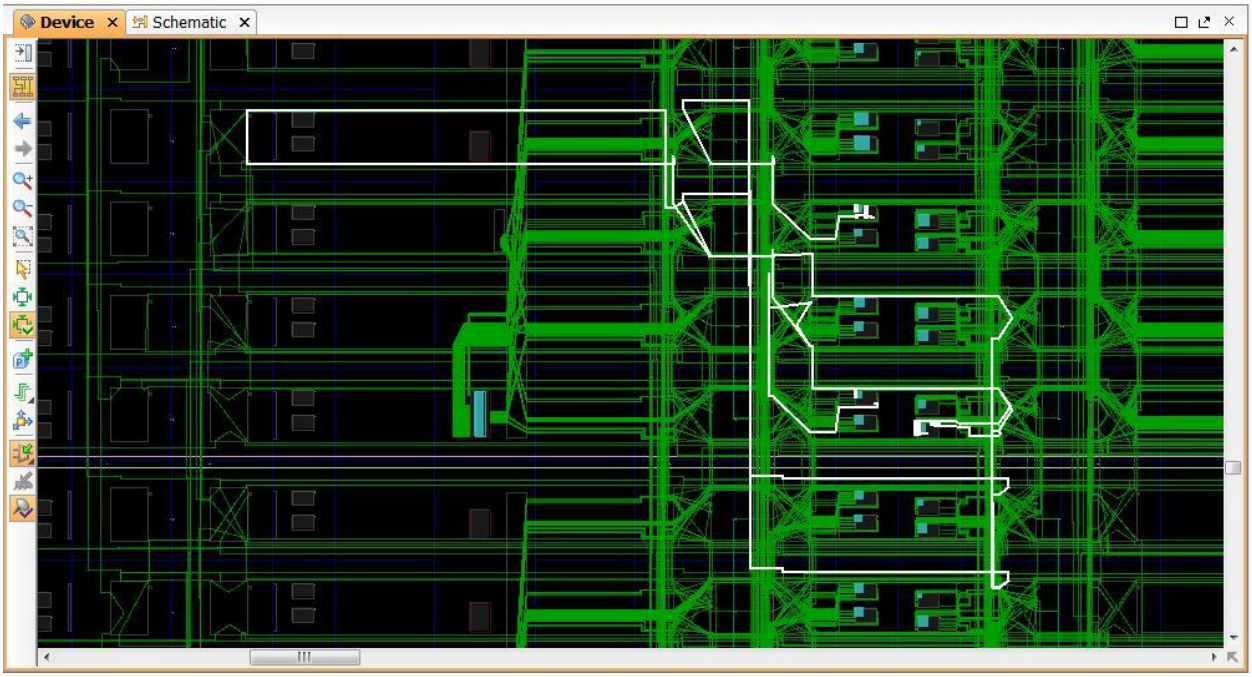

Figure 8: Displaying the Device Routing

14. Select the Auto Fit Selection button in the Device window toolbar menu to enable the Vivado IDE to automatically zoom into selected objects.
15. Select some additional paths from the Timing results window.
16. Examine the routing for the selected paths in the Device window.
17. Expand the **Tools** main menu and examine the available analysis features under the different sub-menus such as **Timing** and **Report**.
18. Run some of the available analysis commands: **Report Power**, **Report Clock Interaction**, **Report Clock Networks**, **Report Utilization**, etc.

    Many of these Design Analysis features are covered in other Vivado tutorials.

Step 10: Exiting the Vivado Tool

The Vivado tool writes a log file, called `vivado.log`, and a journal file called `vivado.jou` into the directory from which Vivado was launched. The log file is a record of the Tcl commands run during the design session, and the messages returned by the tool as a result of those commands. The journal is a record of the Tcl commands run during the session that can be used as a starting point to create new Tcl scripts.

Exit the Vivado IDE:

1. Select the Tcl Console window tab and type the following:

   ```
   stop_gui
   ```

2. Exit Vivado:

   ```
   Vivado% exit
   ```

3. Examine the Vivado log (`vivado.log`) file.

   On Windows, it may be easier to use the file browser to locate and open the log file. The location of the Vivado log and journal file will be the directory from which the Vivado tool was launched, or can be separately configured in the Windows desktop icon. You will configure this in Lab #2.

   In this case, look for the log file at the following location:

   `<Extract_Dir>/Vivado_Tutorial/vivado.log`

   > ***Note**: The `vivado.log` and `vivado.jou` may also be written to `%APPDATA%\Xilinx\Vivado`, or to your `/home` directory.*

   Notice the log file contains the history and results of all Tcl commands executed during the Vivado session.

4. Examine the Vivado journal (`vivado.jou`) file.

   On Windows, it may be easier to use the file browser. Look for the journal file at the following location:

   `<Extract_Dir>/Vivado_Tutorial/vivado.jou`

   Notice the journal file contains only the Tcl commands executed during the Vivado session, without the added details recorded in the log file. The journal file is often helpful when creating Tcl scripts from prior design sessions, as you will see in the next lab.

Introduction

In this lab, you will learn about the Project mode features for project creation, source file management, design analysis, constraint definition, and synthesis and implementation run management.

You will walk through the entire FPGA design flow using an example design, starting in the Vivado IDE. Then you will examine some of the major features in the IDE. Most of these features are covered in detail in other tutorials. Finally, you will create a batch run script to implement the design project and see how easy it is to switch between running Tcl scripts and working in the Vivado IDE.

Step 1: Creating a Project

Launching Vivado

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. Before clicking the desktop icon to launch the Vivado tool, configure the icon to indicate where to write the `vivado.log` and `vivado.jou` files.
2. Right-click the Vivado 2015.x Desktop icon and select **Properties** from the popup menu.
3. Under the **Shortcut** tab, set the **Start in** value to the extracted Vivado Tutorial directory, as shown in Figure 9:

   `<Extract_Dir>/Vivado_Tutorial/`

4. Click **OK** to close the Properties dialog box.

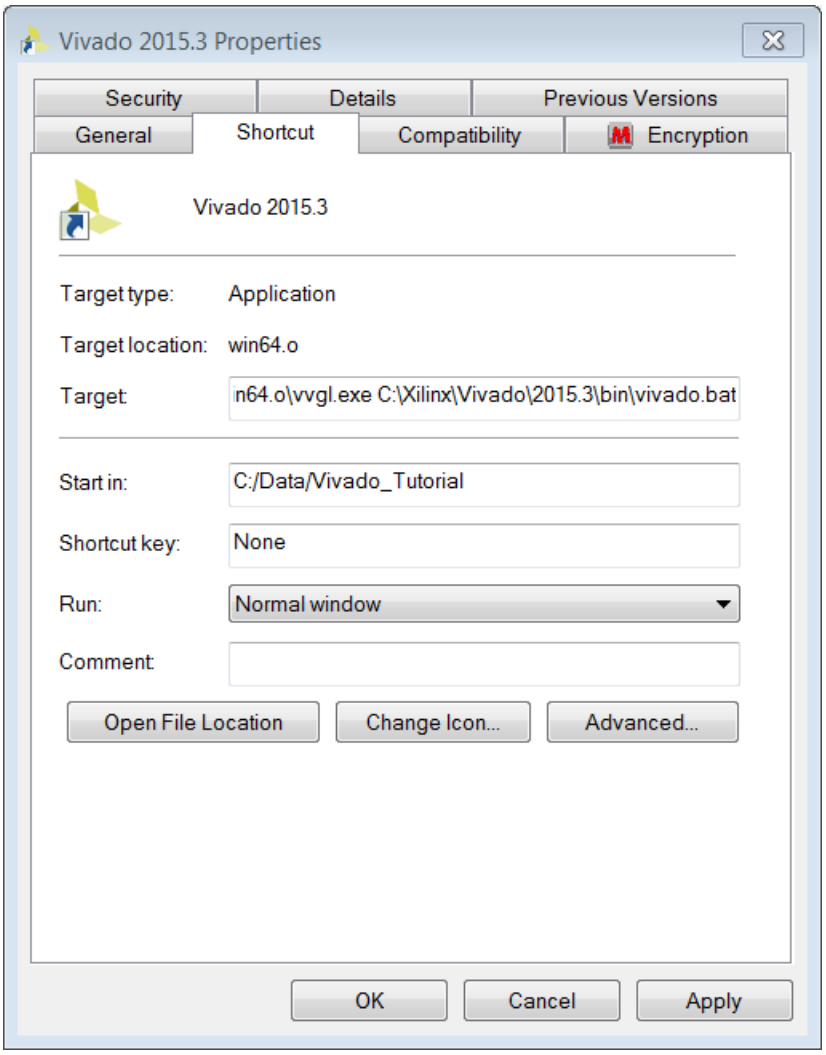

Figure 9: Configuring the Start in Directory

5. Double-click the **Vivado 2015.x** Desktop icon to start the Vivado IDE.

Creating a New Project

1. After Vivado opens, select **Create New Project** on the Getting Started page.
2. Click **Next** in the New Project wizard.
3. Specify the Project Name and Location:
   a. Project name: **`project_bft`**
   b. Project Location: **`<Extract_Dir>/Vivado_Tutorial/Tutorial_Created_Data`**
4. Click **Next**.

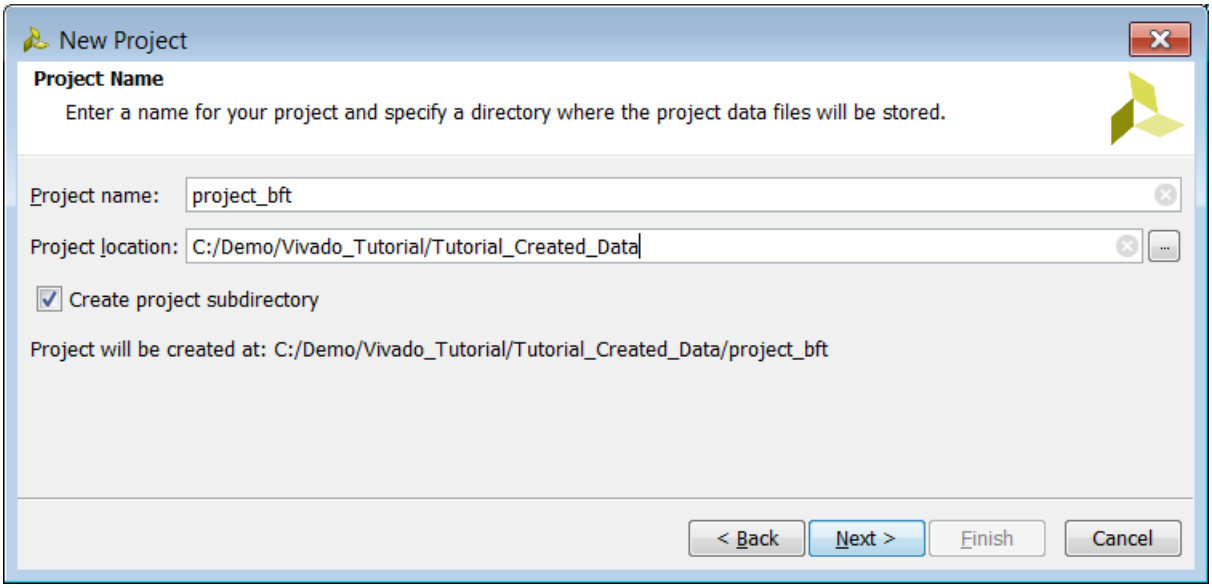

Figure 10: Create New Project

5. Select **RTL Project** as the **Project Type** and click **Next**.
6. Click the + button and select **Add Files**.
   a. Browse to `<Extract_Dir>/Vivado_Tutorial/Sources/hdl/`
   b. Press and hold the **Ctrl** key, and click to select the following files:
      `async_fifo.v, bft.vhdl, bft_tb.v, FifoBuffer.v`
   c. Click **OK** to close the File Browser.
7. Click the + button and select **Add Directories**.
   a. Select the `<Extract_Dir>/Vivado_Tutorial/Sources/hdl/bftLib` directory
   b. Click **Select**.

8. Click in the **HDL Sources For** column for the `bft_tb.v` file and change Synthesis and Simulation to **Simulation only**, as shown in Figure 11.
9. Click in the **Library** column for the `bftLib`, and manually edit the value to change it from `xil_defaultlib` (or work) to **`bftLib`**, as shown in the following figure.

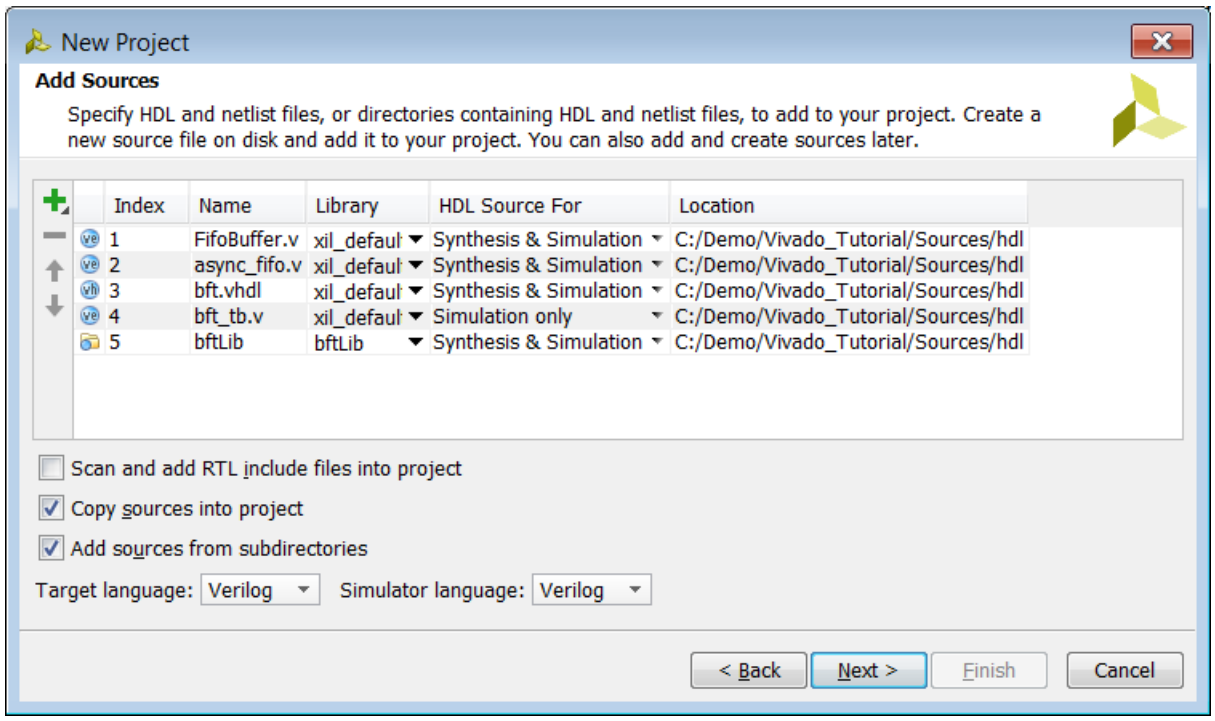

Figure 11: Add RTL Sources

10. Enable the check boxes for **Copy sources into project**, and **Add sources from subdirectories**.
11. Set the **Target Language** to **Verilog** to define the language of the netlist generated by Vivado synthesis.
12. Set the **Simulator Language** to **Verilog** to define the language required by the logic simulator.
13. Click **Next**.
14. Click **Next** again to skip past the Add Existing IP page, since you will not be adding IP at this time.
15. On the Add Constraints page, **click Add Files**.
16. Browse to and select

    `<Extract_Dir>/Vivado_Tutorial/Sources/bft_full_kintex7.xdc`
17. Click **OK** to close the File Browser.
18. Enable the check box for **Copy constraints files into project**.

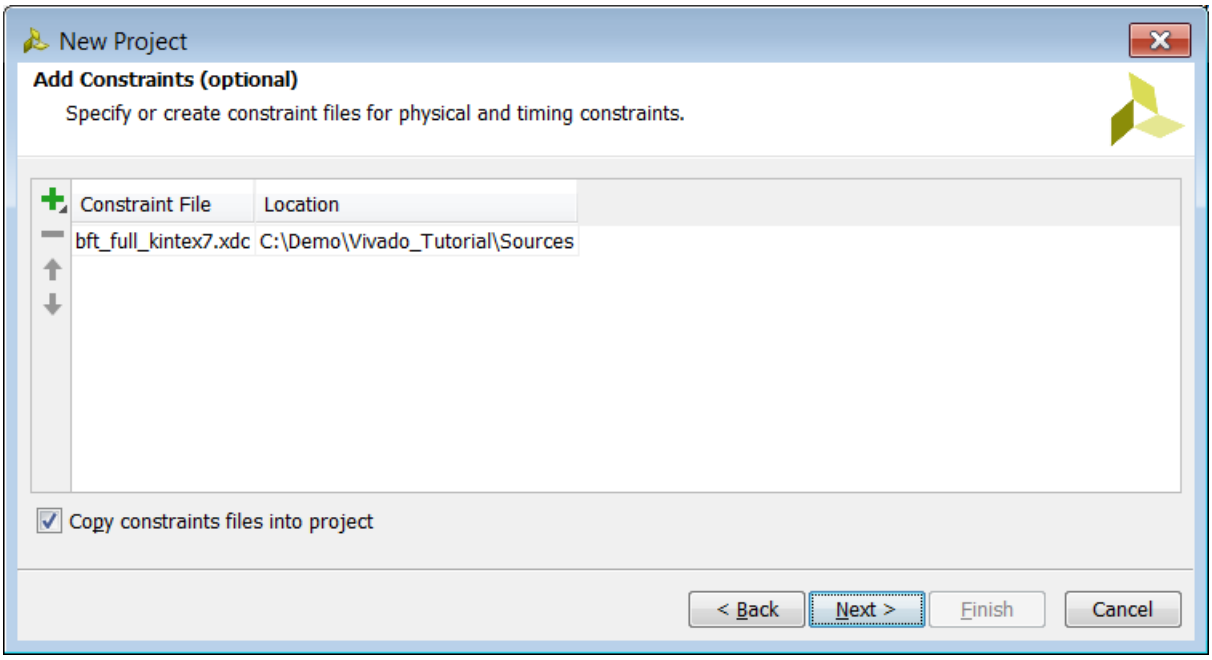

Figure 12: Add Constraints

19. Click **Next** to move to the Default Part page.
20. On the Default Part page, click the **Family** filter and select the **Kintex-7** family.

21. Scroll to the top of the list and select the `xc7k70tfbg484-2` part, and click **Next**.

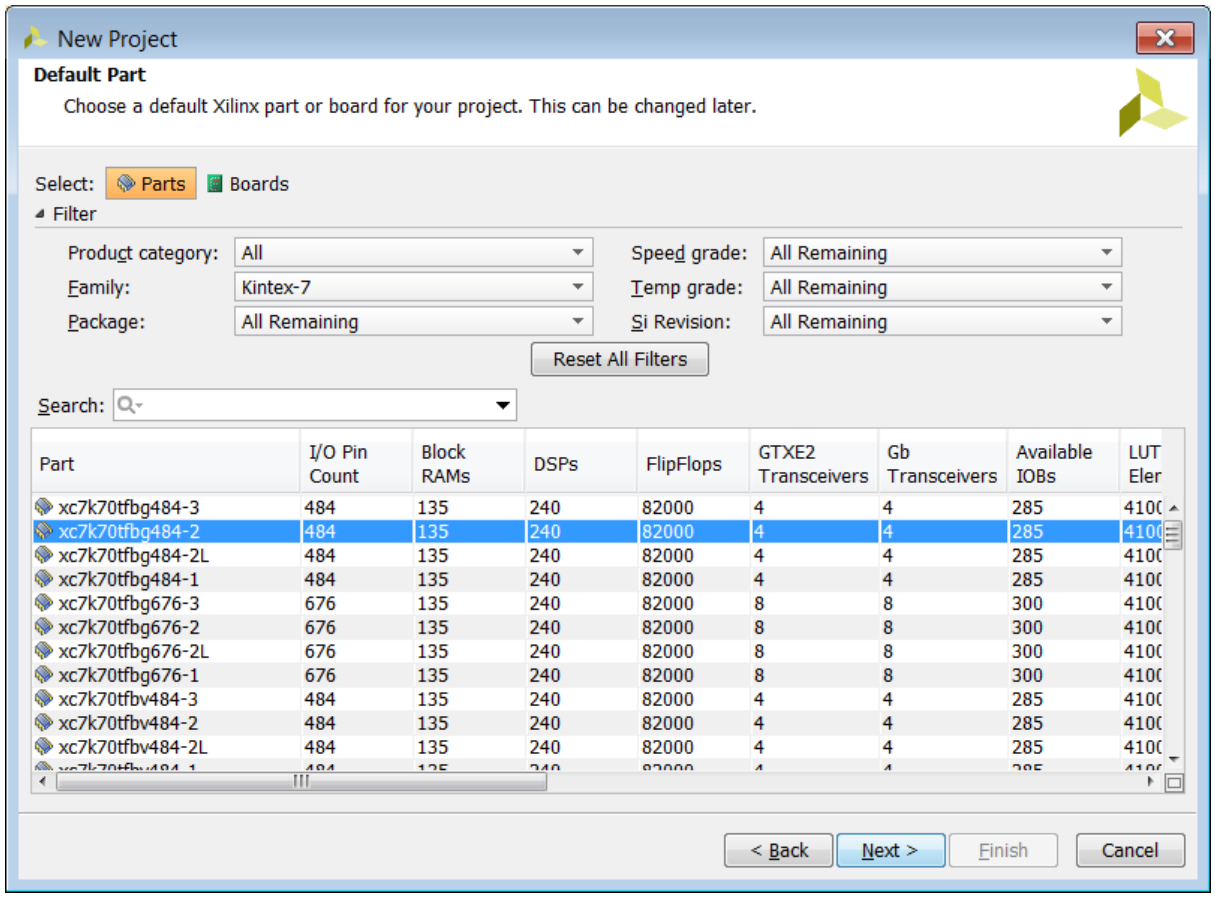

Figure 13: Selecting the Default Part

22. Click **Finish** to close the New Project Summary page, and create the project.

The Vivado IDE opens `project_bft` in the default layout.

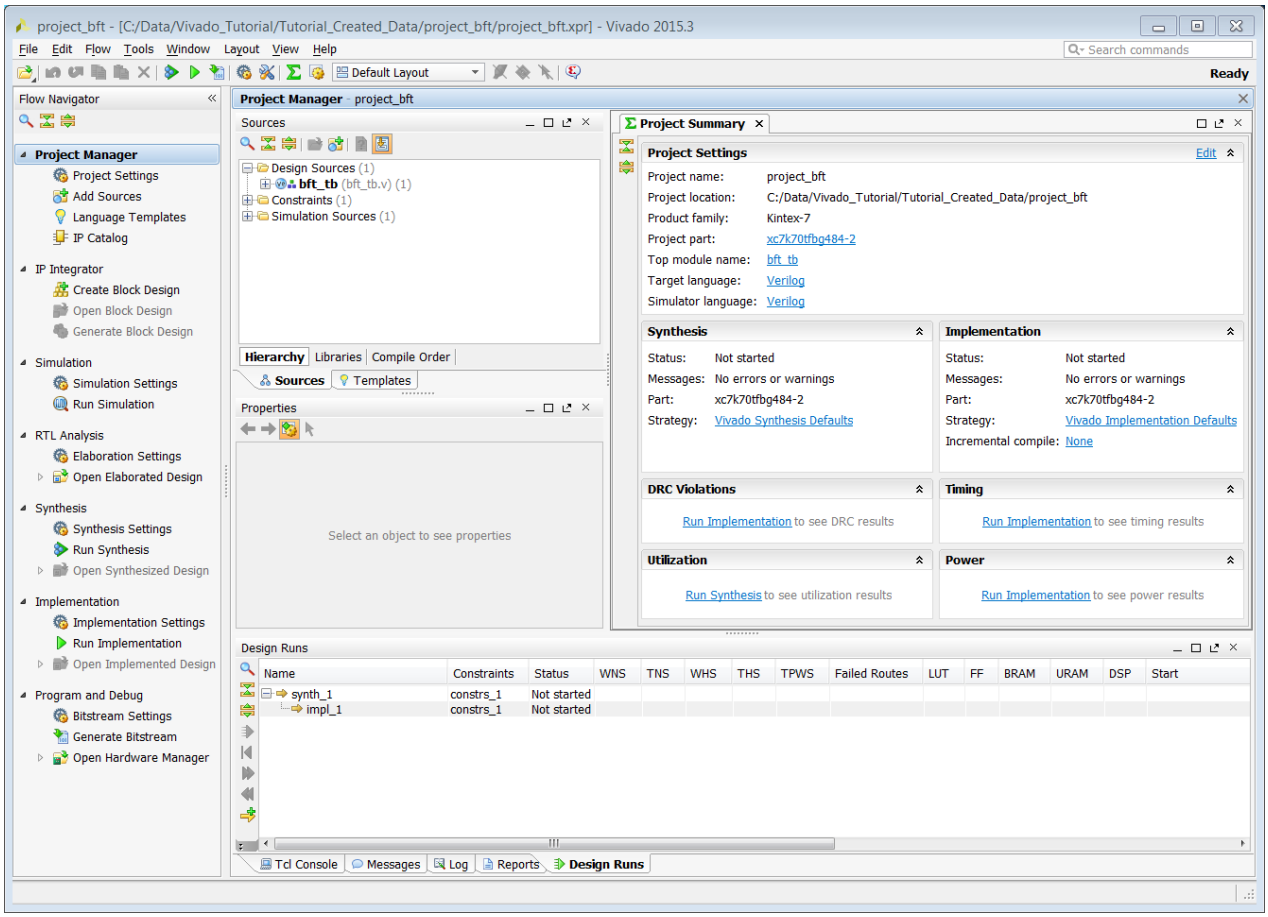

Figure 14: Project BFT in the Vivado IDE

Step 2: Using the Sources Window and Text Editor

The Vivado tool lets you add different design sources including Verilog, VHDL, EDIF, NGC format cores, SDC, XDC, DCP design checkpoints, TCL constraints files, and simulation test benches. These files can be sorted in a variety of ways using the tabs at the bottom of the Sources window (**Hierarchy**, **Library** or **Compile Order**).

IMPORTANT: NGC format files are not supported in the Vivado Design Suite for UltraScale devices. It is recommended that you regenerate the IP using the Vivado Design Suite IP customization tools with native output products. Alternatively, you can use the NGC2EDIF command to migrate the NGC file to EDIF format for importing. However, Xilinx recommends using native Vivado IP rather than XST-generated NGC format files going forward.

The Vivado IDE includes a context sensitive text editor to create and develop RTL sources, constraints files, and Tcl scripts. You can also configure the Vivado IDE to use third party text editors. Refer to this link in the *Vivado Design Suite User Guide: Using the IDE* (UG893) for information on configuring the Vivado tool.

Exploring the Sources Window and Project Summary

1. Examine the information in the **Project Summary**. More detailed information is presented as the design progresses through the design flow.
2. Examine the Sources window and expand the **Design Sources**, **Constraints** and **Simulation Sources** folders.

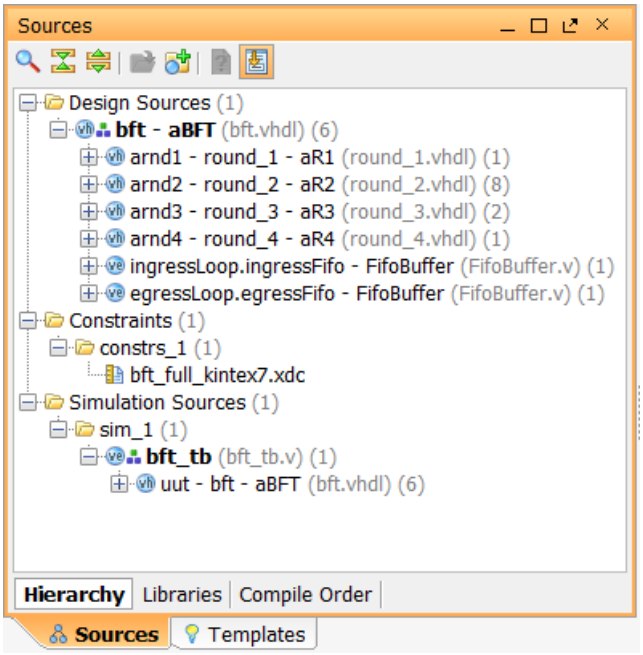

Figure 15: Viewing Sources

The Design Sources folder helps keep track of VHDL and Verilog source files and libraries. Notice the Hierarchy tab is displayed by default.

3. Select the **Libraries** tab and the **Compile Order** tabs in the Sources window and notice the different ways that sources are listed.

   The Libraries tab groups source files by file type. The Compile Order tab shows the file order used for synthesis.

4. Expand the various folders to view the design source information.
5. Select the **Hierarchy** tab.

Exploring the Text Editor

1. Select one of the VHDL sources in the Sources window.
2. Right-click to review the commands available in the popup menu.
3. Select **Open File**, and use the scroll bar to browse the file contents in the Text Editor.

   You can also double-click source files in the Sources window to open them in the Text Editor.

Figure 16: Context Sensitive Text Editor

Notice that the Text Editor displays the RTL code with context sensitive coloring of keywords and comments. The colors and fonts used to display reserved words can be configured using the **Tools > Options** command. Refer to this link in the *Vivado Design Suite User Guide: Using the IDE* (UG893) for more information.

4. With the cursor in the Text Editor, right-click and select **Find in Files**. Note the **Replace in Files** command as well.

   The Find in Files dialog box opens with various search options.

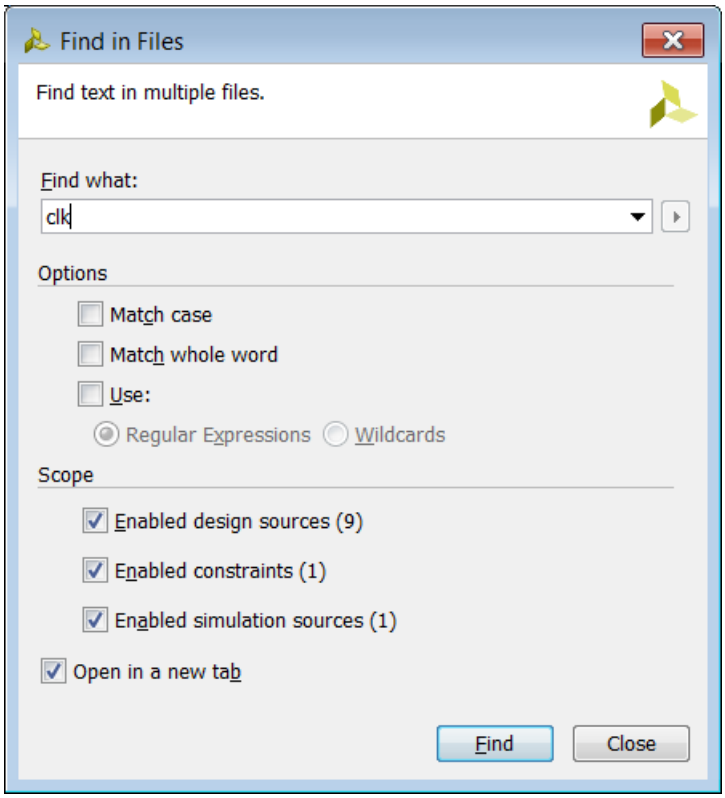

Figure 17: Using Find in Files

5. Enter **`clk`** in the **Find what:** field, and click **Find**.

   The Find in Files window displays in the messaging area at the bottom of the Vivado IDE.

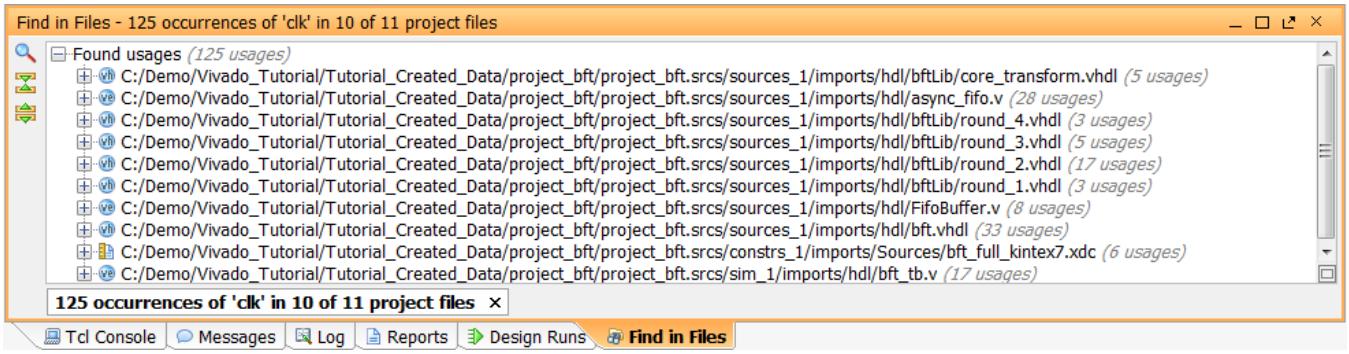

Figure 18: Viewing the Find in Files Results

6. In the Find in Files window, expand one of the displayed files, and select an occurrence of **clk** in the file.

   Notice that the Text Editor opens the selected file and displays the selected occurrence of **clk** in the file.

7. Close the Find in Files – Occurrences window.
8. Close the open Text Editor windows.

The next few steps highlight some of the design configuration and analysis features available prior to running synthesis.

Step 3: Elaborating the RTL Design

The Vivado IDE includes an RTL analysis and IP customizing environment. There are also several RTL Design Rule Checks (DRCs) to examine ways to improve performance or power on the RTL design.

1. Select **Open Elaborated Design** in the Flow Navigator to elaborate the design.

   Note: *A dialog box appears informing you that your current settings will slow down netlist elaboration. You can click* ***OK*** *to continue or* ***Cancel*** *to return to your project and edit your Elaboration Settings, available in the Flow Navigator.*

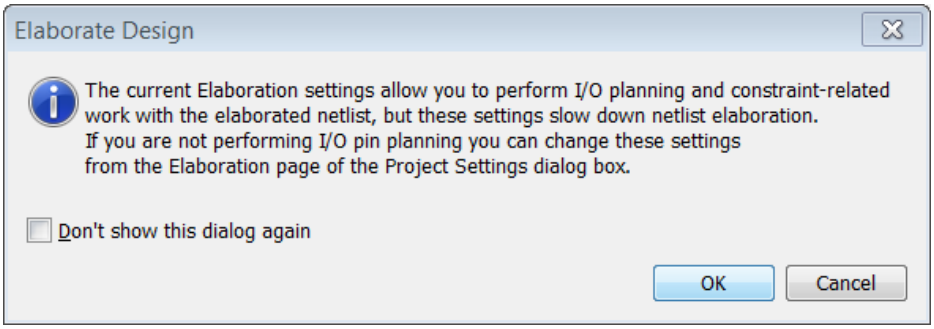

Figure 19: Elaborate Design Dialog Box

2. Ensure that the Layout Selector pull down menu in the main Toolbar has **Default Layout** selected.

   The Elaborated Design enables various analysis views including an RTL Netlist, Schematic, and Graphical Hierarchy. The views have a cross-select feature, which helps you to debug and optimize the RTL.

3. Explore the logic hierarchy in the RTL Netlist window and examine the Schematic.

   You can traverse the schematic by double-clicking on cells to push into the hierarchy, or by using commands like the **Expand Cone** or **Expand/Collapse** from the Schematic popup menu. Refer to this link in the *Vivado Design Suite User Guide: Using the Vivado IDE* (UG893) for more information on using the Schematic window.

4. Select any logic instance in the Schematic and right-click to select the **Go to Source** or **Go to Definition** commands**.**

The Text Editor opens the RTL source file for the selected cell with the logic instance highlighted. In the case of the **Go to Definition** command, the RTL source file containing the module definition is opened. With **Go to Source**, the RTL source containing the instance of the selected cell is opened.

5. Click on the Messages window at the bottom of the Vivado IDE, and examine the messages.
6. Click the **Collapse All** button in the Messages toolbar.
7. Expand the Elaborated Design and the synth_design –name rtl_1 messages

   Notice there are links in the messages to open the RTL source files associated with a message.
8. Click one of the links and the Text Editor opens the RTL source file with the relevant line highlighted.
9. Close the Text Editor windows.
10. Close the Elaborated Design by clicking on the **X** on the right side of the Elaborated Design window banner, and click **OK** to confirm.

Step 4: Using the IP Catalog

The Xilinx IP Catalog provides access to the Vivado IP configuration and generation features. You can sort and search the Catalog in a variety of ways. IP can be customized, generated, and instantiated.

1. Click the **IP Catalog** button in the Flow Navigator.
2. Browse the IP Catalog to examine the various categories and IP filtering capabilities.
3. Click the Group icon and select **Group > Group by taxonomy and repository**
4. Expand the **Basic Elements** folder.
5. Double-click **DSP48 Macro**.

   The Customize IP dialog is opened directly within Vivado Design Suite, which allows you to perform native customization and configuration of IP within the tool. To learn more about IP configuration and implementation, see the *Vivado Design Suite User Guide: Designing with IP* (UG896) and the *Vivado Design Suite Tutorial: Designing with IP* (UG939).
6. Click **Cancel** to close the Customize IP dialog without adding the IP to the current design.
7. Close the IP Catalog tab by clicking on the **X** on the right side of the window banner.

Step 5: Running Behavioral Simulation

The Vivado IDE integrates the Vivado Simulator, which enables you to add and manage simulation sources in the project. You can configure simulation options, and create and manage simulation source sets. You can run behavioral simulation on RTL sources, prior to synthesis.

1. In the Flow Navigator, under Simulation, click the **Simulation Settings** command.

   The Project Settings dialog box opens.

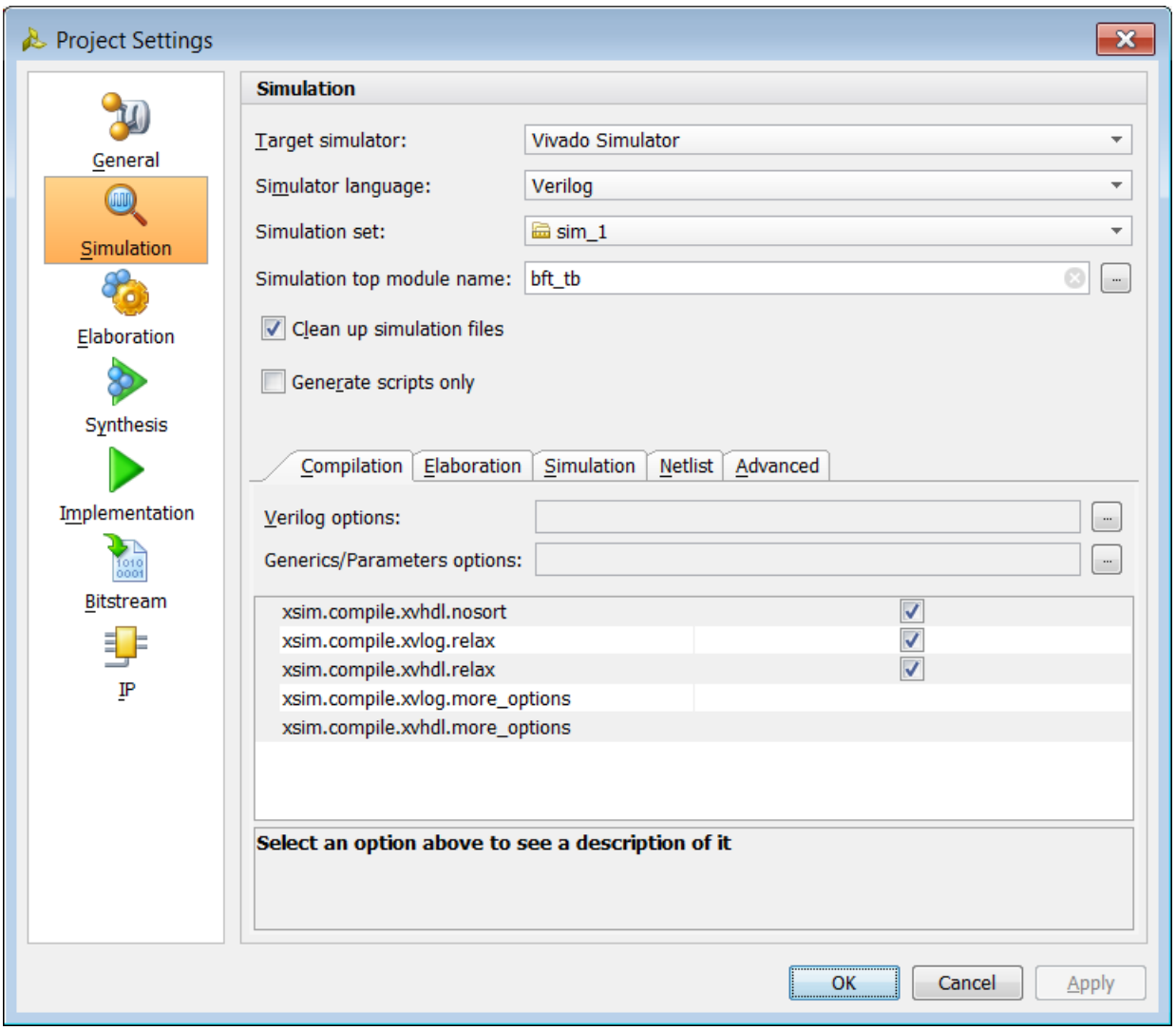

Figure 20: Simulation Settings - Update

2. Examine the settings available under the **Simulation** tab: then click **Cancel** to close the dialog box.
3. Click the **Run Simulation** command in the Flow Navigator, then click the **Run Behavioral Simulation** in the sub-menu.

4. Examine and explore the Simulation environment.

   Simulation is covered in detail in the *Vivado Design Suite User Guide: Logic Simulation* (UG900) and the *Vivado Design Suite Tutorial: Logic Simulation* (UG937).

5. Close the simulation by clicking the **X** icon on the Behavioral Simulation view banner.
6. Click **No** if prompted to save changes.

Step 6: Reviewing Design Run Settings

One of the main differences between the Non-Project mode you used in Lab #1 and the Project mode, which you are now using, is the support of design runs for synthesis and implementation. Non-Project mode does not support design runs.

Design runs are a way of configuring and storing the many options available in the different steps of the synthesis and implementation process. You can configure these options and save the configurations as strategies to be used in future runs. You can also define `Tcl.pre` and `Tcl.post` scripts to run before and after each step of the process, to generate reports before and after the design progresses.

Before launching the synthesis and implementation runs you will review the settings and strategies for these runs.

1. In the Flow Navigator, select **Synthesis Settings** under Synthesis.

   The Project Settings dialog box opens. The Synthesis Settings provide you access to the many options available for configuring Vivado synthesis. For a complete description of these options, see this link in the *Vivado Design Suite User Guide: Synthesis* (UG901).

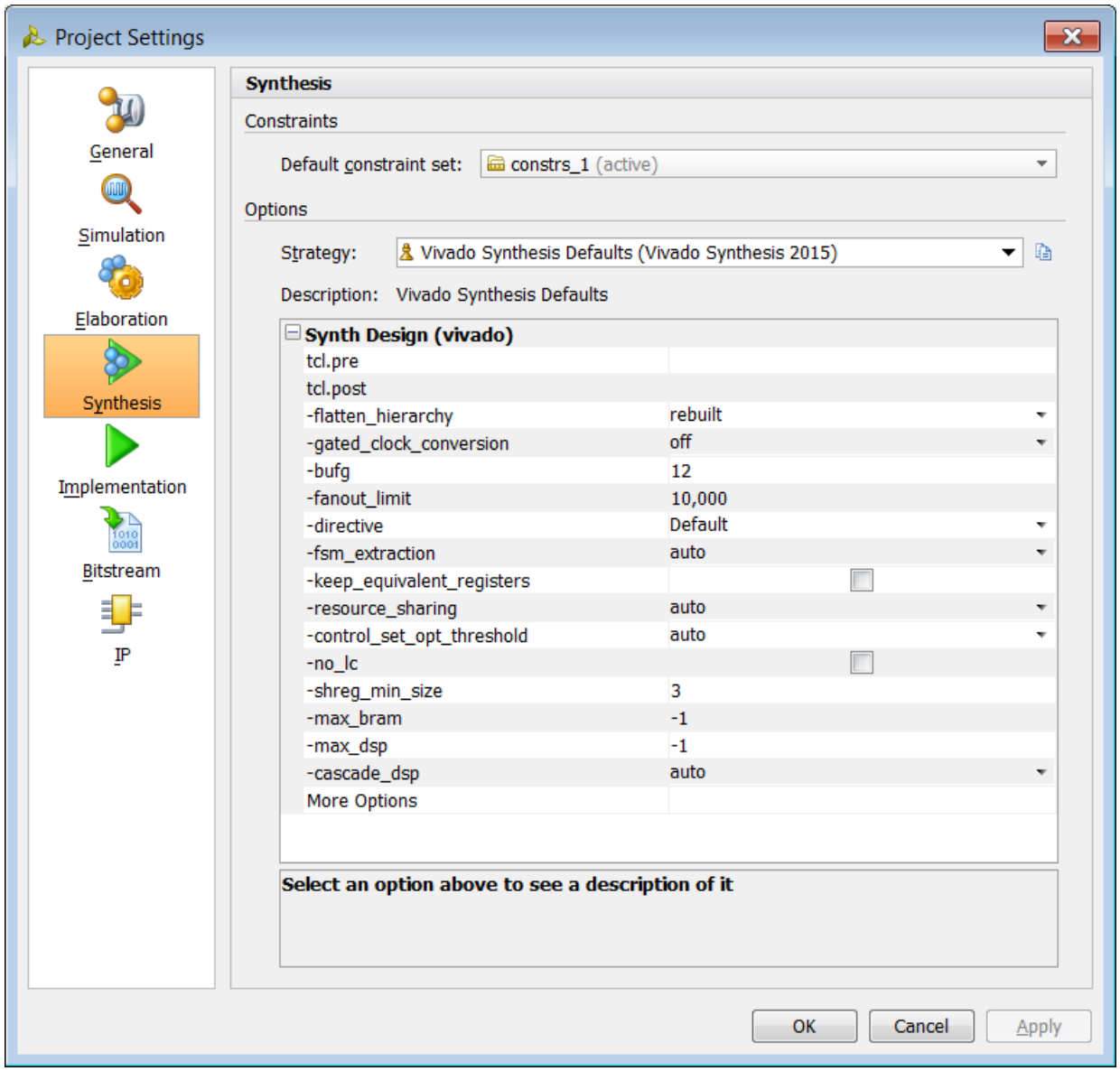

Figure 21: Synthesis Settings

2. After reviewing the various synthesis options, select the **Implementation** button on the left side of the Project Settings dialog box, as shown in Figure 22.

   The Project Settings change to reflect the Implementation settings. You can view the available options for implementation runs. For a complete description of these options, see this link in the *Vivado Design Suite User Guide: Implementation* (UG904).

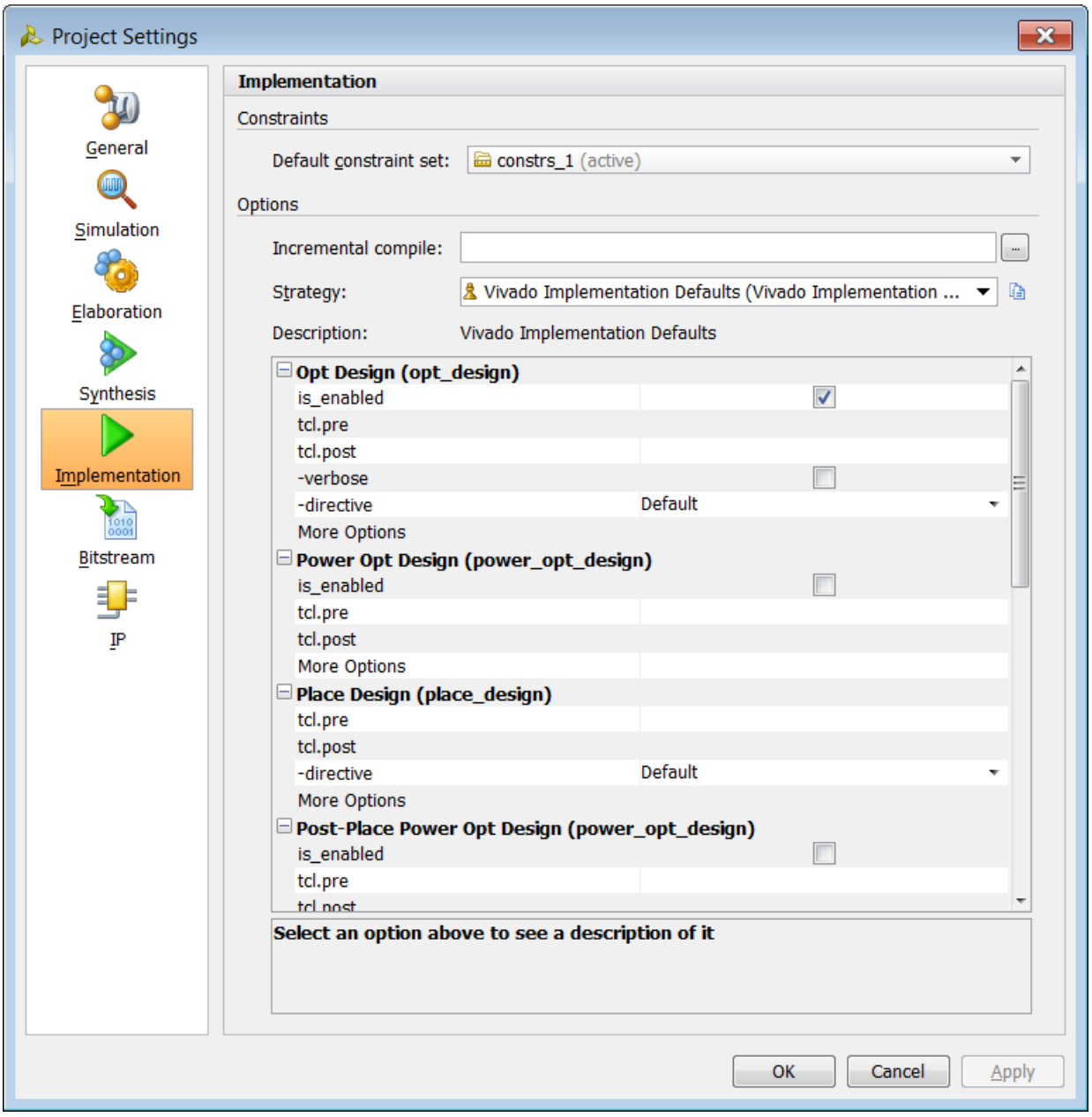

Figure 22: Implementation Settings

3. Click **Cancel** to close the Project Settings dialog box.

You are now ready to launch Vivado synthesis and implementation.

Step 7: Synthesizing and Implementing the Design

After configuring the synthesis and implementation run options, you can:

- Use the **Run Synthesis** command to run only synthesis.
- Use the **Run Implementation** command, which will first run synthesis if it has not been run and then run implementation.
- Use the **Generate Bitstream** command, which will first run synthesis, then run implementation if they have not been run, and then write the bitstream for programming the Xilinx device.

For this tutorial, we will run these steps one at a time.

1. In the Flow Navigator, click the **Run Synthesis** button and wait for the task to complete.

   Notice the progress bar in the upper-right corner of the Vivado IDE, indicating the run is in progress. Vivado launches the synthesis engine in a background process to free up the tool for other actions. While the synthesis process is running in the background, you can continue browsing Vivado IDE windows, run reports, and further evaluate the design. You will notice that the Log window displays the synthesis log at the bottom of the IDE. This is also available through the Reports window.

   After synthesis has completed, the Synthesis Completed dialog box prompts you to choose the next step.

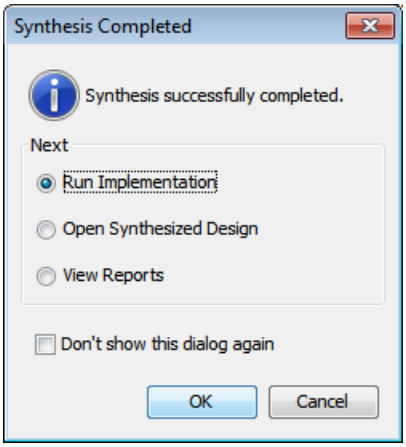

Figure 23: Synthesis Complete

2. Select **Run Implementation**, and click **OK**.

   The implementation process is launched, and placed into a background process after some initialization.

The next step in this tutorial shows you how to perform design analysis of the synthesized design while waiting for implementation to complete.

Step 8: Analyzing the Synthesized Design

Opening the synthesized design enables design analysis, timing constraint definition, I/O planning, floorplanning and debug core insertion. These features are covered in other tutorials, but you can take a quick look in this step.

1. While implementation is running, select **Open Synthesized Design** in the Flow Navigator and wait for the design to load.

   Notice that as the Vivado IDE opens the synthesized design, the implementation continues running in the background. At some point while you are exploring the synthesized design, implementation will complete, and the Implementation Completed dialog box prompts you to choose the next step.

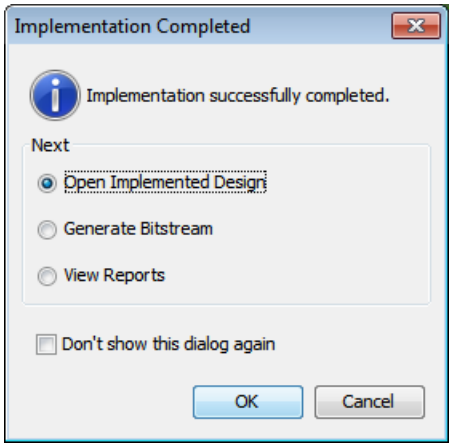

Figure 24: Implementation Complete

2. Click **Cancel** to close the dialog without taking any action.

   This leaves the synthesized design open. You will open the implemented design after you are finished examining the features of the synthesized design.
3. Ensure that the Layout Selector pull-down menu in the main Toolbar has **Default Layout** selected.
4. Click the **Reports** tab at the bottom of Vivado IDE.

   If the Reports window is not open, you can open it with **Windows > Reports**.
5. Double-click **Vivado Synthesis Report** to examine the report.
6. Double-click **Utilization Report** to examine the report.
7. **Close all reports** when you have finished examining them.
8. Click the Messages tab at the bottom of the Vivado IDE.

   If the Messages window is not open, you can open it with **Windows > Messages**.

The Messages window provides message type filters in its banner that display or hide Error, Critical Warning, Warning, Info, and Status messages.

9. Click the **Collapse All** button to condense all of the Messages.
10. Expand the Synthesis messages.
11. Scroll through the Synthesis messages and notice the links to specific lines within source files. Click some of the links and notice the source file opens in the Text Editor with the appropriate line highlighted.

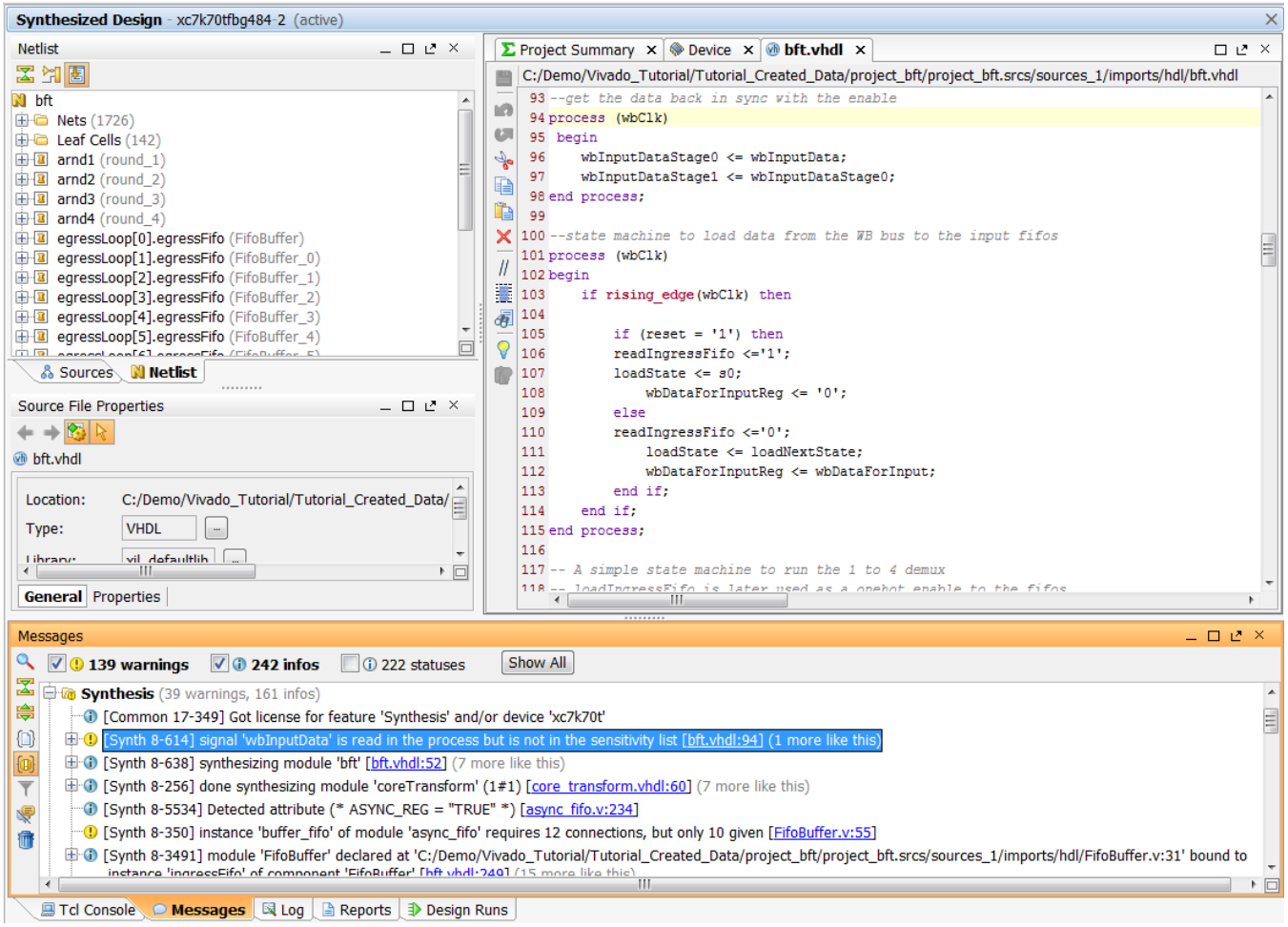

Figure 25: Synthesis Messages Linked to Source Files

12. In the Flow Navigator, under Synthesized Design, select **Report Timing Summary**.

    The Report Timing Summary dialog box opens. Examine the various fields and options of this command.

13. Click **OK** to run with default options.

    The Timing Summary Results window opens.

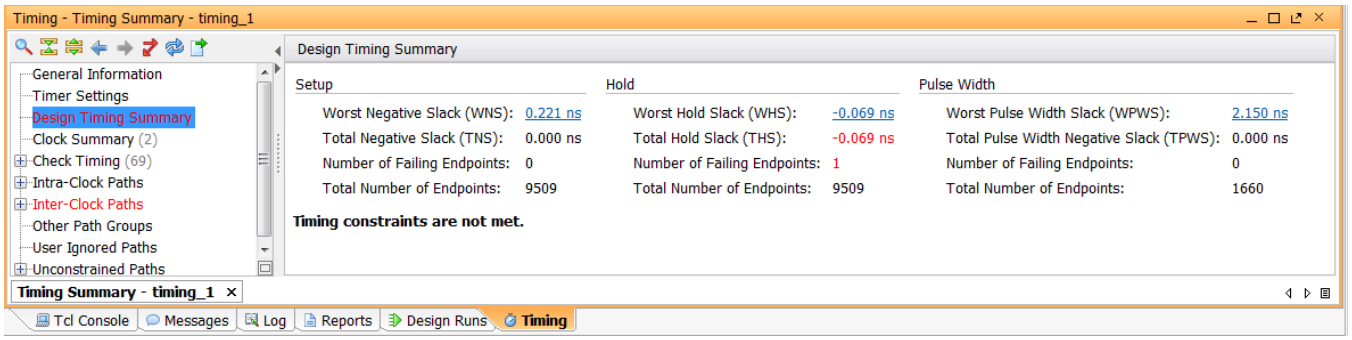

Figure 26: Report Timing Summary

Examine the Timing Summary Results window showing timing estimates prior to implementation. Click on some of the reporting categories in the tree on the left side of the Timing Summary Results window.

14. Select **Report Power** in the Flow Navigator.

    The Report Power dialog box opens. Examine the various fields and options of this command.

15. Click **OK** to run with default options.

    The Power Results window opens. Examine the Power Results window showing power estimates prior to implementation. The report is dynamic, with tooltips providing details of the specific sections of the report when you move the mouse over the report, as shown in Figure 27.

16. Click some of the reporting categories in the tree on the left side of the Power Results window to examine the different information presented.

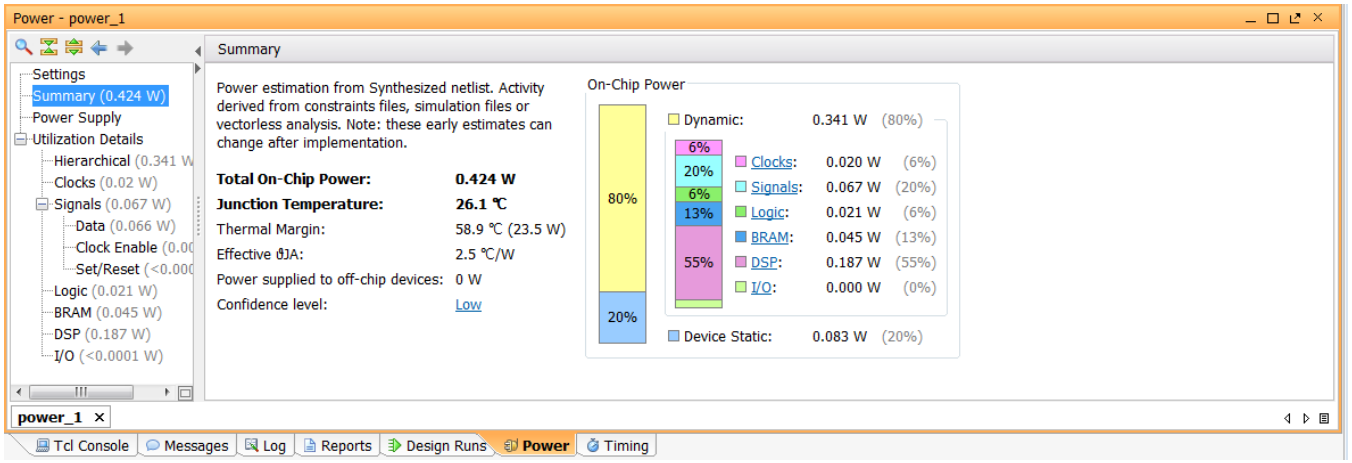

Figure 27: Report Power

17. Close the Report Timing Summary window, the Power Report window, and any open Text Editor windows.

Step 9: Analyzing the Implemented Design

The Vivado IDE is interactive, enabling editing of design constraints and netlists on the in-memory design. When you save the design, constraint changes are written back to the original source XDC files. Alternatively, you can save the changes to a new constraints file to preserve the original constraints. This flexibility supports exploration of alternate timing and physical constraints, including floorplanning, while keeping the original source files intact.

Opening the Implemented Design

1. Select **Open Implemented Design** in the Flow Navigator.
2. If prompted, select **Yes** to close the synthesized design and **Don't Save**, if prompted.

   After the Implemented Design has loaded, you can see the implementation results in the Device window.
3. Click on the **Reports** tab at the bottom of the Vivado IDE.

   If the Reports window is not open, you can open it with **Windows > Reports**. Select and examine some of the reports from Place Design and Route Design. Close each of the reports when you are done.
4. Select the **Messages** tab at the bottom of the IDE.

   If the Messages window is not open, you can open it with **Windows > Messages**.
5. Click the Collapse All button to condense all of the Messages.
6. Expand the **Implementation** folder
7. View the messages from Design Initialization, Opt_Design, Place_Design, and Route_Design.

Analyzing Routing

After the design has been placed and routed, you can generate a timing report to verify that all the timing constraints are met. You can select paths from the Timing Report window to examine the routed path in the Device window. If there are timing problems, you can revisit the RTL source files or design constraints to address any problems.

1. In the Device window, select the **Routing Resources** button to display the device routing.

   This lets you see the routed connection in the Device window. Though you will need to zoom closely into the device to see elements of the route, a zoomed-out view lets you see the route in its entirety.
2. Select the **Auto Fit Selection** button in the Device window toolbar menu to enable the Vivado IDE to automatically zoom into and center the selected objects.

3. On the left side pane of the Timing Summary Results window, select:

   Intra-Clock Paths > wbClk > HOLD

4. In the table view on the right side of the Timing Summary Report window, click any timing path to select it and highlight it in the Device window. Select various paths in the Timing Summary window and examine the path routing.

5. On the left side pane of the Timing Summary Results window, select:
   Intra-Clock Paths > wbClk > SETUP

6. Click any path in the table view on the right side of the Timing Summary Results window to select it and highlight it in the Device window. Select various paths in the Timing Summary Results window and examine the path routing.

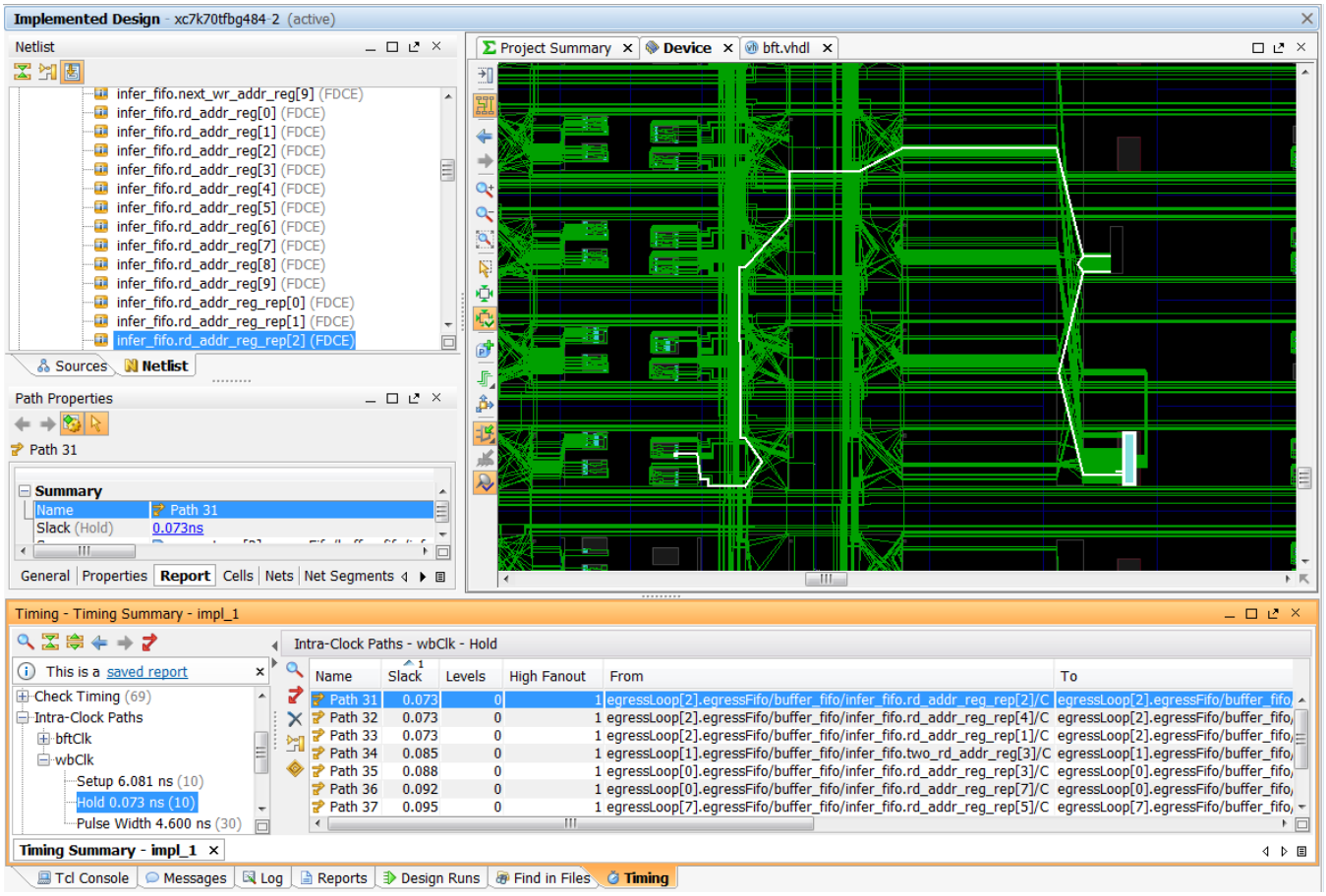

Figure 28: Examine Routing for Timing Paths

7. With a timing path selected, right-click to open the popup menu in either the Device view or the Timing Summary window, and select the **Schematic** command.

   Note: *Alternatively, you can press the **F4** function key to open the Schematic window.*

   The Schematic window opens, displaying the schematic for the selected timing path. Use commands from the popup menu in the Schematic window, such as Expand or Collapse Outside, or Expand Cone, to traverse the schematic and examine the logic cells on the timing path.

8. Close the Schematic window.

9. In the Device window, select the **Routing Resources** toolbar button again to turn *off* routing resources.

The Device window now displays just the placed instances, without the added details of the routed connections.

Step 10: Generating a Bitstream file

With IOSTANDARD constraints defined for all of the I/O ports, and the logic of the design placed with assigned LOCs, you can generate a bitstream. Before launching Write Bitstream, you will review the settings for this command.

1. In the Flow Navigator, select **Bitstream Settings** under Program and Debug.

   The Project Settings dialog box opens. The Bitstream Settings provides you access to the options available for the `write_bitstream` command. For a complete description of these options and how to use them, see this link in the *Vivado Design Suite User Guide: Programming and Debugging* (UG908).

   Note: *There is a **Configure additional bitstream settings** link at the top of the Project Settings dialog box. Click this to open the Edit Device Properties dialog box and configure general bitstream properties.*

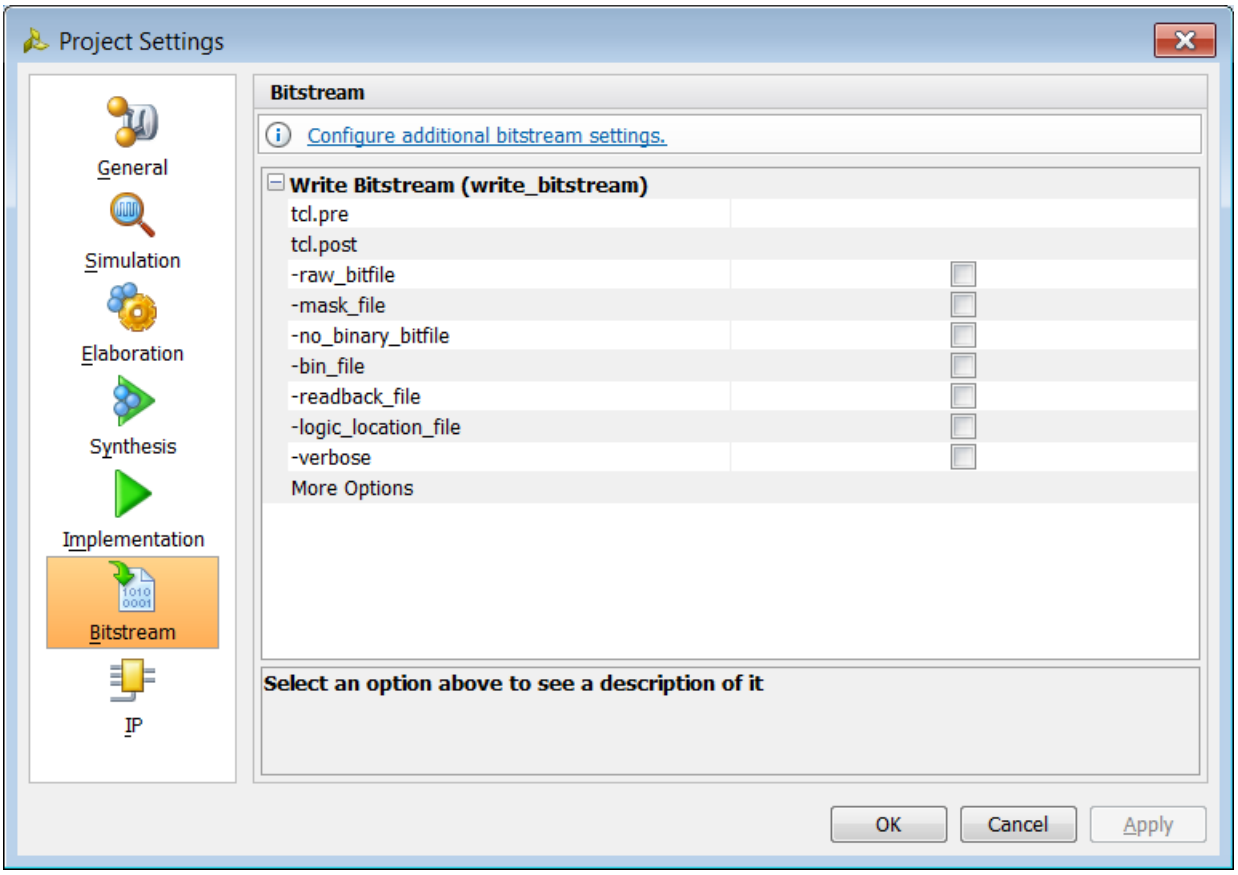

Figure 29: Bitstream Settings

2. Click **Cancel** to close the Project Settings dialog box.
3. In the Flow Navigator, under the Program and Debug section, click **Generate Bitstream**.
4. If prompted, click **Don't Save**.
5. After the bitstream generates, click **OK** in the Bitstream Generation Completed dialog box to view the reports from the command.

Conclusion

After completing this tutorial, you should be able to do the following:

- Use Project mode and Non-Project mode.
- Create an RTL project in Vivado IDE.
- Configure the Vivado synthesis, simulation, and implementation tools.
- Launch Vivado simulator, synthesis and implementation.
- Apply constraints to the synthesized design.
- Generate timing and power reports.
- Examine routing results in the Device editor.
- Generate a bitstream file.
- Switch between the Vivado Design Suite Tcl shell and the Vivado IDE.

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 http://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 http://www.xilinx.com/legal.htm#tos.

© Copyright 2012-2015 Xilinx, Inc. Xilinx, the Xilinx logo, Artix, ISE, Kintex, Spartan, Virtex, Zynq, UtlraScale, 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.