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 2012-2014 Xilinx, Inc. Xilinx, the Xilinx logo, Artix, 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

The following table shows the revision history for this document.

<table>
<thead>
<tr>
<th>Date</th>
<th>Version</th>
<th>Revision</th>
</tr>
</thead>
</table>
# Table of Contents

Revision History ................................................................................................................. 2

Chapter 1 Tutorial Description ......................................................................................... 6
  Overview .................................................................................................................. 6
  Software Requirements ......................................................................................... 7
  Hardware Requirements ....................................................................................... 7
  Locating the Tutorial Design Files ........................................................................ 8
  Preparing the Tutorial Design Files ..................................................................... 8

Chapter 2 High-Level Synthesis Introductory Tutorial ................................................. 9
  Overview .................................................................................................................. 9
  Tutorial Design Description ..................................................................................... 9
  HLS Lab 1: Creating a High-Level Synthesis Project ........................................ 10
  HLS: Lab 2: Using the Tcl Command Interface .................................................. 26
  HLS: Lab 3: Using Solutions for Design Optimization ...................................... 30

Chapter 3 C Validation ................................................................................................. 42
  Overview .................................................................................................................. 42
  Tutorial Design Description ..................................................................................... 42
  Lab 1: C Validation and Debug .............................................................................. 43
  Lab 2: C Validation with ANSI C Arbitrary Precision Types ............................. 51
  Lab 3: C Validation with C++ Arbitrary Precision Types ..................................... 56

Chapter 4 Interface Synthesis ...................................................................................... 61
  Overview .................................................................................................................. 61
  Tutorial Design Description ..................................................................................... 61
  Interface Synthesis Lab 1: Block-Level I/O protocols ....................................... 62
  Interface Synthesis Lab 2: Port I/O protocols ...................................................... 70
  Interface Synthesis Lab 3: Implementing Arrays as RTL Interfaces .................. 75
  Interface Synthesis Lab 4: Implementing AXI4 Interfaces .................................. 90

Chapter 5 Arbitrary Precision Types .......................................................................... 100
Chapter 1  Tutorial Description

Overview

This Vivado® tutorial is a collection of smaller tutorials that explain and demonstrate all steps in the process of transforming C, C++ and SystemC code to an RTL implementation using High-Level Synthesis. TH: Sample Paste: Using The binding process. The tutorial shows how you create an initial RTL implementation and then you transform it into both a low-area and high-throughput implementation by using optimization directives without changing the C code.

High-Level Synthesis Introduction

This tutorial introduces Vivado High-Level Synthesis (HLS). You can learn the primary tasks for performing High-Level Synthesis using both the Graphical User Interface (GUI) and Tcl environments.

The tutorial shows how you create an initial RTL implementation and then you transform it into both a low-area and high-throughput implementation by using optimization directives without changing the C code.

C Validation

This tutorial reviews the aspects of a good C test bench and demonstrates the basic operations of the Vivado High-Level Synthesis C debug environment. The tutorial also shows how to debug arbitrary precision data types.

Interface Synthesis

The interface synthesis tutorial reviews all aspect of creating ports for the RTL design. You can learn how to control block-level I/O port protocols and port I/O protocols, how arrays in the C function can be implemented as multiple ports and types of interface protocol (RAM, FIFO, AXI4 Stream), and how AXI4 bus interfaces are implemented.

The tutorial completes with a design example in which the I/O accesses and the logic are optimized together to create an optimal implementation of the design.

Arbitrary Precision Types

The lab exercises in this tutorial contrast a C design written in native C types with the same design written with Vivado High-Level Synthesis arbitrary precision types, showing how the latter improves the quality of the hardware results without sacrificing accuracy.

Design Analysis

This tutorial uses a DCT function to explain the features of the interactive design analysis features in Vivado High-Level Synthesis. The initial design takes you through a number of
analysis and optimization stages that highlight all the features of the analysis perspective and provide the basis for a design optimization methodology.

**Design Optimization**

Using a matrix multiplier example, this tutorial reviews two-design optimization techniques. The first lab explains how a design can be pipelined, contrasting the approach of pipelining the loops versus pipelining the functions.

The tutorial shows you how to use the insights learned from analyzing to update the initial C code and create a more optimal implementation of the design.

**RTL Verification**

This tutorial shows how you can use the RTL cosimulation feature to verify automatically the RTL created by synthesis. The tutorial demonstrates the importance of the C test bench and shows you how to use the output from RTL verification to view the waveform diagrams in the Vivado and Mentor Graphics ModelSim simulators.

**Using HLS IP in IP Integrator**

This tutorial shows how RTL designs created by High-Level Synthesis are packaged as IP, added to the Vivado IP Catalog, and used inside the Vivado Design Suite.

**Using HLS IP in a Zynq Processor Design**

In addition to using an HLS IP block in a Zynq®-7000 SoC design, this tutorial shows how the C driver files created by High-Level Synthesis are incorporated into the software on the Zynq Processing System (PS).

**Using HLS IP in System Generator for DSP**

This tutorial shows how RTL designs created by High-Level Synthesis can be packaged as IP and used inside System Generator for DSP.

---

**Software Requirements**

This tutorial requires that the Vivado Design Suite 2014.1 release or later is installed.

---

**Hardware Requirements**

Xilinx recommends a minimum of 2 GB of RAM when using the Vivado tools.
Locating the Tutorial Design Files

As shown in Figure 1, designs for the tutorial exercises are available as a zipped archive on the Xilinx Website, tutorial documentation page.

**IMPORTANT:** All the tutorial examples for Vivado High-Level Synthesis are available for download at:


---

Preparing the Tutorial Design Files

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

This tutorial assumes that you have placed the unzipped design files in the location C:\Vivado_HLS_Tutorial.

**IMPORTANT:** If the Vivado_HLS_Tutorial directory is unzipped to a different location, or if it resides on Linux, adjust the pathnames to the location at which you have placed the Vivado_HLS_Tutorial directory.
Overview

This tutorial introduces Vivado® High-Level Synthesis (HLS). You can learn the primary tasks for performing High-Level Synthesis using both the Graphical User Interface (GUI) and Tcl environments.

The tutorial shows how use of optimization directives transforms an initial RTL implementation into both a low-area and high-throughput implementation.

Lab 1

Explains how to:
• Set up a High-Level Synthesis (HLS) project
• Perform all major steps in the HLS design flow:
  o Validate the C code
  o Create and synthesize a solution
  o Verify the RTL and package the IP.

Lab 2

Demonstrates how to use the Tcl interface.

Lab 3

Shows you how to optimize the design using optimization directives. This lab creates multiple versions of the RTL implementation and compares the different solutions.

Tutorial Design Description

To obtain the tutorial design file, refer to the section
Obtaining the Tutorial Designs.

This tutorial uses the design files in the tutorial directory `Vivado_HLS_Tutorial\Introduction`.

The sample design used in this tutorial is a FIR filter. The hardware goals for this FIR design project are:

- Create a version of this design with the highest throughput

The final design must process data supplied with an input valid signal and produce output data accompanied by an output valid signal. The filter coefficients are to be stored externally to the FIR design, in a single port RAM.

HLS Lab 1: Creating a High-Level Synthesis Project

Introduction

This lab shows how to create a High-Level Synthesis project, validate the C code, synthesize the design to RTL, and verify the RTL.

**IMPORTANT:** The figures and commands in this tutorial assume the tutorial data directory `Vivado_HLS_Tutorial` files are unzipped and placed in the location `C:\Vivado_HLS_Tutorial`.

Step 1: Creating a New Project

1. Open the Vivado® HLS Graphical User Interface (GUI):
   - On Windows systems, open Vivado HLS by double-clicking the **Vivado HLS 2014.1** desktop icon.
   - On Linux systems, type `vivado_hls` at the command prompt.
Vivado HLS opens with the Welcome Screen as shown in Figure 3. If any projects were previously opened, they are shown in the Recent Project pane, otherwise this window is not shown in the Welcome screen.

![Vivado HLS Welcome Page](image)

2. In the Welcome Page, select **Create New Project** to open the Project wizard.

3. As shown in Figure 4:
   a. Enter the project name `fir_prj`.
   b. Click **Browse** to navigate to the location of the `lab1` directory.
   c. Select the `lab1` directory and click **OK**.
   d. Click **Next**.
This information defines the name and location of the Vivado HLS project directory. In this case, the project directory is `fir_prj` and it resides in the `lab1` folder.

4. Enter the following information to specify the C design files:
   a. Specify `fir` as the top-level function.
   b. Click Add Files.
   c. Select `fir.c` and click Open.
   d. Click Next.
Figure 5: Project Design Files
**IMPORTANT:** In this lab there is only one C design file. When there are multiple C files to be synthesized, you must add all of them to the project at this stage.

Any header files that exist in the local directory `lab1` are automatically included in the project. If the header resides in a different location, use the **Edit CFLAGS** button to add the standard gcc/g++ search path information (for example, `-I<path_to_header_file_dir>`).

**Figure 6** shows the input window for specifying the test bench files. The test bench and all files used by the test bench (except header files) must be included. You can add files one at a time, or select multiple files to add using the Ctrl and Shift keys.

5. Click the **Add Files** button to include both test bench files: `fir_test.c` and `out.gold.dat`.

6. Click **Next**.
Both C simulation (and RTL cosimulation) execute in sub-directories of the solution. If you do not include all the files used by the test bench (for example, data files read by the test bench, such as `out.gold.dat`), C and RTL simulation might fail after synthesis due to an inability to find the data files.

The Solution Configuration window (shown in Figure 7) specifies the technical specifications of the first solution.

A project can have multiple solutions, each using a different target technology, package, constraints, and/or synthesis directives.

![Figure 7: Solution Configuration](image)

7. Accept the default solution name (`solution1`), clock period (**10 ns**) and clock uncertainty (defaults to 12.5% of the clock period, when left blank/undefined).

8. Click the part selection button ![part selection button](image) to open the part selection window.

9. Select **Device xc7k160tfbg484-2** from the list of available devices. Select the following from the dropdown filters to help refine the parts list:
High-Level Synthesis Introductory Tutorial

a. Product Category: **General Purpose**
b. Family: **Kintex®-7**
c. Sub-Family: **Kintex-7**
d. Package: **fbg484**
e. Speed Grade: -2
f. Temp Grade: **All**

10. Click **OK**.

In the Solution Configuration dialog box (shown in Figure 7, above), the selected part name now appears under the Part Selection heading.

11. Click **Finish** to open the Vivado HLS project, as shown in **Figure 8**.

![Figure 8: Vivado HLS Project](image)

- The project name appears on the top line of the Explorer window.
- A Vivado HLS project arranges data in a hierarchical form.
- The project holds information on the design source, test bench, and solutions.
- The solution holds information on the target technology, design directives, and results.
• There can be multiple solutions within a project, and each solution is an implementation of the same source code.

**TIP:** At any time, you can change project or solution settings using the corresponding Project Settings and/or Solution Settings buttons in the toolbar.

**Understanding the Graphical User Interface (GUI)**

Before proceeding, review the regions in the Graphical User Interface (GUI) and their functions. Figure 9 shows an overview of the regions, and each is described below.

**Explorer Pane**

Shows the project hierarchy. As you proceed through the validation, synthesis, verification, and IP packaging steps, sub-folders with the results of each step are created automatically inside the solution directory (named `csim`, `syn`, `sim`, and `impl` respectively).

When you create new solutions, they appear inside the project hierarchy alongside solution1.
Information Pane

Shows the contents of any files opened from the Explorer pane. When operations complete, the report file opens automatically in this pane.

Auxiliary Pane

Cross-links with the Information pane. The information shown in this pane dynamically adjusts, depending on the file open in the Information pane.

Console Pane

Shows the messages produced when Vivado HLS runs. Errors and warnings appear in Console pane tabs.

Toolbar Buttons

You can perform the most common operations using the Toolbar buttons.

When you hold the cursor over the button, a popup dialog box opens, explaining the function. Each button also has an associated menu item available from the pulldown menus.

Perspectives

The perspectives provide convenient ways to adjust the windows within the Vivado HLS GUI.

- **Synthesis Perspective**
  
The default perspective allows you to synthesize designs, run simulations, and package the IP.

- **Debug Perspective**
  
Includes panes associated with debugging the C code. You can open the Debug Perspective after the C code compiles (unless you use the Optimizing Compile mode as this disable debug information).

- **Analysis Perspective**
  
Windows in this perspective are configured to support analysis of synthesis results. You can use the Analysis Perspective only after synthesis completes.

Step 2: Validate the C Source Code

The first step in an HLS project is to confirm that the C code is correct. This process is called C Validation or C Simulation.

In this project, the test bench compares the output data from the *fir* function with known good values.

1. Expand the **Test Bench** folder in the Explorer pane.
2. Double-click the file `fir_test.c` to view it in the Information pane.
3. In the Auxiliary pane, select `main()` in the Outline tab to jump directly to the `main()` function.
Figure 10 shows the result of these actions

The test bench file, fir_test.c, contains the top-level C function main(), which in turn calls the function to be synthesized (fir). A useful characteristic of this test bench is that it is self-checking:

- The test bench saves the output from the fir function into the output file, out.dat.
- The output file is compared with the golden results, stored in file out.gold.dat.
- If the output matches the golden data, a message confirms that the results are correct, and the return value of the test bench main() function is set to 0.
- If the output is different from the golden results, a message indicates this, and the return value of main() is set to 1.

The Vivado HLS tool can reuse the C test bench to perform verification of the RTL.

If the test bench has the previously described self-checking characteristics, the RTL results are automatically checked during RTL verification. Vivado HLS re-uses the test bench during RTL verification and confirms the successful verification of the RTL if the test bench returns a value of 0. If any other value is returned by main(), including no return value, it indicates that the RTL verification failed. There is no requirement to create an RTL test bench. This provides a robust and productive verification methodology.

4. Click the Run C Simulation button, or use menu Project > Run C Simulation, to compile and execute the C design.

5. In the C Simulation dialog box, click OK.
The Console pane (Figure 11) confirms the simulation executed successfully.

![Figure 11: Results of C Simulation](image)

**TIP:** If the C simulation failed, select the **Debug** option in the C Simulation dialog box, compile the design, and automatically switch to the Debug perspective. There you can use a C debugger to fix any problems.

The C Validation tutorial module provides more details on using the Debug environment.

The design is now ready for synthesis.

### Step 3: High-Level Synthesis

In this step, you synthesize the C design into an RTL design and review the synthesis report.

1. Click the **Run C Synthesis** toolbar button or use the menu **Solution > Run C Synthesis**.

When synthesis completes, the report file opens automatically. Because the synthesis report is open in the Information pane, the Outline tab in the Auxiliary pane automatically updates to reflect the report information.

2. Click **Performance Estimate** in the Outline tab (Figure 12).

3. In the Details section of the Performance Estimates, expand the **Loop** view.
In the Performance Estimates pane, shown in **Figure 12**, you can see that the clock period is set to 10 ns. Vivado HLS targets a clock period of Clock Target minus Clock Uncertainty (10.00 - 1.25 = 8.75 ns in this example).

The clock uncertainty ensures there is some timing margin available for the (at this stage) unknown net delays due to place and routing.

The estimated clock period (worst-case delay) is 8.43 ns.

In the Summary section, you can see:

- The design has a latency of 78-clock cycles: it takes 78 clocks to output the results.
- The interval is 79 clock cycles: the next set of inputs is read after 79 clocks. This is one cycle after the final output is written. This indicates the design is not pipelined. The next execution of this function (or next transaction) can only start when the current transaction completes.
- The message “design is not pipelined” is also included under the pipelined type: no pipelining is performed.

The Details section shows:

- There are no sub-blocks in this design. Expanding the Instance section shows no sub-modules in the hierarchy.
- All the delay is due to the RTL logic synthesized from the loop named Shift_Accum_Loop. This logic executes 11 times (Trip Count). Each execution requires 7
clock cycles (Iteration Latency), for a total of 88 clock cycles, to execute all iterations of the logic synthesized from this loop (Latency).

- The total latency is one clock cycle greater than the loop latency. It requires one clock cycle to enter and exit the loop (in this case, the design finishes when the loop finishes, so there is no exit cycle).

4. In the Outline tab, click **Utilization Estimate** (Figure 13).

![Figure 13: Utilization Estimates](image)

5. In the **Details** section of the Utilization Estimates, expand the Instance view.

   The design uses a single memory implemented as LUTRAM (since it contains less than 1024 elements), 4 DSP48s, and approximately 200 flip-flops and LUTs. At this stage, the area numbers are estimates.

   - RTL synthesis might be able to perform additional optimizations, and these figures might change after RTL synthesis.
   - The number of DSP48s seems larger than expected for a FIR filter. This is because the data is a C integer type, which is 32-bit. It requires more than 1 DSP48 to multiply 32-bit data values.
   - The multiplier instance shown in the Instance view accounts for all the DSP48s.
   - The multiplier is a pipelined multiplier. It appears in the Instance section indicating it is a sub-block. Standard combinational multipliers have no hierarchy, and listed in the Expressions section (indicating a component at this level of hierarchy).
In **HLS**: Lab 3: Using Solutions for Design Optimization, you optimize this design.

6. In the Outline tab, click **Interface** *(Figure 14).*

![Figure 14: Interface Report](image)

The Interface section shows the ports and I/O protocols created by interface synthesis:

- The design has a clock and reset port (ap_clk and ap_reset). These are associated with the Source Object fir: the design itself.

- There are additional ports associated with the design as Source Object. Synthesis has automatically added some block level control ports: ap_start, ap_done, ap_idle and ap_ready.

- The **Interface Synthesis** tutorial provides more information about these ports.

- The function output y is now a 32-bit data port with an associated output valid signal indicator y_ap_vld.

- Function input argument c (an array) has been implemented as a block RAM interface with a 4-bit output address port, an output CE port and a 32-bit input data port.

- Finally, input argument x is simply implemented as a data port with no I/O protocol (ap_none).

Later in this tutorial, **HLS**: Lab 3: Using Solutions for Design Optimization explains how to optimize the I/O protocol for port x.

**Step 4: RTL Verification**

High-Level Synthesis can re-use the C test bench to verify the RTL using simulation.

1. Click the **Run C/RTL Cosimulation** toolbar button or use the menu **Solution > Run C/RTL Cosimulation**.

2. Click **OK** in the C/RTL Co-simulation dialog box to execute the RTL simulation.
The default option for RTL Co-simulation is to perform the simulation using the Vivado simulator and Verilog RTL. To perform the verification using a different simulator, VHDL or SystemC RTL use the options in the C/RTL Co-simulation dialog box.

When RTL co-simulation completes, the report opens automatically in the Information pane, and the Console displays the message shown in Figure 15. This is the same message produced at the end of C simulation.

- The C test bench generates input vectors for the RTL design.
- The RTL design is simulated.
- The output vectors from the RTL are applied back into the C test bench and the results-checking in the test bench verify whether or not the results are correct.

![Figure 15: RTL Verification Results](image)

The RTL Verification tutorial (page 168) provides additional information.

**Step 5: IP Creation**

The final step in the High-Level Synthesis flow is to package the design as an IP block for use with other tools in the Xilinx Design Suite.

1. Click the Export RTL toolbar button or use the menu Solution > Export RTL.
2. Ensure the Format Selection dropdown menu shows IP Catalog.
3. Click OK.

The IP packager creates a package for the Vivado IP Catalog. (Other options available from the drop-down menu allow you to create IP packages for System Generator for DSP, a Synthesized Checkpoint format for Vivado or a Pcore for Xilinx Platform Studio.)

4. Expand Solution1 in the Explorer.
5. Expand the impl folder created by the Export RTL command.
6. Expand the ip folder and find the IP packaged as a zip file, ready for adding to the Vivado IP Catalog (Figure 16).
Also note, in Figure 16, that if you expand the Verilog or VHDL folders inside the impl folder, there is a Vivado project ready for opening in the Vivado Design Suite.

**RECOMMENDED:** In this Vivado project, the HLS design is the top-level. This project provides an additional means of analyzing the design. The recommended approach is to add the IP package to the Vivado IP catalog, and add it as IP to the design that uses the HLS design.

**Note:** There is no project file created for devices synthesized by ISE (6 series or earlier devices).

At this stage, leave the Vivado HLS GUI open. You will return to this in the next lab exercise.
HLS: Lab 2: Using the Tcl Command Interface

Introduction

This lab exercise shows how to create a Tcl command file based on an existing Vivado HLS project and use the Tcl interface.

Step 1: Create a Tcl file

1. Open the Vivado HLS Command Prompt.
2. On Windows, use **Start > All Programs > Xilinx Design Tools > Vivado 2014.1 > Vivado HLS > Vivado HLS 2014.1** Command Prompt (**Figure 17**).
3. On Linux, open a new shell.

![Figure 17: The Vivado HLS Command Prompt](image)

When you create a Vivado HLS project, Tcl files are automatically saved in the project hierarchy. In the GUI still open from Lab 1, a review of the project shows two Tcl files in the project hierarchy (**Figure 18**).

4. In the GUI, still open from Lab 1, expand the Constraints folder in solution1 and double-click the file `script.tcl` to view it in the Information pane.
Figure 18: The Vivado HLS Project Tcl Files

- The file `script.tcl` contains the Tcl commands to create a project with the files specified during the project setup and run synthesis.
- The file `directives.tcl` contains any optimizations applied to the design. No optimization directives were used in Lab 1 so this file is empty.

In this lab exercise, you use the `script.tcl` from Lab 1 to create a Tcl file for the Lab 2 project.

5. Close the Vivado HLS GUI from Lab 1. This is project no longer needed.

6. In the Vivado HLS Command Prompt, use the following commands (also shown in Figure 19) to create a new Tcl file for Lab 2.
   a. Change directory to the Introduction tutorial directory
      C:\Vivado_HLS_Tutorial\Introduction.
   b. Use the command `cp lab1\fir_prj\solution1\script.tcl lab2\run_hls.tcl` to copy the existing Tcl file to Lab 2. (The Windows command prompt supports auto-completion using the Tab key: press the tab key repeatedly to see new selections).
   c. Use the command `cd lab2` to change into the `lab2` directory.
d. Using any text editor, perform the following edits to the file `run_hls.tcl` in the `lab2` directory. The final edits are shown in Figure 20.

i. Add a `–reset` option to the `open_project` command. Because you typically run Tcl files repeatedly on the same project, it is best to overwrite any existing project information.

ii. Add a `–reset` option to the `open_solution` command. This removes any existing solution information when the Tcl file is re-run on the same solution.

iii. Delete the source command. If a previous project contains any directives you wish to re-use, you can copy the `directives.tcl` file from that project to a local path, or you can copy the directives directly into this file.

iv. Add the `exit` command.

v. Save the file.
You can run the Vivado HLS in batch mode using this Tcl file.

e. In the Vivado HLS Command Prompt window, type `vivado_hls -f run_hls.tcl`.

Vivado HLS executes all the steps covered in lab1. When finished, the results are available inside the project directory `fir_prj`.

- The synthesis report is available in `fir_prj\solution1\syn\report`.
- The simulation results are available in `fir_prj\solution\sim\report`.
- The output package is available in `fir_prj\solution1\impl\ip`.
- The final output RTL is available in `fir_prj\solution1\impl` and then Verilog or VHDL.

**CAUTION!** When copying the RTL results from a Vivado HLS project, you must use the RTL from the `impl` directory.

For designs using floating-point operators or AXI4 interfaces, the RTL files in the `syn` directory are only the output from synthesis. Additional processing is performed by Vivado HLS during `export_design` before you can use this RTL in other design tools.
HLS: Lab 3: Using Solutions for Design Optimization

Introduction

This lab exercise uses the design from Lab 1 and optimizes it.

Step 1: Creating a New Project

1. Open the Vivado HLS Command Prompt.
   a. On Windows, use Start > All Programs > Xilinx Design Tools > Vivado 2014.1 > Vivado HLS > Vivado HLS 2014.1 Command Prompt
   b. On Linux, open a new shell.
2. Change to the Lab 3 directory: cd C:\Vivado_HLS_Tutorial\Introduction\lab3.
3. In the command prompt window, type: vivado_hls –f run_hls.tcl
   This sets up the project.
4. In the command prompt window, type vivado_hls -p fir_prj to open the project in the Vivado HLS GUI.
   Vivado HLS opens, as shown in Figure 21, with the synthesis for solution1 already complete.

Figure 21: Introduction Lab 3 Initial Solution
As stated earlier, the design goals for this design are:

- Create a version of this design with the highest throughput
- The final design should be able to process data supplied with an input valid signal.
- Produce output data accompanied by an output valid signal.
- The filter coefficients are to be stored externally to the FIR design, in a single-port RAM.

**Step 2: Optimize the I/O Interfaces**

Because the design specification includes I/O protocols, the first optimization you perform creates the correct I/O protocol and ports. The type of I/O protocol you select might affect what design optimizations are possible. If there is an I/O protocol requirement, you should set the I/O protocol as early as possible in the design cycle.

You reviewed the I/O protocol for this design in Lab 1 ([Figure 14](#)), and you can review the synthesis report again by navigating to the report folder inside the `solution1\syn` folder. The I/O requirements are:

- Port C must have a single-port RAM access.
- Port X must have an input data valid signal.
- Port Y must have an output data valid signal.

Port C already is a single-port RAM access. However, if you do not explicitly specify the RAM access type, High-Level Synthesis might use a dual-port interface. HLS takes this action if doing so creates a design with a higher throughput. If a single-port is required, you should explicitly add to the design the I/O protocol requirement to use a single-port RAM.

Input port X is by default a simple 32-bit data port. You can implement it as an input data port with an associated data valid signal by specifying the I/O protocol `ap_vld`.

Output port Y already has an associated output valid signal. This is the default for pointer arguments. You do not have to specify an explicit port protocol for this port, since the default implementation is what is required, but if it is a requirement, it is a good practice to specify it.

To preserve the existing results, create a new solution, `solution2`.

1. Click the **New Solution** toolbar button to create a new solution.
2. Leave the default solution name as `solution2`. Do not change any of the technology or clock settings.
3. Click **Finish**.

This creates `solution2` and set it as the default solution - confirm that `solution2` is highlighted in bold in the Explorer pane, indicating that it is the current active solution.

To add optimization directives to define the desired I/O interfaces to the solution, perform the following steps.

4. In the **Explorer** pane, expand the **Source** container (as shown in [Figure 22](#)).
5. Double-click fir.c to open the file in the Information pane.

6. Activate the **Directives** tab in the Auxiliary pane and select the top-level function fir to jump to the top of the fir function in the source code view (**Figure 22**).

   ![Figure 22: Opening the Directives Tab](image)

The Directives tab, shown on the right side of **Figure 22**, lists all of the objects in the design that can be optimized. In the Directives tab, you can add optimization directives to the design. You can view the Directives tab only when the source code is open in the Information pane.

Apply the optimization directives to the design.

7. In the Directive tab, select the c argument/port (green dot).

8. Right-click and select **Insert Directives**.

9. Implement the single-port RAM interface by performing the following:
   a. Select **RESOURCE** from the Directive drop-down menu.
   b. Click the core box.
   c. Select **RAM_1P_BRAM**, as shown in **Figure 23**.

   The steps above specify that array c be implemented using a single-port block RAM resource. Because array c is in the function argument list, and hence is outside the function, a set of data ports are automatically created to access a single-port block RAM outside the RTL implementation.

Because I/O protocols are unlikely to change, you can add these optimization directives to the source code as pragmas to ensure that the correct I/O protocols are embedded in the design.

10. In the **Destination** section of the Directives Editor, select **Source File**.

11. To apply the directive, click **OK**.
12. Next, specify port x to have an associated valid signal/port.
   a. In the Directives tab, select input port x (green dot).
   b. Right-click and select Insert Directives.
   c. Select Interface from the Directive Editor drop-down menu.
   d. Select Source File from the Destination section of the dialog box
   e. Select ap_vld as the mode.
   f. Click OK to apply the directive.

13. Finally, explicitly specify port y to have an associated valid signal/port.
   a. In the Directives tab, select input port y (green dot).
   b. Right-click and select Insert Directives.
   c. Select Source File from the Destination section of the dialog box
   d. Select Interface from the Directive drop-down menu.
   e. Select ap_vld for the mode.
f. Click **OK** to apply the directive

When complete, verify that the source code and the Directive tab are as shown in Figure 24. Right-click on any incorrect directive to modify it.

14. Click the **Run C Synthesis** toolbar button to synthesize the design.

15. When prompted, click **Yes** to save the contents of the C source file. Adding the directives as pragmas modified the source code.

   When synthesis completes, the report file opens automatically.

16. Click the **Outline** tab to view the Interface results, or simply scroll down to the bottom of the report file.

   **Figure 25** shows the ports now have the correct I/O protocols.
Step 3: Analyze the Results

Before optimizing the design, it is important to understand the current design. It was shown in Lab 1 how the synthesis report can be used to understand the implementation, however, the Analysis perspective provides greater detail in an interactive manner.

While still in solution2, and as shown in Figure 26:

1. Click the Analysis perspective button.

2. Click the Shift_Accum_Loop in the Performance window to expand it.
   - The red-dotted line in Figure 26 is used shortly in an explanation; it is not part of the view.
   - The tutorial Design Analysis provides a more complete understanding of the Analysis perspective, but the following explains what is required to create the smallest and fastest RTL design from this source code.
   - The left column of the Performance pane view shows the operations in this module of the RTL hierarchy.
   - The top row lists the control states in the design. Control states are the internal states High-Level Synthesis uses to schedule operations into clock cycles. There is a close correlation between the control states and the final states in the RTL Finite State Machine (FSM), but there is no one-to-one mapping.
The explanation presented here follows the path of the dotted red line in Figure 26. Some of the objects here correlate directly with the C source code. Right-click the object to cross-reference with the C code.

- The design starts in the first state with a read operation on port x.
- In the next state, it starts to execute the logic created by the for-loop Shift_Accum_Loop. Loops are shown in yellow, and you can expand or collapse them. Holding the cursor over the yellow loop body in this view shows the loop details: 8 cycles, 11 iterations for a total latency of 88.
- In the first state, the loop iteration counter is checked: addition, comparison, and a potential loop exit.
- There is a two-cycle memory read operation on the block RAM synthesized from array data (one cycle to generate the address, one cycle to read the data).
- There are memory reads on the c port.
- A multiplication operations each takes 3 cycles to complete.
- The for-loop is executed 11 times.
- At the end of the final iteration, the loop exits in state c1 and the write to port y occurs.

You can also use the Analysis perspective to analyze the resources used in the design.

3. Click the Resource view, as shown in Figure 27.
4. Expand all the resource groups (also shown in Figure 27).
Figure 27 shows:

- The reads on the ports x and y. Port c is reported in the memory section because this is also a memory port.
- There are two multipliers being used in this design.
- There is a read and write operation on the memory `shift_reg`.
- None of the other resources are being shared because there is only one instance of each operation on each row or clock cycle.

With the insight gained through analysis, you can proceed to optimize the design.

Before concluding the analysis, it is worth commenting on the multi-cycle multiplication operations, which require multiple DSP48s to implement. The source code uses an `int` data-type. This is a 32-bit data-type that results in large multipliers. A DSP48 multiplier is 18-bit and it requires multiple DSP48s to implement a multiplication for data widths greater than 18-bit.

The tutorial **Arbitrary Precision Types** shows how you can create designs with more suitable data types for hardware. Use of arbitrary precision types allows you to define data types of any arbitrary bit size (more than the standard C/C++ 8-, 16-, 32- or 64-bit types).

### Step 4: Optimize for the Highest Throughput (lowest interval)

The two issues that limit the throughput in this design are:
• The \texttt{for} loop. By default loops are kept rolled: one copy of the loop body is synthesized and re-used for each iteration. This ensures each iteration of the loop is executed sequentially. You can unroll the \texttt{for} loop to allow all operations to occur in parallel.

• The block RAM used for \texttt{shift\_reg}. Because the variable \texttt{shift\_reg} is an array in the C source code, it is implemented as a block RAM by default. However, this prevents its implementation as a shift-register. You should therefore partition this block RAM into individual registers.

Begin by creating a new solution.

1. Click the \textbf{New Solution} button.

2. Leave the solution name as \texttt{solution3}.

3. Click \textbf{Finish} to create the new solution.

4. In the Project menu, select \textbf{Close Inactive Solution Tabs} to close any existing tabs from previous solutions.

The following steps, summarized in Figure 28 explain how to unroll the loop.

5. In the Directive tab, select loop \texttt{Shift\_Accum\_Loop}. (Reminder: the source code must be open in the Information pane to see any code objects in the Directive tab).

6. Right-click and select \textbf{Insert Directives}.


When optimizing a design, you must often perform multiple iterations of optimizations to determine what the final optimization should be. By adding the optimizations to the directive file, you can ensure they are \textit{not} automatically carried forward to the next solution.
Storing the optimizations in the solution directive file allows different solutions to have different optimizations. Had you added the optimizations as pragmas in the code, they would be automatically carried forward to new solutions, and you would have to modify the code to go back and re-run a previous solution.

Leave the other options in the Directives window unchecked and blank to ensure that the loop is fully unrolled.

8. Click OK to apply the directive.

9. Apply the directive to partition the array into individual elements.
   a) In the Directive tab, select array shift_reg.
   b) Right-click and select Insert Directives.
   c) Select Array_Partition from the Directive drop-down menu.
   d) Specify the type as complete.
   e) Select OK to apply the directive.

With the directives embedded in the code from solution2 and the two new directives just added, the directive pane for solution4 appears as shown in Figure 29.

![Figure 29: Solution4 Directives](image)

In Figure 29, notice the directives applied in solution2 as pragmas have a different annotation (#HLS) than those just applied and saved to the directive file (%HLS). You can view the newly added directives in the Tcl file.

10. In the Explorer pane, expand the Constraint folder in Solution3 as shown in Figure 30.

11. Double-click the solution4 directives.tcl file to open it in the Information pane.
12. Click the **Synthesis** toolbar button to synthesize the design. When synthesis completes, the synthesis report automatically opens.

13. Compare the results of the different solutions.

14. Click the **Compare Reports** toolbar button.

   Alternatively, use **Project > Compare Reports**.

15. Add **solution1**, **solution2**, and **solution3** to the comparison.

16. Click **OK**.

**Figure 31** shows the comparison of the reports. **solution3** has the smallest initiation interval and can process data much faster. As the interval is only 16, it starts to process a new set of inputs every 16 clock cycles.
It is possible to perform additional optimizations on this design. For example, you could use Pipelining to further improve the throughput and lower the interval. The tutorial Design Optimization provides details on using pipelining to improve the interval.

As mentioned earlier, you could modify the code itself to use arbitrary precision types. For example, if the data types are not required to be 32-bit int types, you could use bit-accurate types (for example, 6-bit, 14-bit or 22-bit types), provided that they satisfy the required accuracy. For more details on using arbitrary precision type see the tutorial Arbitrary Precision Types.

**Conclusion**

In this tutorial, you learned how to:

- Create a Vivado High-Level Synthesis project in the GUI and Tcl environments.
- Execute the major steps in the HLS design flow.
- Create and use a Tcl file to run Vivado HLS.
- Create new solutions, add optimization directives, and compare the results of different solutions.
Chapter 3  C Validation

Overview

Validation of the C algorithm is an important part of the High-Level Synthesis (HLS) process. The time spent ensuring the C algorithm is performing the correct operation and creating a C test bench, which confirms the results are correct, reduces the time spent analyzing designs which are incorrect “by design” and ensures the RTL verification can be performed automatically.

This tutorial consists of three lab exercises.

- Lab1: Review the aspects of a good C test bench, the basic operations for C validation and the C debugger.
- Lab2: Validate and debug a C design using arbitrary precision C types.
- Lab3: Validate and debug a design using arbitrary precision C++ types.

Tutorial Design Description

You can download the tutorial design file from the Xilinx website. See the information in Obtaining the Tutorial Designs.

This tutorial uses the design files in the tutorial directory Vivado_HLS_Tutorial\C_Validation.

The sample design used in this tutorial is a Hamming Window FIR. There are three versions of this design:

- Using native C data types.
- Using ANSI C arbitrary precision data types.
- Using C++ arbitrary precision data types.

This tutorial explains the operation and methodology for C validation using High-Level Synthesis. There are no design goals for this tutorial.
Lab 1: C Validation and Debug

Overview

This exercise reviews the aspects of a good C test bench and explains the basic operations of the High-Level Synthesis C debug environment.

IMPORTANT: The figures and commands in this tutorial assume the tutorial data directory \texttt{Vivado\_HLS\_Tutorial} is unzipped and placed in the location \texttt{C:\Vivado\_HLS\_Tutorial}

If the tutorial data directory is unzipped to a different location, or on Linux systems, adjust the few pathnames referenced, to the location you have chosen to place the \texttt{Vivado\_HLS\_Tutorial} directory.

Step 1: Create and Open the Project

1. Open the Vivado HLS Command Prompt.
   a. On Windows use Start > All Programs > Xilinx Design Tools > Vivado 2014.1 > Vivado HLS > Vivado HLS 2014.1 Command Prompt (Figure 32).
   b. On Linux, open a new shell.

   ![Figure 32: Vivado HLS Command Prompt](image)

2. Using the command prompt window (Figure 33), change the directory to the C Validation tutorial, lab1.

3. Execute the Tcl script to setup the Vivado HLS project, using the command \texttt{vivado\_hls –f run\_hls.tcl} as shown in Figure 33.

![Figure 33: Setup the Tutorial Project](image)
4. When Vivado HLS completes, open the project in the Vivado HLS GUI using the command `vivado_hls -p hamming_window_prj` as shown in Figure 34.

![Figure 34: Initial Project for C Validation Lab 1]

```bash
C:\Viado_HLS_Tutorial\C_Validation\lab1>vivado_hls -p hamming_window_prj

Test Passed
```

Figure 34: Initial Project for C Validation Lab 1
Step 2: Review Test Bench and Run C Simulation

1. Open the C test bench for review by double-clicking `hamming_window.c` in the Test Bench folder (Figure 35).

A review of the test bench source code shows the following good practices:

- The test bench:
  - Creates a set of expected results that confirm the function is correct.
  - Stores the results in array `sw_result`.

- The Design Under Test (DUT) is called to generate results, which are stored in array `hw_result`. Because the synthesized functions use the `hw_result` array, it is this array that holds the RTL-generated results later in the design flow.

- The actual and expected results are compared. If the comparison fails, the value of variable `err_cnt` is set to a non-zero value.

- The test bench issues a message to the console if the comparison failed, but more importantly returns the results of the comparison. If the return value is zero the test bench validates the results are good.
This process of checking the results and returning a value of zero if they are correct automates RTL verification.

You can execute the C code and test bench to confirm that the code is working as expected.

2. Click the **Run C Simulation** toolbar button to open the C Simulation Dialog box, shown in **Figure 36**.

![Figure 36: Run C Simulation Dialog box](image)

3. Select **OK** to run the C simulation.

As shown in **Figure 37**, the following actions occur when C simulation executes:

- The simulation output is shown in the Console window.
- Any print statements in the C code are echoed in the Console window. This example shows the simulation passed correctly.
The C simulation executes in the solution sub-directory `csim`. You can find any output from the C simulation in the build folder, which is the location at which you can see the output file `result.dat` written by the `fprintf` command highlighted in Figure 37.

Because the C simulation is not executed in the project directory, you must add any data files to the project as C test bench files (so they can be copied to the `csim/build` directory when the simulation runs). Such files would include, for example, input data read by the test bench.

![Figure 37: C Simulation Results](image)

**Step 3: Run the C Debugger**

A C debugger is included as part of High-Level Synthesis.

1. Click the **Run C Simulation** toolbar button to open the C Simulation Dialog box.
2. Select the **Debug** option as shown in Figure 38.
3. Click **OK** to run the simulation.
Figure 38: C Simulation Dialog Box
The Debug option compiles the C code and then opens the Debug environment, as shown in Figure 39. Before proceeding, note the following:

- Highlighted at the top-left in Figure 39, you can see that the perspective has changed from Synthesis to Debug. Click the perspective buttons to return to the synthesis environment at any time.

- By default, the code compiles in debug mode. The Debug option automatically opens the debug perspective at time 0, ready for debug to begin. To compile the code without debug information, select the Optimizing Compile option in the C Simulation dialog box.

You can use the Step Into button (Figure 40) to step through the code line-by-line.

4. Expand the Variables window to see the `sw_results` array.
5. Expand the sw_results array to the view shown in Figure 41.

6. Click the Step Into button (or key F5) repeatedly until you see the values being updated in the Variables window.

![Figure 41: Analysis of C Variables](image)

In this manner, you can analyze the C code and debug it if the behavior is incorrect.

For more detailed analysis, to the right of the Step Into button are the Step Over (F6), Step Return (F7) and the Resume (F8) buttons.

7. Scroll to line 69 in the source code window.

8. Double-click in the left margin to create a breakpoint (blue dot), as shown in Figure 42.

9. Activate the Breakpoints tab, also shown in Figure 42, to confirm there is a breakpoint set at line 69.

10. Click the Resume button (highlighted in Figure 42) or the F8 key to execute up to the breakpoint.
11. Click the **Step Into** button (or key F5) multiple times to step into the `hamming_window` function.

12. Click the **Step Return** button (or key F7) to return to the main function.

13. Click the red **Terminate** button to end the debug session.

   The Terminate button becomes the Run C Simulation button. You can restart the debug session from within the Debug perspective.

14. Exit the Vivado HLS GUI and return to the command prompt.

---

**Lab 2: C Validation with ANSI C Arbitrary Precision Types**

**Introduction**

This exercise uses a design with arbitrary precision C types. You will review and debug the design in the GUI.

**Step 1: Create and Open the Project**

1. From the Vivado HLS command prompt used in Lab 1, change to the `lab2` directory, as shown in Figure 43.

2. To create a new Vivado HLS project, type `vivado_hls -f run_hls.tcl`. 

---

**Figure 42: Using Breakpoints**

![Image](https://example.com/image.png)
3. To open the Vivado HLS GUI project, type `vivado_hls -p hamming_window_prj`.

4. Open the Source folder in the explorer pane and double-click `hamming_window.c` to open the code, as shown in Figure 44.

5. Hold down the `Ctrl` key and click `hamming_window.h` on line 45 to open this header file.

6. Scroll down to view the type definitions (Figure 45).
In this lab, the design is the same as Lab 1, however, the types have been updated from the standard C data types (int16_t and int32_t) to the arbitrary precision types provided by Vivado High-Level Synthesis and defined in header file ap_cint.h.

More details for using arbitrary precision types are discussed in the tutorial Arbitrary Precision Types. An example of using arbitrary precision types would be to change this file to use 12-bit input data types: standard C types only support data widths on 8-bit boundaries.

This exercise demonstrates how such types can be debugged.

Step 2: Run the C Debugger

1. Click the Run C Simulation toolbar button to open the C Simulation Dialog box.
2. Select the Debug option.
3. Click OK to run the simulation.

The warning and error message shown in Figure 46 appears.

You cannot debug the arbitrary precision types used for ANSI C designs in the debug environment.

---

**IMPORTANT!** When working with arbitrary precision types you can use the Vivado HLS debug environment only with C++ or SystemC. When using arbitrary precision types with ANSI C, the debug environment cannot be used. With ANSI C, you must instead use printf or fprintf statements for debugging.
4. Expand the Test Bench folder in the Explorer pane.
5. Double-click the file `hamming_window_test.c`.
6. Scroll to line 78 and remove the comments in front of the `printf` statement (as shown in Figure 47).

![Figure 46: C Simulation Dialog Box](image)

```
Vivado HLS C Simulation could not complete...
Please check the error and warning messages:
- There are 2 errors
```

![Figure 47: Enable Printing of the Results](image)

```
Compiling C:\Vivado_HLS_Tutorial\C\Validation\lab2\hamming_window_test.c in debug mode ...
@E [SIM-34] 'apcc' is required to include the header file. Do not select 'Debug' in GUI a
@E [SIM-1] CSim file generation failed: compilation error(s).
@I [LIC-101] Checked in feature [VIVADO_HLS]
```

7. Save the file.
8. Click the **Run C Simulation** toolbar button or the menu **Project > Run C simulation** to open the C Simulation Dialog box.
9. Ensure the **Debug** option is **not selected**.
10. Click **OK** to run the simulation.
    
    The results appear in the console window (**Figure 48**).
11. Exit the Vivado HLS GUI and return to the command prompt.
Lab 3: C Validation with C++ Arbitrary Precision Types

Overview

This exercise uses a design with arbitrary precision C++ types. You will review and debug the design in the GUI.

Step 1: Create and Open the Project

1. From the Vivado HLS command prompt used in Lab 2, change to the lab3 directory.
2. Create a new Vivado HLS project by typing `vivado_hls -f run_hls.tcl`.
3. Open the Vivado HLS GUI project by typing `vivado_hls -p hamming_window_prj`.
4. Open the Source folder in the explorer pane and double-click `hamming_window.cpp` to open the code, as shown in Figure 49.

![Figure 49: C++ Code for C Validation Lab 3](image)

5. Hold down the Ctrl key down and click `hamming_window.h` on line 45 to open this header file.

Figure 49: C++ Code for C Validation Lab 3
6. Scroll down to view the type definitions (Figure 50).

![Image of code snippet]

**Figure 50: Type Definitions for C Validation Lab 3**

**Note:** In this lab, the design is the same as in Lab 1 and Lab 2, with one exception. The design is now C++ and the types have been updated to use the C++ arbitrary precision types, `ap_int<N>`, provided by Vivado HLS and defined in header file `ap_int.h`. 
Step 2: Run the C Debugger

1. Click the Run C Simulation toolbar button to open the C Simulation Dialog box.
2. Select the Debug option.
3. Click OK.
   The debug environment opens.
4. Select the hamming_window.cpp code tab.
5. Set a breakpoint at line 61 as shown in Figure 51.
6. Click the Resume button (or key F8) to execute the code up to the breakpoint.

7. Click the Step Into button (or the F5 key) twice to see the view in Figure 52.
   The variables in the design are now C++ arbitrary precision types. These types are defined in header file ap_int.h. When the debugger encounters these types, it follows the definition into the header file.
   As you continue stepping through the code, you have the opportunity to observe in greater detail how the results for arbitrary precision types are calculated.
A more productive methodology is to exit the `ap_int.h` header file and return to view the results.

8. Click the **Step Return** button (or the **F7** key) to return to the calling function.

9. Select the **Variables** tab.

10. Expand the `outdata` variable, as shown in Figure 53 to see the value of the variable shown in the VAL parameter.

Arbitrary precision types are a powerful means to create high-performance, bit-accurate hardware designs. However, in a debug environment, your productivity can be reduced by
stepping through the header file definitions. Use breakpoints and the step return feature to skip over the low-level calculations and view the value of variables in the Variables tab.

**Conclusion**

In this tutorial, you learned:

- The importance of the C test bench in the simulation process.
- How to use the C debug environment, set breakpoints and step through the code.
- How to debug C and C++ arbitrary precision types.
Chapter 4 Interface Synthesis

Overview

Interface synthesis is the process of adding RTL ports to the C design. In addition to adding the physical ports to the RTL design, interface synthesis includes an associated I/O protocol, allowing the data transfer through the port to be synchronized automatically and optimally with the internal logic.

This tutorial consists of four lab exercises that cover the primary features and capabilities of interface synthesis.

- Lab 1: Review the function return and block-level protocols.
- Lab 2: Understand the default I/O protocol for ports and learn how to select an I/O protocol.
- Lab 3: Review how array ports are implemented and can be partitioned.
- Lab 4: Create an optimized implementation of the design and add AXI4 interfaces.

Tutorial Design Description

Download tutorial design file from the Xilinx website. Refer to the information in Obtaining the Tutorial Designs.

This tutorial uses the design files in the tutorial directory Vivado_HLS_Tutorial\Interface_Synthesis.

About the Labs

- The sample design used in the first two labs in this tutorial is a simple one, which helps the focus to remain on the interfaces.
- The final two lab exercises use a multi-channel accumulator.
- This tutorial explains how to implement I/O ports and protocols using High-Level Synthesis.
- In Lab 4, you create an optimal implementation of the design used in Lab3.
Interface Synthesis Lab 1: Block-Level I/O protocols

Overview
This lab explains what block-level I/O protocols are and to control them.

**IMPORTANT:** The figures and commands in this tutorial assume the tutorial data directory Vivado_HLS_Tutorial is unzipped and placed in the location C:\Vivado_HLS_Tutorial. If the tutorial data directory is unzipped to a different location, or on Linux systems, adjust the few pathnames referenced, to the location you have chosen to place the Vivado_HLS_Tutorial directory.

Step 1: Create and Open the Project
1. Open the Vivado HLS Command Prompt.
   a. On Windows use Start > All Programs > Xilinx Design Tools > Vivado 2014.1 > Vivado HLS > Vivado HLS 2014.1 Command Prompt (Figure 54).
   b. In Linux, open a new shell.

   ![Figure 54: Vivado HLS Command Prompt](Image)
2. Using the command prompt window (Figure 55), change directory to the Interface Synthesis tutorial, lab1.

3. Execute the Tcl script to setup the Vivado HLS project, using the command `vivado_hls -f run_hls.tcl`, as shown in **Figure 55**.

![Figure 55: Setup the Tutorial Project](image)

4. When Vivado HLS completes, open the project in the Vivado HLS GUI using the command `vivado_hls -p adders_prj`, as shown in **Figure 56**.

![Figure 56: Initial Project for Interface Synthesis Lab 1](image)

**Step 2: Create and Review the Default Block-Level I/O Protocol**

1. Double-click `adders.c` in the Source folder to pen the source code for review (**Figure 57**).

   This example uses a simple design to focus on the I/O implementation (and not the logic in the design). The important points to take from this code are:
   
   - Directives in the form of pragmas have been added to the source code to prevent any I/O protocol being synthesized for any of the data ports (inA, inB and inC). I/O port protocols are reviewed in the next lab exercise.
   
   - This function returns a value and this is the only output from the function. As seen in later exercises, not all functions return a value. The port created for the function return is discussed in this lab exercise.
2. Execute the **Run C Synthesis** command using the dedicated toolbar button or the **Solution** menu.

   When synthesis completes, the synthesis report opens automatically.

3. To review the RTL interfaces scroll to the Interface summary at the end of the synthesis report.

   The Interface summary and Outline tab are shown in **Figure 58**.

   ![Figure 58: Interface Summary](image)

There are three types of ports to review:

- The design takes more than one clock cycle to complete, so a clock and reset have been added to the design: ap_clk and ap_rst. Both are single-bit inputs.
- A block-level I/O protocol has been added to control the RTL design: ports ap_start, ap_done, ap_idle and ap_ready. These ports will be discussed shortly.
- The design has four data ports.
  - Input ports In1, In2, and In3 are 32-bit inputs and have the I/O protocol ap_none (as specified by the directives in Figure 58).
  - The design also has a 32-bit output port for the function return, ap_return.

The block-level I/O protocol allows the RTL design to be controlled by via additional ports independently of the data I/O ports. This I/O protocol is associated with the function itself, not with any of the data ports. The default block-level I/O protocol is called ap_ctrl_hs. Figure 58 shows this protocol is associated with the function return value (this is true even if the function has no return value specified in the code).

Table 1 summarizes the behavior of the signals for block-level I/O protocol ap_ctrl_hs.

**Note:** The explanation here uses the term “transaction”. In the context of high-level synthesis, a transaction is equivalent to one execution of the C function (or the equivalent operation in the synthesized RTL design).

<table>
<thead>
<tr>
<th>Exercise</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>ap_start</td>
<td>This signal controls the block execution and must be asserted to logic 1 for the design to begin operation. It should be held at logic 1 until the associated output handshake ap_ready is asserted. When ap_ready goes high, the decision can be made on whether to keep ap_start asserted and perform another transaction or set ap_start to logic 0 and allow the design to halt at the end of the current transaction. If ap_start is asserted low before ap_ready is high, the design might not have read all input ports and might stall operation on the next input read.</td>
</tr>
<tr>
<td>ap_ready</td>
<td>This output signal indicates when the design is ready for new inputs. The ap_ready signal is set to logic 1 when the design is ready to accept new inputs, indicating that all input reads for this transaction have been completed. If the design has no pipelined operations, new reads are not performed until the next transaction starts. This signal is used to make a decision on when to apply new values to the inputs ports and whether to start a new transaction should using the ap_start input signal. If the ap_start signal is not asserted high, this signal goes low when the design completes all operations in the current transaction.</td>
</tr>
<tr>
<td>ap_done</td>
<td>This signal indicates when the design has completed all operations in the current transaction. A logic 1 on this output indicates the design has completed all operations in this</td>
</tr>
</tbody>
</table>
## Exercise Description

transaction. Because this is the end of the transaction, a logic 1 on this signal also indicates the data on the ap_return port is valid.

Not all functions have a function return argument and hence not all RTL designs have an ap_return port.

<table>
<thead>
<tr>
<th>Signal</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>ap_idle</strong></td>
<td>This signal indicates if the design is operating or idle (no operation). The idle state is indicated by logic 1 on this output port. This signal is asserted low once the design starts operating. This signal is asserted high when the design completes operation and no further operations are performed.</td>
</tr>
</tbody>
</table>

### Table 1: Block Level I/O protocol ap_ctrl_hs

You can observe the behavior of these signals by viewing the trace file produced by RTL cosimulation. This is discussed in the tutorial RTL Verification, but Figure 59 shows the waveforms for the current synthesis results.

### Figure 59: RTL Waveforms for Block Protocol Signals

The waveforms in Figure 56 show the behavior of the block-level I/O signals.

- The design does not start operation until `ap_start` is set to logic 1.
• The design indicates it is no longer idle by setting port ap_idle low.

• Five transactions are shown. The first three input values (10, 20 and 30) are applied to input ports In1, In2 and In3 respectively.

• Output signal ap_ready goes high to indicate the design is ready for new inputs on the next clock.

• Output signal ap_done indicates when the design is finished and that the value on output port ap_return is valid (the first output value, 60, is the sum of all three inputs).

• Because ap_start is held high, the next transaction starts on the next clock cycle.

Note: In RTL Cosimulation, all design and port input control signals are always enabled. For example, in Figure 59 signal ap_start is always high.

In the 2nd transaction, notice on port ap_return, the first output has the value 70. The result on this port is not valid until the ap_done signal is asserted high.

Step 3: Modify the Block-Level I/O protocol

The default block-level I/O protocol is the ap_ctrl_hs protocol (the Control Handshake protocol). In this step, you create a new solution and modify this protocol.

1. Select New Solution from the toolbar or Project menu to create a new solution.

2. Leave all settings in the new solution dialog box at their default setting and click Finish.

3. Select the C source code tab in the Information pane (or re-open the C source code if it was closed).

4. Activate the Directives tab and select the top-level function, as shown in Figure 60.

Figure 60: Top-level Function Selected
Because the block-level I/O protocols are associated with the function, you must specify them by selecting the top-level function.

5. In the Directives tab, mouse over the top-level function adders, right-click, and select Insert Directives.

The Directives Editor dialog box opens.

**Figure 61** shows this dialog box with the drop-down menu for the interface mode activated.

![Vivado HLS Directive Editor](image)

**Figure 61: Directive Dialog box for ap_ctrl_none**

The drop-down menu shows there are three options for the block-level interface protocol:

- **ap_ctrl_none**: No block-level I/O control protocol.
- **ap_ctrl_hs**: The block-level I/O control handshake protocol we have reviewed.
- **ap_ctrl_chain**: The block-level I/O protocol for control chaining. This I/O protocol is primarily used for chaining pipelined blocks together.
- **s_axilite**: May be applied in addition to **ap_ctrl_hs** or **ap_ctrl_chain** to implement the block-level I/O protocol as an AXI Slave Lite interface in place of separate discrete IO ports.
The block-level IO protocol `ap_ctrl_chain` is not covered in this tutorial. This protocol is similar to `ap_ctrl_hs` protocol but with an additional input signal, `ap_continue`, which must be high when `ap_done` is asserted for the next transaction to proceed. This allows downstream blocks to apply back-pressure on the system and halt further processing when they are unable to continue accepting new data.

6. In the **Destination** section of the **Directives Editor** dialog box, select **Source File**.

   By default, directives are placed in the `directives.tcl` file. In this example, the directive is placed in the source file with the the existing I/O directives.

7. From the drop-down menu, select **ap_ctrl_none**.

8. Click **OK**.

The source file now has a new directive, highlighted in both the source code and directives tab in **Figure 62**.

The new directive shows the associated function argument/port called `return`. All interface directives are attached to a function argument. For block-level I/O protocols, the return argument is used to specify the block-level interface. This is true even if the function has no return argument in the source code.

9. Click the **Run C Synthesis** toolbar button or use the menu **Solution > Run C Synthesis** to synthesize the design.

   Adding the directive to the source file modified the source file. **Figure 62** shows the source file name as `*adders.c`. The asterisk indicates that the file is modified but not saved.

10. Click **Yes** to accept the changes to the source file.

   When the report opens, the Interface summary appears, as shown in **Figure 63**.
When the interface protocol `ap_ctrl_none` is used, no block-level I/O protocols are added to the design. The only ports are those for the clock, reset and the data ports.

Note that without the `ap_done` signal, the consumer block that accepts data from the `ap_return` port now has no indication when the data is valid.

In addition, the RTL cosimulation feature requires a block-level I/O protocol to sequence the test bench and RTL design for cosimulation automatically. Any attempt to use RTL cosimulation results in the following error message and RTL cosimulation with halt:

```
@E [SIM-345] Cosim only supports the following 'ap_ctrl_none' designs: (1) combinational designs; (2) pipelined design with task interval of 1; (3) designs with array streaming or hls_stream ports.
@E [SIM-4] *** C/RTL co-simulation finished: FAIL ***
```

Exit the Vivado HLS GUI and return to the command prompt.

**Interface Synthesis Lab 2: Port I/O protocols**

**Overview**

This exercise explains how to specify port I/O protocols.

**Step 1: Create and Open the Project**

1. From the Vivado HLS command prompt used in Lab 1, change to the `lab2` directory as shown in Figure 64.

2. Type `vivado_hls -f run_hls.tcl` to create a new Vivado HLS project.
3. Type `vivado_hls -p adders_io_prj` to open the Vivado HLS GUI project.

4. Open the source code as shown in **Figure 65**.

![Figure 65: C Code for Interface Synthesis Lab 2](image)

The source code for this exercise is similar to the simple code used in Lab 1. For similar reasons, it helps focus on the interface behavior and not the core logic.

This time, the code does not have a function return, but instead passes the output of the function through the pointer argument `*in_out1`. This also provides the opportunity to explore the interface options for bi-directional (input and output) ports.

The types of I/O protocol that you can add to C function arguments by interface synthesis depends on the argument type. These options are fully described in the Vivado High-Level Synthesis User Guide (UG902).

The pointer argument in this example is both an input and output to the function. In the RTL design, this argument is implemented as separate input and output ports.
For the code shown in Figure 65, the possible options for each function argument are described in Table 2.

<table>
<thead>
<tr>
<th>Function Argument</th>
<th>I/O protocol Options</th>
</tr>
</thead>
</table>
| In1 and In2        | These are pass-by-value arguments that can be implemented with the following I/O Protocols:  
|                   | • ap_none: No I/O protocol. This is the default for inputs.  
|                   | • ap_stable: No I/O protocol.  
|                   | • ap_ack: Implemented with an associated output acknowledge port.  
|                   | • ap_vld: Implemented with an associated input valid port.  
|                   | • ap_hs: Implemented with both input valid and output acknowledge ports.  
| in_out1            | This is a pass-by-reference output that can be implemented with the following I/O protocols:  
|                   | • ap_none: No I/O protocol. This is the default for inputs.  
|                   | • ap_stable: No I/O protocol.  
|                   | • ap_ack: Implemented with an associated input acknowledge port.  
|                   | • ap_vld: Implemented with an associated output valid port. This is the default for outputs.  
|                   | • ap_ovld: Implemented with an associated output valid port (no valid port for the input part of any inout ports).  
|                   | • ap_hs: Implemented with both input valid port and output acknowledge ports.  
|                   | • ap_fifo: A FIFO interface with associated output write and input FIFO full ports.  
|                   | • ap_bus: A Vivado HLS bus interface protocol.  

Table 2: Port Level I/O Protocol Options for Lab 2

Note: The port directives applied in Lab 1 were not actually necessary because ap_none is the default I/O protocol for these C arguments. The directives were provided to avoid addressing any I/O port protocol behavior in that exercise, default behavior or not.

In this exercise, you implement a selection of I/O protocols.
Step 2: Specify the I/O Protocol for Ports

1. Ensure that you can see the C source code in the Information pane.
2. Activate the Directives tab and select input argument in1, as shown in Figure 66.

3. Right-click and select Insert Directives.
4. When the Directives Editor opens leave the directives drop-down menu as INTERFACE.
   a. Leave the destination at the default value. This time, the directives are stored in the directives.tcl file.
   b. Select ap_vld from the mode drop-down menu
   c. Click OK.
5. Select argument in2 and add an interface directive to specify the I/O protocol ap_ack.
6. Select argument in_out1 and add an interface directive to specify the I/O protocol ap_hs.
7. In the Explorer pane, expand the Constraints folder and double-click the `directives.tcl` file to open it, as shown in Figure 67.

![Figure 67: Directives for Lab 2](image)

8. Synthesize the design.

9. Review the Interface summary when the report file opens (Figure 68).

![Figure 68: Interface summary for Lab 2](image)

- The design has a clock and reset.
- The default block-level I/O protocol signals are present.
- Port in1 is implemented with a data port and an associated input valid signal.
• The data on port in1 is only read when port in1_ap_vld is active high.
• Port in2 is implemented with a data port and an associated output acknowledge signal.
• Port in2_ap_ack will be active high when data port in2 is read.
• The `inout_i` identifies the input part of argument `inout1`. This has associated input valid port `inout1_i_ap_vld` and output acknowledge port `inout1_i_ap_ack`.
• The output part of argument `inout1` is identified as `inout_o`. This has associated output valid port `inout1_o_ap_vld` and input acknowledge port `inout1_o_ap_ack`.

10. **Exit** the Vivado HLS GUI and return to the command prompt.

---

**Interface Synthesis Lab 3: Implementing Arrays as RTL Interfaces**

**Introduction**

This exercise shows how array arguments on functions you can implement as a number of different types of RTL port.

**Step 1: Create and Open the Project**

1. From the Vivado HLS command prompt window used in the previous lab, change to the `lab3` directory.
2. Create a new Vivado HLS project by typing `vivado_hls –f run_hls.tcl`
3. Open the Vivado HLS GUI project by typing `vivado_hls –p array_io_prj`
4. Open the source code as shown in Figure 69.

This design has an input array and an output array. The comments in the C source explain how the data in the input array is ordered as channels and how the channels are accumulated. To understand the design, you can also review the test bench and the input and output data in file result.golden.dat.

![Figure 69: C Code for Interface Synthesis Lab 3](image)

**Step 2: Synthesize Array Function Arguments to RAM ports**

In this step, you review how array ports are synthesized to RAM ports.

1. Synthesize the design and review the Interface summary when the report opens (Figure 70).

   The interface summary shows how array arguments in the C source are by default synthesized into RTL RAM ports.

   - The design has a clock, reset and the default block-level I/O protocol `ap_ctrl_hs` (noted on the clock in the report).
   - The `d_o` argument has been synthesized to a RAM port (I/O protocol `ap_memory`).
   - A data port (`d_o_d0`).
   - An address port (`d_o_address0`).
   - Control ports for chip-enable (`d_o_ce0`) and a write-enable port (`d_o_we0`).
   - The `d_i` argument has been synthesized to a similar RAM interface, but has an input data port (`d_i_q0`) and no write-enable port because this interface only reads data.
In both cases, the data port is the width of the data values in the C source (16-bit integers in this case) and the width of the address port has been automatically sized match to the number of addresses that must be accessed (5-bit for 32 addresses).

Figure 70: Interface Summary for Initial Lab 3 design

Synthesizing array arguments to RAM ports is the default. You can control how these ports are implemented using a number of other options. The remaining steps in Lab 3 demonstrate these options:

- Using a single-port or dual-port RAM interface.
- Using FIFO interfaces.
- Partitioning into discrete port.
Step 3: Using Dual-port RAM and FIFO interfaces

High-Level Synthesis lets you specify a RAM interface a single-port or dual-port. If you do not make such a selection, Vivado HLS automatically analyzes the design and selects the number of ports to maximize the data rate.

Step 2 used a single-port RAM interface because the for-loop in the source code (Figure 69) is by default left rolled: each iteration of the loop is executed in turn:

• Read the input port.
• Read the accumulated result from the internal RAM.
• Sum the accumulated and new data and write into the internal RAM.
• Write the result to the output port.
• Repeat for the next iteration of the loop.

This ensures only a single input read and output write is ever required. Even if multiple input and outputs are made available, the internal logic cannot take advantage of any additional ports.

**Note:** If you specify a dual-port RAM and Vivado HLS can determine only a single port is required, it uses a single-port and over-ride the dual-port specification.

In this design, if you want to implement an array argument using multiple RTL ports, the first thing you must do is unroll the for-loop and allow all internal operations to happen in parallel, otherwise there is no benefit in multiple ports: the rolled for-loop ensure only one data sample can be read (or written) at a time.

1. Select **New Solution** from the toolbar or Project menu to create a new solution.
2. Accept the defaults, and click **Finish**.
3. Ensure the C source code is visible in the Information pane.
4. In the Directives tab, select the for-loop, For_Loop, and right-click to open the **Directives Editor** dialog box.
   a. In the Directives Editor dialog box activate the Directives drop-down menu at the top and select **UNROLL**.
b. With the Directives Editor as shown in Figure 71, click **OK**.

![Vivado HLS Directive Editor](image)

*Figure 71: Directives Editor to Unroll For Loop*
Next, specify a dual-port RAM for input reads. The Resource directive indicates the type of RAM connected to an interface.

5. In the Directives tab, select port d_i and right-click to open the Directives Editor dialog box.
   a. In the Directives Editor activate the Directives drop-down menu at the top and select RESOURCE.
   b. Click the core options box and select RAM_2P_BRAM.
   c. Verify that the settings in the Directives Editor dialog box are as shown in Figure 72 and click OK.

![Figure 72: Directives Editor for Specifying a Dual-port RAM](image-url)
Implement the output port using a FIFO interface.

6. In the Directives tab, select port \texttt{d\_o} and right-click to open the \textbf{Directives Editor} dialog box.
   a. In the Directives Editor, leave the directive as \textbf{Interface}.
   b. From the Mode drop-down menu, select \texttt{ap\_fifo}.
   c. Click \textbf{OK}.

   The \textbf{Directive} tab shows the directives now applied to the design (\textbf{Figure 73}).

   \begin{figure}[h]
   \centering
   \includegraphics[width=\textwidth]{figure73.png}
   \caption{Directives Summary for Lab 2 Solution2}
   \end{figure}

7. Synthesize the design.
When the report opens in the Information pane, the Interface summary is as shown in Figure 74.

- The design has the standard clock, reset and block-level I/O ports.
- Array argument d_o has been implemented as a FIFO interface with a 16-bit data port (d_o_din) and associated output write (d_o_write) and input FIFO full (d_o_full_n) ports.
- Argument d_i has been implemented as a dual-port RAM interface.

![Figure 74: Directives Editor Specifying Block RAM Interface](image)

By using a dual-port RAM interface, this design can accept input data at twice the rate of the previous design. However, by using a single-port FIFO interface on the output the output data rate is the same as before.
Step 4: Partitioned RAM and FIFO Array interfaces

In this step, you learn how to partition an array interface into any arbitrary number of ports.

1. Select **New Solution** from the toolbar or the Project menu and create a new solution.
2. Accept the defaults, and click **Finish**. This includes copying existing directives from solution2.
3. Ensure the C source code is visible in the Information pane.
4. In the directives tab, select d_o and right-click to open the **Directives Editor** dialog box.
   a. In the Directives Editor dialog box activate the Directives drop-down menu at the top and select ARRAY_PARTITION.
   b. Activate the type drop-down menu and select block to partition the array into blocks.
   c. In the Factor dialog box, enter the value 4.
   d. With the Directives Editor as shown in Figure 75, click **OK**.

![Directives Editor for Partitioning Array d_o](image)

Now, partition the input array into two blocks **(not four)**.
5. In the Directives tab, select d_i and repeat the previous step, but this time partition the port with a factor of 2.

The directives tab shows the directives now applied to the design (Figure 76).

![Figure 76: Directives Summary for Lab 2 Solution3](image)

6. Synthesize the design.

When the report opens in the Information pane, the Interface summary is as shown in Figure 77.

- The design has the standard clock, reset and block-level I/O ports.
- Array argument d_o has been implemented as a four separate FIFO interfaces.
- Argument d_i has been implemented as a two separate RAM interfaces, each of which uses a dual-port interface. (If you see 4 separate RAM interfaces, confirm a partition factor for d_i is 2 and not 4).
If input port d_i was partitioned into four, only a single-port RAM interface would be required for each port. Because the output port can only output four values at once, there would be no benefit in reading 8 inputs at once.

The final step in this tutorial on arrays is to partition the arrays completely.
Step 5: Fully Partitioned Array interfaces

This step shows you how to partition an array interface into individual ports.

1. Select **New Solution** from the toolbar and create a new solution.
2. Click **Finish** and accept the defaults. This includes copying existing directives from solution.
3. Ensure the C source code is visible in the Information pane.
4. In the Directive tab, select the existing partition directive for d_o as shown in Figure 78.
5. Right-click and select **Modify Directive**.

![Figure 78: Modifying the Directive for d_o](image)

6. In the Directives Editor dialog box:
   a. Activate the **Type** drop-down menu and modify the partitioning style to **Complete**.
   b. In the Factor dialog box, you can remove the value 4 or leave it as-is. The factor is ignored for this type of partitioning.
c. With the Directives Editor as shown in Figure 79, click OK.

![Vivado HLS Directive Editor](image)

**Figure 79: Directives Editor for Partitioning Array d_o**

7. In the Directives tab, select d_i and repeat the previous step to completely partition the d_i array.

Optionally, you can delete the directive on d_i specifying the resource. If the array is partitioned into individual elements, the Resource directive, which specifies a RAM resource, is ignored.
The Directives tab shows the directives now applied to the design (Figure 80).

![Directive Summary](image.png)

**Figure 80: Directives Summary for Lab 2 Solution3**

8. Synthesize the design.

9. When the report opens in the Information pane, review the interface summary. Note the following:

   - The design has the standard clock, reset and block-level I/O ports.
   - Array argument d_o has been implemented as a 32 separate FIFO interfaces.
   - Argument d_i has been implemented as a 32 separate scalar port. Because the default interface for input scalars in no I/O protocol, they have the I/O protocol ap_none.

   Although this tutorial has focused exclusively on the I/O interfaces, at this point it is worth examining the differences in performance across all four solutions.

10. Select Compare Reports from the toolbar or the Project menu to open a comparison of the solutions.
11. In the Solution Selection dialog box, add each of the four solutions to the Selected Solutions pane (Figure 81).

12. Click OK.

When the solutions comparison report opens (Figure 82), it shows that solution 4, using a unique port for each array element, is much faster than the previous solutions. The internal logic can access the data as soon as it is required. (There is no performance bottleneck due to port accesses.)
Scroll further down the comparison report (Figure 83) and note that solutions with more I/O ports (solutions 2, 3 and 4), allowing more parallel processing, also use considerably more resources.

![Figure 83: Resource Comparisons for All Lab 3 Solutions](image)

In the next exercise, you implement this same design with an optimum balance between the ports and resources. In addition to this more optimal implementation, the next exercise shows how to add AXI4 interfaces to the design.

13. Exit the Vivado HLS GUI and return to the command prompt.

Interface Synthesis Lab 4: Implementing AXI4 Interfaces

Introduction

This exercise explains how to specify AXI4 bus interfaces for the I/O ports. In addition to adding AXI4 interfaces this exercise also shows how to create an optimal design by using interface and logic directives together.

Step 1: Create and Open the Project

1. From the Vivado HLS command prompt window used in the previous lab, change to the lab4 directory.
2. Create a new Vivado HLS project by typing `vivado_hls –f run_hls.tcl`
3. Open the Vivado HLS GUI project by typing `vivado_hls –p axi_interface_prj`
4. Open the source code as shown in Figure 84.
This design uses similar source C code as Lab 3: with the design renamed to `axi_interfaces`.

**Step 2: Create an Optimized Design with AXI4 Stream Interfaces**

In the optimal performance implementation of this design, the data for each channel would be processed in parallel, with dedicated hardware for each channel.

The key to understanding how best to perform this optimization is to recognize that the channels in the input and output arrays lend themselves to cyclic partitioning. Cyclic partitioning is fully explained in the *Vivado HLS User Guide* (UG902, but basically means each array element is, in turn, sorted into a different partition.

In this exercise, you specify the array arguments to be implemented as AXI4 Stream interfaces. If the arrays are partitioned into channels, you can stream the samples for each channel through the design in parallel.

Finally, if the I/O ports are configured to supply and consume individual streams of channel data, partial unrolling of the for-loop can ensure dedicated hardware processes each channel.

First, partition the arrays:

1. Ensure the C source code is visible in the Information pane.
2. In the Directives tab, select d_o and right-click to open the Directives Editor dialog box.
a. Select the Directives drop-down menu at the top and select ARRAY_PARTITION.
b. Click the Type drop-down menu to specify cyclic partitioning.
c. In the Factor dialog box, enter the value 8, to create eight separate partitions. (This results in eight ports.)
d. With the Directives Editor dialog box filled in as shown in Figure 85, click OK.

![Figure 85: Directives Editor for Cyclic Partitioning](image)

3. In the Directives tab, select d_o again and right-click to open the Directives Editor dialog box.
   a. Activate the Directives drop-down menu at the top and select INTERFACE.
   b. Click the Mode drop-down menu to specify an axis interface.
   c. Click OK.
4. In the Directives tab, select d_i and repeat steps 2 and 3 above.
   a. Apply cyclic partitioning with a factor of 8.
   b. Apply an axis interface.
5. Next, partially unroll and pipeline the for-loop:
   a. In the Directives tab, select For_Loop and right-click to open the Directives Editor dialog box.
   b. Activate the Directives drop-down menu at the top and select UNROLL.
      i. Select a factor of 8 to partially unroll the for-loop. This is equivalent to re-writing the C code to execute eight copies of the loop-body in each iteration of the loop (where the new loop only executes for four iterations in total, not 32).
      ii. Click OK.
   c. In the Directives tab, select For_Loop again and right-click to open the Directives Editor dialog box.
      i. Activate the Directives drop-down menu at the top and select PIPELINE.
      ii. Leave the Interval blank and let it default to 1.
      iii. Select enable loop rewinding.
      iv. Click OK.

When the top-level of the design is a loop, you can use the pipeline rewind option. This informs Vivado HLS that when implemented in RTL, this loop runs continuously (with no end of function and function re-start cycles).

After performing the above steps, the Directives tab should be as shown in Figure 86. Be sure to check all options are correctly applied. If not, double-click the directive to re-open the Directives Editor.

![Figure 86: Directives tab for Lab 4 Solution1](image)

6. Synthesize the design.
When the report opens in the information pane, confirm both d_i and d_o are implemented as eight separate AXI4 Stream ports.

7. In the performance section of the design, confirm that the for-loop processes one sample every clock cycle (Interval 1) with a latency of 3, and that the design has less area than solutions 2, 3, or 4 in Lab 3 (Figure 83).

Cyclic partitioning of the array interfaces and partial for-loop unrolling has allowed implementation of this C code as eight separate channels in the hardware.

Step 3: Implementing an AXI4-Lite Interfaces

In this exercise, you group block-level I/O protocol ports into a single AXI4 Lite interface, which allows these block-level control signals to be controlled and accessed from a CPU.

1. Select New Solution from the toolbar or the Project menu to create and new solution.
2. Accept the defaults and click Finish. This includes copying existing directives from solution1.
3. Ensure the C source code is visible in the Information pane.
4. In the Directives tab, select the top-level function axi_interfaces and right-click to open the Directives Editor dialog box.
   a. Activate the Directives drop-down menu at the top and select INTERFACE.
   b. Activate the mode drop-down menu and select s_axilite. This specifies the ports associated with the function return (the block-level I/O ports) are implemented as an AXI4Lite interface. Since the default mode for the function return is ap_hs, there is requirement to specify this I/O protocol.
   c. Click OK.

The Directives tab appears, as shown in Figure 87.

![Figure 87: Directives for Specifying AXI4 Interfaces](image)
5. Synthesize the design.  
   When the report opens, only the RTL ports for the AXI4 Slave Lite interface appear in the Interface summary.

6. Select Export RTL from the toolbar or the Solution menu, to create an IP package.

7. Leave the Format Selection as IP Catalog and click OK.
You can see the IP package in the `solution2/impl` folder (Figure 88). Because you used the Vivado IP Catalog format, the package is in the `ip` folder.

Figure 88: IP Package with AXI4 Interfaces
The `ip` folder includes the `drivers` subfolder, as shown in Figure 88.

When you add an AXI4-Lite interface to the design, the IP packaging process also creates software driver files to enable an external block, typically a CPU, to control this block (start it, stop it, set port values, review the interrupt status).

8. Double-click the `xaxi_interfaces_hw.h` file to open it in the Information pane.
This shows the addresses to access and control the block-level interface signals. For example, setting control register 0x0 bit 0 to the value 1 will enable the ap_start port, or alternatively, setting bit 7 will enable the auto-restart and the design will re-start automatically at the end of each transaction.

The remaining C driver files are used to integrate control of the AXI4 Slave Lite interface into the code running on a CPU or microcontroller and are included in the packaged IP.

Figure 89: IP HDL with AXI4 Interfaces
Conclusion

In this tutorial, you learned:

- What block-level I/O protocols are and how to control them.
- How to specify and apply port-level I/O protocols.
- How to specify array ports as RAM and FIFO interfaces.
- How to partition RAM and FIFO interfaces into sub-ports.
- How to use both I/O directives and optimization directives to create an optimal design with AXI4 interfaces.
Chapter 5  Arbitrary Precision Types

Overview

C/C++ provided data types are fixed to 8-bit boundaries:

- char (8-bit)
- short (16-bit)
- int (32-bit)
- long long (64-bit)
- float (32-bit)
- double (64-bit)
- Exact width integer types such as int16_t (16-bit) and int32_t (32-bit)

When creating hardware, it is often the case that more accurate bit-widths are required. Consider, for example, a case in which the input to a filter is 12-bit and the accumulation of the results only requires a maximum range of 27 bits. Using standard C data types for hardware design results in unnecessary hardware costs. Operations can use more LUTs and registers than needed for the required accuracy, and delays might even exceed the clock cycle, requiring more cycles to compute the result.

Vivado High-Level Synthesis (HLS) provides a number of bit-accurate or arbitrary precision data types, allowing you to model variables using any (arbitrary) width.

This tutorial consists of a two lab exercises:

- Lab1 - Synthesize a design using floating-point types and review the results. The design uses standard C++ floating-point types.
- Lab2 - Synthesize the same function used in Lab 1 using arbitrary precision fixed-types highlighting the benefits in accuracy and results. This exercise shows how this same design can be converted to the Vivado HLS ap_fixed types, retaining the required accuracy but creating a more optimal hardware implementation

Tutorial Design Description

Download the tutorial design file from the Xilinx website. See the information in
Obtaining the Tutorial Designs. This tutorial uses the design files in the tutorial directory Vivado_HLS_Tutorial\Arbitrary_Precision.

Arbitrary Precision: Lab 1

Arbitrary Precision Lab 1: Review a Design using Standard C/C++ types

In this lab, you synthesize a design using standard C types. You use this design as a reference for the design using arbitrary precision types, which is the basis for Lab 2.

**IMPORTANT:** The figures and commands in this tutorial assume the tutorial data directory Vivado_HLS_Tutorial is unzipped and placed in the location C:\Vivado_HLS_Tutorial.

If the tutorial data directory is unzipped to a different location, or on Linux systems, adjust the few pathnames referenced, to the location you have chosen to place the Vivado_HLS_Tutorial directory.

Step 1: Create and Open the Project

1. Open the Vivado HLS Command Prompt.
   a. On Windows use Start > All Programs > Xilinx Design Tools > Vivado 2014.1 > Vivado HLS > Vivado HLS 2014.1 Command Prompt (Figure 91).
   b. On Linux, open a new shell.

   ![Vivado HLS Command Prompt](Image)

   Figure 91: Vivado HLS Command Prompt
2. In the command prompt window (Figure 92), change the directory to the Arbitrary Precision tutorial, lab1.

3. Execute the Tcl script to setup the Vivado HLS project, using the command as shown in Figure 92:
   
   ```
   vivado_hls -f run_hls.tcl
   ```

   ![Figure 92: Setup the Tutorial Project](image)

4. When Vivado HLS completes, open the project in the Vivado HLS GUI using the command vivado_hls -p window_fn_prj as shown in Figure 93.

   ![Figure 93: Initial Project for Arbitrary Precision Lab 1](image)

### Step 2: Review Test Bench and Run C Simulation

1. Open the Source folder in the explorer pane and double-click `window_fn_top.cpp` to open the code as shown in Figure 94.
2. Hold down the Control key and click the `window_fn_top.h` on line 45 to open this header file.

3. Scroll down to view the type definitions (Figure 95).

![Figure 95: Type Definitions for C Validation Lab 3](image)

This design uses standard C/C++ floating-point types for all data operations. Vivado High-Level Synthesis can synthesize floating-point types directly into hardware, provided the operations are standard arithmetic operations (+, -, *, % etc.).

When using math functions from math.h or cmath.h, refer to the Vivado HLS User Guide (ug902) for details on which math functions are supported for synthesis.

4. Click the **Run C Simulation** toolbar button to open the C Simulation Dialog box

5. Accept the default setting (no options selected) and click **OK**.
The Console pane shows that the design simulates with the expected results.

**Step 3: Synthesize the Design and Review Results**

1. Click the **Run C Synthesis** toolbar button to synthesize the design to RTL.

   When synthesis completes, the synthesis report opens automatically. **Figure 96** shows the synthesis report.

   ![Figure 96: Synthesis Report for Floating Point Design](image)

   Instances in the top-level design account for most of the area used.

2. Scroll down the report and expand the Instances in the Details section of the Area Estimates (**Figure 97**).
The details show this is a floating-point multiplier (fmul). Floating-point operations are costly in terms of area and clock cycles. The Analysis perspective (Figure 98) shows this operator is also responsible for most of the clock cycles (five of the eight states it takes to execute the logic created by loop winfn).

More details on using the Analysis perspective are available in the tutorial Design Analysis. For the purposes of understanding this design, two of the operations in the first state are two-cycle read-from-memory operations, and the operation in the final state is a write-to-memory operation.

3. Exit the Vivado HLS GUI and return to the command prompt.
Arbitrary Precision: Lab 2

Review a Design using Arbitrary Precision types

Introduction

This lab exercise uses the same design as Lab 1, however, the data types are now arbitrary precision types. You first review the design and then examine the synthesis results.

Step 1: Create and Simulate the Project

1. From the Vivado HLS command prompt used in Lab 1, change to the lab2 directory as shown in Figure 99.

2. Create a new Vivado HLS project by typing `vivado_hls –f run_hls.tcl`

3. Open the Vivado HLS GUI project by typing `vivado_hls –p window_fn_prj`.

Figure 99: Setup for Interface Synthesis Lab 2
4. Open the Source folder in the explorer pane and double-click `window_fn_top.cpp` to open the code as shown in Figure 100.

![Figure 100: C Code for Arbitrary Precision Lab 2](image)

5. Hold the Control key down and click `window_fn_top.h` on line 45 to open this header file.

6. Scroll down to view the type definitions (Figure 101).

![Figure 101: Type Definitions for Arbitrary Precision Lab 2](image)

This header file, `window_fn_top.h`, is the only file that is different from Lab 1. The data types have been changed to `ap_fixed` point types, which are similar to float and double types in that they support integer and fractional bit representations. These data types are defined in the header file `ap_fixed.h`. The definitions in the header file define sizes of the data types:

- The first term defines the total word length.
- The Second term defines the number of integer bits.
- The number of fractional bits is therefore the first term minus the second.
When you revise C code to use arbitrary precision types instead of standard C types, one of the most common changes you must make is to reduce the size of the data types. In this case, you change the design to use 8-bit, 24-bit, and 18-bit words instead of 32-bit float types. This results in smaller operators, reduced area, and fewer clock cycles to complete.

Similar optimizations help when you change more common C types such as int, short, and char. For example, changing a data type that only needs to be 18-bit from int (32-bit) ensures that only a single DSP48 is required to perform any multiplications.

In both cases, you must confirm that the design still performs the correct operation and that it does so with the required accuracy. The benefit of the arbitrary precision types provided with Vivado High-Level Synthesis is that you can simulate the updated C code to confirm its function and accuracy.

7. Open the Test Bench folder in the Explorer pane and double-click window_fn_top_test.cpp to open the code.

8. Scroll down to see the view shown in Figure 102.

The test bench for this design contains code to check the accuracy of the results. The expected results are still generated using float types. The result checking verifies that the results are within a specified range of accuracy (in this case, within 0.001 of the expected result).

This allows the updated design to be validated quickly and efficiently in C, with fast compile and run times.

9. Click the Run C Simulation toolbar button to open the C Simulation Dialog box

10. Accept the default setting (no options selected) and click OK.

The Console pane shows the results of the C simulation. With the updated data types, the results are no longer identical to the expected results. However, they are within tolerance.
Step 2: Synthesize the Design and Review Results

1. Click the Run C Synthesis toolbar button to synthesize the design to RTL.

   When synthesis completes, the synthesis report opens automatically. Figure 104 shows the synthesis report.

![C Simulation Results for Fixed Point Types](image)

**Figure 103: C Simulation Results for Fixed Point Types**
Figure 104: Synthesis Report for Fixed Point Design

Note that through use of arbitrary precision types, you have reduced both the latency and the area (by 25% and 60% respectively), and the operations in the RTL hardware are no larger than necessary.

2. Scroll down the report to the Interface summary (Figure 105).

Figure 105 shows the data ports are now 8-bit and 24-bit.
3. **Exit** the Vivado HLS GUI and return to the command prompt.

**Conclusion**

In this tutorial, you learned:

- How to update the existing standard C types to Vivado High-Level Synthesis arbitrary precision types.
- The advantages in terms of hardware performance and area of using bit-accurate data-types.
Overview

The general design methodology for creating an RTL implementation from C, C++ or SystemC includes the following tasks:

- Synthesizing the design.
- Reviewing the results of the initial implementation.
- Applying optimization directives to improve performance.

You can repeat the steps above until the required performance is achieved. Subsequently, you can revisit the design to improve area.

A key part of this process is the analysis of the results. This tutorial explains how to use the reports and the GUI Analysis perspective to analyze the design and determine which optimizations to apply.

This tutorial consists of a single lab exercise that:

- Demonstrates the HLS interactive analysis feature
- Takes you through one design from the initial implementation through six steps and multiple optimizations to produce the final optimized design

As demonstrated throughout the tutorial, performing these steps in a single project gives you the ability to compare the different solutions easily.

Lab1

Synthesize and analyze a DCT design. Use the insights from the design analysis to apply optimizations and judge the effectiveness of the optimization.

Tutorial Design Description

You can download the tutorial design file from the Xilinx Website. Refer to the information in Obtaining the Tutorial Designs.

This tutorial uses the design files in the tutorial directory Vivado_HLS_Tutorial\Design_Analysis.
The sample designs used in the lab exercise is a 2-D DCT function. To highlight the design analysis feature, your goal is to have this design operate with an interval of 100 or less. The design should be able to process a new set of input data at least every 100 clock cycles.

Lab 1: Design Optimization

This exercise explains the basic operations of the GUI Analysis perspective and how you can use it to drive design optimization.

**IMPORTANT:** The figures and commands in this tutorial assume the tutorial data directory Vivado_HLS_Tutorial is unzipped and placed in the location C:\Vivado_HLS_Tutorial.

If the tutorial data directory is unzipped to a different location, or if it is on a Linux system, adjust the few pathnames referenced to the location at which you placed the Vivado_HLS_Tutorial directory.

Step 1: Create and Open the Project

1. Open the Vivado HLS Command Prompt.
   
a. On Windows click **Start > All Programs > Xilinx Design Tools > Vivado 2014.1 > Vivado HLS > Vivado HLS 2014.1 Command Prompt** (Figure 106).

   b. On Linux, open a new shell.

   ![Vivado HLS Command Prompt](image)

   **Figure 106: Vivado HLS Command Prompt**

2. Using the command prompt window (Figure 107), change the directory to the Design Analysis tutorial, lab1.

3. Execute the Tcl script to setup the Vivado HLS project, using the command `vivado_hls -f run_hls.tcl`, as shown in Figure 107.
4. When Vivado HLS completes, open the project in the Vivado HLS GUI using the command `vivado_hls -p dct_prj` as shown in Figure 108.

Step 2: Review the source Code and Create the Initial Design

1. Double-click the file `dct.cpp` in the Source folder to open the source code for review.

   This example uses a DCT function. Figure 109 shows an overview of this code.
The left side of **Figure 109** shows the code hierarchy.
- Top-level function dct has three sub-functions: read_data, dct_2d and write_data.
- Function dct_2d has a single sub-function dct_1d.

The center of **Figure 109** shows loops inside each of the functions.

The right side of **Figure 109** shows the how the data is processed through the functions and loops.
- The read_data function executes, and the data is processed through loop RD_Loop_Row, which has a sub-loop RD_Loop_Col.
- After the read_data function completes, function dct_2d executes.
- In function dct_2d, Row_DCT_Loop processes the data. Row_DCT_Loop has two nested loops inside it: DCT_output_loop and DCT_inner_loop.
- DCT_inner_loop calls function dct_1d.

And so on, until the function write_data processes the data.

2. Click the **Run C Synthesis** toolbar button to synthesize the design to RTL.
Step 3: Review the performance using the Synthesis Report

When synthesis completes, the synthesis report opens automatically. Figure 110 shows the performance section of the report.

Figure 110: Report for initial DCT Design

Figure 110 highlights the following information.

- The clock frequency of 8 ns has been met.
- The top-level design takes 3959 clock cycles to write all the outputs.
- You can apply new inputs after 3960 clock cycles. This is one clock cycle after the output data has been written. This immediately reveals that the design is not pipelined, but this fact is also noted in the report: type is set to none and not pipelined.
- The top level has a single instance, which has a latency and initiation interval of 3668.
  - This block also has no pipelining and accounts for most of the clock cycles.
- Notice that the functions read_data and write_data are not noted here as instances of the top level.
Figure 111 shows that, during synthesis, these blocks were automatically inlined (the hierarchy was removed).

High-level synthesis might automatically inline small functions to improve the quality of results (QoR). You can prevent this by adding the Inline directive with the -off option the function.

- The loops in the read_data and write_data functions are therefore implemented at the top level and are reported as loops in the top-level function (Figure 110).
- Each loop has a latency of 144 clock cycles. (Because the loops are not pipelined, there is no initiation interval.)
- Using RD_Loop_Row as an example, you can see why the loop latency is 144.
  - Sub-loop RD_Loop_Col has a latency of 2 cycles for each iteration of the loop (iteration latency) and a tripcount of 8: \(2 \times 8 = 16\) clock cycles total latency for the loop.
  - From RD_Loop_Row, it takes 1 clock to enter loop RD_Loop_Col and 1 clock cycle to return to RD_Loop_Row. The iteration latency for RD_Loop_Row is therefore \(1 + 16 + 1\) 18 clock cycles.
  - RD_Loop_Row has a tripcount of 8 so the total loop latency is \(8 \times 18 = 144\) clock cycles.
- The total latency for the dct block is therefore:
  - 144 clocks for RD_Loop_Row.
  - Plus 3668 clock cycles for dct_2d.
  - Plus 144 clock cycles for WR_Loop_Row.
  - Plus a clock cycle to enter each block.

To review the details of the instantiated sub-blocks dct_2d and dct_1d, open their respective reports from the syn/reports folder under solution1 in the Explorer pane.

You can also use the design analysis perspective to review these details in a more interactive manner.
Step 4: Review the Performance using the Analysis Perspective

Invoke the Analysis perspective any time after synthesis completes.

1. Click the Analysis perspective button (Figure 112) to begin interactive design analysis.

![Figure 112: Opening the Analysis perspective](image)

The Analysis perspective consists of five panes, each of which is highlighted in Figure 113. You use all of these in the tutorial. The module and loops hierarchies are shown expanded (by default, they are shown collapsed).

![Figure 113: Overview of the Analysis perspective](image)

Use the Module Hierarchy pane to navigate through the hierarchy. The Module Hierarchy pane shows both the performance and area information for the entire design. The Performance Profile pane shows the performance details for this level of hierarchy. The information in these two panes is similar to the information you reviewed earlier in the report (for the top-level dct block).

The Performance view is also shown (on the right side of Figure 113). This view shows how the operations in this particular block are scheduled into clock cycles.

- The left column lists the resources.
Sub-blocks are green.
Operations resulting from loops in the source code are yellow.
Standard operations are purple.

- Notice that the dct has three main resources:
  - A loop called RD_Loop_Row. The plus symbol (+) indicates that the loop has hierarchy and that you can expand the loop to view it.
  - A sub-block called dct_2d.
  - A loop called WR_Loop_Row.

The top row lists the control states in the design. Control states are the internal states High-Level Synthesis uses to schedule operations into clock cycles. There is a close correlation between the control states and the final states in the RTL Finite State Machine (FSM), but there is no one-to-one mapping.

2. Click loop **RD_Loop_Row** and sub-loop **RD_Loop_Col** to fully expand the loop hierarchy **(Figure 114)**.

![Figure 114: Expanded View of RD_Loop_Row](image)

From this, you can see that in the first state (C1) of the RD_Loop_Row, the loop exit condition is checked and an add operation performed. This addition is likely the counter for the loop iterations, and we can confirm this.

3. Select the adder in state C1, right-click and select **C source** code **(Figure 115)**.
This opens the C source code to highlight which operation in the C source created this adder. From the details on screen (also shown in Figure 115), you can determine it is indeed the loop counter. It is the only addition on this line, and the variable is named “r”.

![Figure 115: C Source Code View](image)

In the next state of loop RD_Loop_Row (state C2), loop RD_Loop_Col starts to execute.

4. Click on any of the operations in the RD_Loop_Col to see the source code highlighting update.

This should help confirm your understanding of how the operations in the C source code are implemented in the RTL.

- The loop exit condition is checked.
- This is an adder for loop count variable “c”.
- A read from a RAM performed (one cycle to generate the address, one cycle to read the data).
- A write operation is performed to a RAM.

Loops in the Performance view mean that the design iterates around these states multiple times. The number of iterations is noted as the loop tripcount and shown in the Performance Profile.

To improve performance, these loops should be pipelined. You can review the rest of the design for other performance optimization opportunities.

5. Click on the X in the C Source pane tab to close this window.

6. In the Module Hierarchy pane, click the function dct_2d to navigate into the view for this function (Figure 116).
Again, you can see a number of loops (shown in yellow in Figure 116). Loops ensure the design will have small area but the design will take multiple iterative states to complete: each iteration of the loop will complete before the next iteration starts.

You can pipeline the loops to improve the performance. The details in the Performance Profile show that most of the latency is caused by loops Row_DCT_Loop and Col_DCT_Loop.

7. Click loops Row_DCT_Loop and Col_DCT_Loop in the performance viewer to fully expand them, as shown in Figure 117.

Expanding these loops in Performance view shows both loops call function dct_1d. Unless this function itself is pipelined, there is no benefit in pipelining the loop. The Module Hierarchy shows the interval for dct_1d is 210 clock cycles, which means it can only accept a new input every 210 clock cycles.

8. In the Module Hierarchy, click function dct_1d to navigate into the view for this function.

9. Expand the loops in the Performance Profile and Performance view to see the view shown in Figure 117.
In Figure 117 you can see a series of nested loops which can be pipelined.

You can choose to do one of the following:

- You can pipeline the function and then pipeline the loop that calls it. (Because the function is pipelined, the loop can take advantage of using a pipelined part.)
- You can pipeline the loops within this function and simply make this function execute faster.

Pipelining the function unrolls all the loops within it, and thus greatly increases the area. If the objective is to get the highest possible performance with no regard for area, this may be the best optimization to perform.

You can find more details on pipelining loops and functions in the tutorial Design Optimization. For this case, the approach is to optimize the loops and keep the area at a minimum.

10. Click the Synthesis perspective button to return to the main synthesis view.
Step 5: Apply Loop Pipelining & Review for Loop Optimization

In this step, you create a new solution and add pipelining directives to the loops.

When pipelining nested loops, it is generally best to pipeline the inner-most loop. Typically, High-Level Synthesis can generally flatten the loop nest automatically (allowing the outer loop to simply feed the inner loop). For more information on why it is better to perform certain loop optimizations rather than others, refer to the tutorial “Design Optimization”.

1. Select the New Solution toolbar button or use the menu Project > New Solution to create a new solution.
2. Click Finish and accept the defaults.
3. Ensure that you can see the C source code in the Information pane.
4. In the Directives tab, add a pipeline directive to loop DCT_Inner_Loop in function dct_1d.
   a. Right-click DCT_Inner_Loop in the Directives pane and select Insert Directive
   b. In the Directives Editor dialog box activate the Directives drop-down menu at the top and select PIPELINE.
   c. Click OK and select the default maximum pipeline rate (II=1)
5. Repeat step 4 for the following loops:
   a. In function dct_2d loop Xpose_Row_Inner_Loop
   b. In function dct_2d loop Xpose_Col_Inner_Loop
   c. In function read_data loop RD_Loop_Col
   d. In function write_data loop WR_Loop_Col

The Directive pane shows the following (highlighted) optimization directives applied.
6. Click the Run C Synthesis toolbar button to synthesize the design to RTL.

7. When synthesis completes, use the Compare Reports toolbar button or the menu Project > Compare Reports to compare solutions 1 and 2.

Figure 120 shows the results of comparing solution1 and solution2. Pipelining the loops has improved the latency of the design with an almost 50% reduction in solution2.
Next, you once again open the Analysis perspective, analyze the results, and determine whether or not there are more opportunities to for optimization.

8. Click the Analysis perspective button to begin interactive design analysis.

When the Analysis perspective opens, you can see that the majority of the latency is still due to block dct_2d. Before proceeding to analyze further, you can review how the loops at this level have been optimized.

The Performance Profile (Figure 121) shows that the latency of both loops has been reduced from 144 clock cycles in solution1 to only 65 clock cycles.

Pipelining loops transforms the latency from

\[
\text{Latency} = \text{iteration latency} \times (\text{tripcount} \times \text{interval})
\]

to

\[
\text{Latency} = \text{iteration latency} + (\text{tripcount} \times \text{interval})
\]

HLS also made this possible by automatically performing loop flattening (there is no longer any loop hierarchy). You can see this by reviewing the Console pane, or log file, for solution2. Figure 122 shows the loops that have been automatically optimized.
9. In the Module Hierarchy, click function dct_2d to navigate into the view for this function.

In the Performance Profile you can see that the latency of all the loops has been substantially reduced (Row_DCT_Loop and Col_DCT_Loop have been approximately halved from the earlier report in Figure 116). However, the majority of the latency is still due to these two loops, each of which calls the dct_1b block.

10. In the Module Hierarchy, click function dct_1d to navigate into the view for this function.

The Performance Profile (Figure 123) shows the loop latencies have been reduced, but there is still a loop hierarchy here. (There is still loop DCT_Outer_Loop, shown in Figure 123, so no loop flattening occurred).

Viewing these loops in Performance view shows why this loop was not optimized further.

11. In the Performance view, click loops DCT_Outer_Loop and DCT_Inner_Loop to view the loop hierarchy (Figure 124).

12. Select the write operation in state C5.

13. Right-click and select Goto Source.
Figure 124 shows that this loop was not flattened because additional operations outside of DCT_Inner_Loop, at the level of DCT_Outer_Loop, prevented loop flattening. One of the operations that prevented loop flattening is highlighted in Figure 124, below.

The write to the array cannot be flattened into the inner loop. To achieve an interval of 1 on DCT Outer Loop you will need to pipeline the output loop - there is no benefit in simply pipelining the inner loop itself.

You should pipeline the outer loop instead. This causes the inner loop to be completely unrolled. An increase in area results, but you are still far from the throughput goal of 100 and not yet ready to pipeline the entire function (and see an even greater area increase, as the outer loop is also completely unrolled).

14. Click the Synthesis perspective button to return to the main synthesis view.

Step 6: Apply Loop Optimization and Review for Bottlenecks

1. Select the New Solution toolbar button or use the menu Project > New Solution to create a new solution.

2. Click Finish and accept the defaults to create solution3.

3. Ensure the C source code is visible in the Information pane.

4. In the Directives tab
   - a. In function dct_1d, select the pipeline directive on loop DCT_Inner_Loop.
   - b. Right-Click and select Remove Directive.
   - c. Still in function dct_1d, select loop DCT_Outer_Loop.
d. Right-click and select **Insert Directive**.

 e. In the **Directives Editor** dialog box activate the **Directives** drop-down menu at the top and select **PIPELINE**.

 f. Click **OK** and select the default maximum pipeline rate (II=1).

 The Directive pane should show the following (highlighted) optimization directives applied.

 ![Image of Updated Optimization Directives for DCT Loop Pipelines]

**Figure 125: Updated Optimization Directives for DCT Loop Pipelines**

5. Click the **Run C Synthesis** toolbar button to synthesize the design to RTL.

6. When synthesis completes, click the **Compare Reports** toolbar button to compare solutions 2 and 3.

   **Figure 126** shows the results of comparing solution2 and solution3. Pipelining the outer-loop has in fact resulted in an increase to the performance and the area.

   The significant latency benefit is achieved because multiple loops in the design call the `dct_1d` function multiple times. Saving latency in this block is multiplied because this function is used inside many loops.
Now that all the loops are pipelined, it is worthwhile to review the design to see if there are performance-limiting “bottlenecks.” Bottlenecks are limitations in the flow of data that can prevent the logic blocks from working at their maximum data rate.

Such limitations in the data flow can come from a number of sources, for example, I/O ports and arrays implemented as block RAM. In both cases, the finite number of ports (on the I/O or block RAM) limits the rate at which data can be read or written.

Another source of bottlenecks is data dependencies in the original source code. In some cases, these data dependencies are inherent in how the algorithm operates, as when a calculation cannot be performed until an earlier calculation has completed. Sometimes, however, the use of an optimization directive or a minor change to the C code can remove them.

The first task is to identify such issues in the RTL design. There are a number of approaches you can take:

- Start with the largest latency of interval in the Module Hierarchy report and navigate down the hierarchy to find the source of any large latency or interval.
- Click the Resource Profile to examine I/O and memory usage.
• Use the power of the graphical viewer and look for patterns in the Performance view which indicate a limitation in data flow.

In this case, you will use the latter approach. You can use the Analysis perspective to identify such places in the design quickly.

7. Click the **Analysis** perspective button to begin interactive design analysis.

8. In the **Module Hierarchy**, ensure **module dct** is selected.

9. In the **Performance** view, expand the first loop in the design as shown in Figure 127, **RD_Loop_Row_RD_Loop_Col** (these loops were flattened and the name is now a concatenation of both loops).

This loop is implemented in two states. The red arrow in Figure 127 shows the path from the start of the loop to the end of the loop: the arrow is almost vertical (everything happens in two clock cycles) and this loop is well implemented in terms of latency.

10. In the **Performance** view, expand the **WR_Loop_Row** and perform similar analysis. It is similarly well optimized for latency.

11. **Double-click function dct_2d** and navigate into the **dct_2d** function.

   You can use same analysis process down through the hierarchy. If you perform this analysis you will discover that all the function blocks and loops have a similar optimal (few cycles) implementation, until the dct_1d block is examined.
12. In the **Performance** view, double-click function **dct_1d** and navigate into the **dct_1d** function.

13. Expand the **DCT_Outer_Loop** to see the view shown in **Figure 128**.

**Figure 128** shows a very different view from the earlier loop schedules (which had only a few cycles of latency). The schedule shows a long drift from input to output (as shown by the red arrow).

There are typically two things that cause this type of schedule: data dependencies in the source code and limitations due to I/O or block RAM. You will now examine the resources sharing in this block.

14. In the **Performance** view, click the **Resource tab** at the bottom of the window.
15. Expand the **Memory Ports**, as shown in **Figure 129**.

![Figure 129: Resource Sharing of Memory Ports in dct_1d](image)

The Resource Sharing view shows how the resources in the design are used in different control states.

The rows list the resources in the design. In **Figure 129**, the memory resources are expanded.

The columns show the control states in which the resource is used. If a resource is active in multiple states, the resource is being re-used in different clock cycles.

**Figure 129** shows the memory accesses on BRAM src are being used to the maximum in every clock cycle. (At most, a block RAM can be dual-port and both ports are being used). This is a good indication the design may be bandwidth-limited by the memory resource. To determine if this really is the case, you can examine further.

16. Select one of the read operations for the src block RAM.

17. Right-click and select **Goto Source** to see the view shown in **Figure 130**.
Figure 130 shows this read on the src variable is from the read operation inside loop DCT_Inner_Loop. This loop was automatically unrolled when DCT_Outer_Loop was pipelined and all operations in this loop can occur in parallel (if data dependencies allow).

The eight reads are being forced to occur over multiple cycles because the array src is implemented as a block RAM in the RTL and a block RAM can only allow two reads (maximum) in any one clock cycle. In Figure 130, the read operations take 2 clocks cycles: a cycle to generate the address for the block RAM and a cycle to read the data. Only the launch (address generation cycle) is shown because it overlaps with the operation in the next clock cycle.

You can optimize the block RAM accesses using optimization directives to partition the block RAM. The array that function dct_1d accesses is defined as an input argument to the function and therefore resides outside this block.

- The input array to the first instance of dct_1d is buf_2d_in in function dct.
- The input array to the second instance of dct_1d is col_inbuf in function dct_2d.

In both cases, the arrays are 2-dimensional of size DCT_SIZE by DCT_SIZE (8x8). By default, this results in a single block RAM with 64 elements. Because the arrays are configured in the code in the form of Row by Column, we can partition the 2nd dimension and create eight separate Block RAMs: one for each row, allowing the row data to be accessed in parallel.

18. Click the Synthesis perspective button to return to the main synthesis view.

Step 7: Partition Block RAMs and Analyze Concurrency

1. Select the New Solution toolbar button or use the menu Project > New Solution to create a new solution, solution4.
2. Click Finish and accept the defaults to create solution4.
3. Ensure the C source code is visible in the Information pane.
4. In the Directives tab:
a. In function dct, select array buf_2d_in.
b. Right-click and select **Insert Directive**.
c. In the **Directives Editor** dialog box, activate the **Directives** drop-down menu at the top and select **ARRAY_PARTITION**.
d. Leave the type as **Complete**.
e. Change the dimension setting to 2 to partition the array along the 2\textsuperscript{nd} dimension.
f. Click **OK**.

5. Repeat this process for array **col_inbuf** in function **dct_2d**.

The **Directive** pane displays optimization directives, as shown in Figure 131 (the two new directives are highlighted).

![Figure 131: Optimization Directives for Array Partitioning](image)
6. Click the **Run C Synthesis** toolbar button to synthesize the design to RTL.

7. When synthesis completes, use the **Compare Reports** toolbar button to compare solutions 3 and 4.

   **Figure 132** shows the results of comparing solution3 and solution4. Improving access to the data in the src block RAM in the dct_1d block has improved the overall performance because the dct_1d block executes frequently.

![Figure 132: DCT Solution3 and Solution4 Comparison](image)

You can review the impact of the partitioning directive on the device resource.

8. Click the **Analysis** perspective button to begin interactive design analysis.

9. In the **Module Hierarchy**, ensure module dct is selected.

10. Select the **Resource Profile** in the lower-left by selecting the **Resource Profile** tab.

11. Expand the **Memories and Expressions** see the view in **Figure 133**.
The Resource Profile shows the resources being used at the current level of hierarchy (the block selected in the Module Hierarchy pane). Figure 133 shows:

- This block has two I/O ports.
- Most of the area is due to instances (sub-blocks) within this block.
- There are nine memories, eight of which are the partitioned buf_2d_in block RAM. Since they are less than 1024 bits they are automatically implemented as LUTRAM.
- Most of the logic (expressions) at this level of hierarchy is due to adders, with some due to comparators and selectors.

The important point from the previous optimization is that you can see there are now additional memories due to the array partitioning optimization.

You still have a goal to ensure that the design can accept a new set of samples every 100 clock cycles. Figure 132, however, shows that you can only accept new data every 525 clocks. This is much better than the original, pre-optimized design (approx. 3700 clock cycles), but further optimization is required.
Up to this point, you have focused on improving the latency and interval of each of the individual loops and functions in the design. You must now apply the dataflow optimization, which enables the individual loops and functions to execute in parallel, thus improving the overall design interval.

12. Click the **Synthesis** perspective button to return to the main synthesis view.

### Step 8: Partition Block RAMs and Apply Dataflow optimization

1. Select the **New Solution** toolbar button or use the menu **Project > New Solution** to create a new solution, solution5.
2. Click **Finish** and accept the defaults to create solution5.
3. Ensure the C source code is visible in the Information pane.
4. In the **Directives** tab
   a. Select the top-level function `dct`.
   b. Right-click and select **Insert Directive**.
   c. In the **Directives Editor** dialog box activate the **Directives** drop-down menu and select **DATAFLOW**.
   d. Click **OK**.

The Directive pane now displays the following optimization directives (the new directive is highlighted).
5. Click the Click the **Run C Synthesis** toolbar button to synthesize the design to RTL.

6. When synthesis completes, use the **Compare Reports** toolbar button or the menu **Project > Compare Reports** to compare solutions 4 and 5.

   **Figure 135** shows the results of comparing solution 4 and solution 5, and you can see the interval has improved. The design takes 525 clocks cycles to produce the outputs but can now accept new inputs every 390 clocks.
This is still greater than the 100 cycles required, so you must analyze the current performance.

7. Click the Analysis perspective button to begin interactive design analysis.

8. In the Module Hierarchy, you can see `dct_2d` accounts for most of the interval. Ensure module `dct_2d` is selected to see the view in Figure 136.

Here, you can see two things:

- The interval of the `dct` block is less than the sum of the individual latencies (for `read_data`, `dct_2d` and `write_data`). This means the blocks are operating in parallel.
The interval of dct is the same as the interval for sub-block dct_2d. The dct_2d block is therefore the limiting factor.

Because the dct_2d block is selected in the Module Hierarchy, the Performance Profile shows the details for this block. Figure 136 shows the interval is the same as the latency, so none of these blocks operate in parallel.

One way to have the blocks in dct_2d operate in parallel would be to pipeline the entire function. This, however, would unroll all the loops, which can sometimes lead to a large area increase. An alternative is use dataflow optimization on function dct_2d.

Another alternative is to use a less obvious technique: raise these loops up to the top-level of hierarchy, where they will be included in the dataflow optimization already applied to the top-level. This can be achieved by using an optimization directive to remove the dct_2d hierarchy: inline the dct_2d function.

Before performing this optimization, review the area increase caused by using dataflow optimization.

9. In the Module Hierarchy, ensure module dct is selected.

10. Activate the Resource Profile view.

11. Expand the memories to see the view in Figure 137.

As compared with Figure 133, you can see there are now twice as many memories at this level of hierarchy (the number of banks, flip-flops and LUTs has doubled). Each memory has been
transformed into a Ping-Pong buffer to support dataflow. In this case, no “new” memories were added; the existing memories were converted into dataflow Ping-Pong memory channels. This doubled the number.

12. Click the **Synthesis** perspective button to return to the main synthesis view.

**Step 9: Optimize the Hierarchy for Dataflow**

1. Select the **New Solution** toolbar button to create a new solution, solution6.
2. Click **Finish** and accept the defaults to create solution6.
3. Ensure the C source code is visible in the Information pane.
4. In the **Directives** tab:
   a. Select function `dct_2d`.
   b. Right-click and select **Insert Directive**.
   c. In the **Directives Editor** dialog box activate the **Directives** drop-down menu at the top and select **INLINE**.
   d. Click **OK**.

The Directive pane now shows the following optimization directives (the new directive is highlighted).
5. Click the **Run C Synthesis** toolbar button to synthesize the design to RTL.

6. When synthesis completes, use the **Compare Reports** toolbar button or the menu **Project > Compare Reports** to compare solutions 5 and 6.

   **Figure 139** shows the results of comparing solution5 and solution6. You can see the interval has improved substantially.
The interval is now below the 100 clock target. This design can accept a new set of input data every 71 clock cycles.

You can also see the details (1) in the synthesis report, which opens automatically after synthesis completes and (2) in the Analysis perspective, as shown in Figure 140.
Conclusion

In this tutorial, you learned:

- How to analyze a design using the analysis perspective.
- How to cross-link operations in the views with the C code.
- How to apply and judge optimizations.
- A methodology for taking the initial design results and creating an implementation which satisfies the design goals.
Chapter 7  Design Optimization

Overview

A crucial part of creating high quality RTL designs using High-Level Synthesis is having the ability to apply optimizations to the C code. High-Level Synthesis always tries to minimize the latency of loops and functions. To achieve this, within the loops and functions, it tries to execute as many operations as possible in parallel. At the level of functions, High-Level Synthesis always tries to execute functions in parallel.

In addition to these automatic optimizations, directives are used to:

- Execute multiple tasks in parallel, for example, multiple executions of the same function or multiple iterations of the same loop. This is pipelining.
- Restructure the physical implementation of arrays (block RAMs), functions, loops and ports to improve the availability of data and help data flow through the design faster.
- Provide information on data dependencies, or lack of them, allowing more optimizations to be performed.

The final optimization technique is to modify the C source code to remove unintended dependencies in the code that may limit the performance of the hardware.

This tutorial consists of two lab exercises. You perform the analysis in these lab exercises using the Analysis perspective. A prerequisite for this tutorial is completion of the Design Analysis tutorial.

Lab 1

Contrast the uses of loop and function pipelining to create a design that can process one sample per clock. This lab includes examples that give you the opportunity to analyze the two most common causes for designs failing to meet performance requirements: loop dependencies and data flow limitations or bottlenecks.

Lab 2

This lab shows how modifications to the code from Lab 1 can help overcome some performance limitations inherent, but unintended, in the code.
Tutorial Design Description

You can download the tutorial design file from the Xilinx Website. Refer to the information in Obtaining the Tutorial Designs.

For this tutorial you use the design files in the tutorial directory Vivado_HLS_Tutorial\Design_Optimization.

The sample design you use in the lab exercise is a matrix multiplier function. The design goal is to process a new sample every clock period and implement the interfaces as streaming data interfaces.

Lab 1: Optimizing a Matrix Multiplier

This exercise uses a matrix multiplier design to show how you can fully optimize a design heavily based on loops. The design goal is to read one sample per clock cycle using a FIFO interface, while minimizing the area.

The analysis includes a comparison of a methodology that optimizes at the loop level with one that optimizes at the function level.

IMPORTANT: The figures and commands in this tutorial assume the tutorial data directory Vivado_HLS_Tutorial is unzipped and placed in the location C:\Vivado_HLS_Tutorial.

If the tutorial data directory is unzipped to a different location, or on Linux systems, adjust the few pathnames referenced, to the location you have chosen to place the Vivado_HLS_Tutorial directory.

Step 1: Create and Open the Project

1. Open the Vivado HLS Command Prompt.
   a. On Windows use Start > All Programs > Xilinx Design Tools > Vivado 2014.1 > Vivado HLS > Vivado HLS 2014.1 Command Prompt (Figure 141).
   b. On Linux, open a new shell.

   ![Figure 141: Vivado HLS Command Prompt](image)
2. Using the command prompt window (Figure 142), change directory to the RTL Verification tutorial, lab1.

3. Execute the Tcl script to set up the Vivado HLS project, using the command vivado_hls –f run_hls.tcl, as shown in Figure 142.

4. When Vivado HLS completes, open the project in the Vivado HLS GUI using the command vivado_hls –p matrixmul_prj, as shown in Figure 143.

5. Expand the Sources folder in the Explorer pane and double-click matrixmul.cpp to view the source code (Figure 144).

Scroll down the file to see that the source code has two input arrays, a and b, and output array res. Hold the mouse over the macros (as shown in Figure 144) to see that each is three-by-three for a total of nine elements.
Figure 144: Source Code for the Matrix Multiplier

Step 2: Synthesize and Analyze the Design

1. Click the Run C Synthesis toolbar button to synthesize the design to RTL.

When synthesis completes, the synthesis report opens (Figure 145), and the Performance estimates appears:

- The interval is 80 clock cycles. Because there are nine elements in each input array, the design takes approximately nine cycles per input read.
- The interval is one cycle longer than the latency, so there is no parallelism in the hardware at this point.
- The latency/interval is due to nested loops.
  - The inner loop called Product:
    - Has a latency of 2 clock cycles
    - Has 6 clock cycles total for all iterations.
  - The Col loop:
    - It requires 1 clock to enter loop Product and 1 clock to exit
    - It takes 8 clock cycles for each iteration (1+6+1)
    - Has 24 cycles for all iterations to complete.
  - The top-level loop has a latency of 26 clock cycles per iteration, for a total of 78 clock cycles for all iterations of the loop.
You can do one of two things to improve the initiation interval: Pipeline the loops or pipeline the entire function. You begin by pipelining the loops and then compare those results to pipelining the entire function.

When pipelining loops, the initiation interval of the loops is the important metric to monitor. As seen in this exercise, even when the design reaches the stage at which the loop can process a sample every clock cycle, the initiation interval of the function is still reported as the time it takes for the loops contained within the function to finish processing all data for the function.

**Step 3: Pipeline the Product Loop**

1. Select the **New Solution** toolbar button or use the menu **Project > New Solution** to create a new solution, solution2.

2. Click **Finish** and accept the defaults to create solution2.

3. Ensure the C source code is visible in the Information pane.

When pipelining nested loops, you realize the greatest benefit by pipelining the inner-most loop, which processes a sample of data. High-Level Synthesis automatically applies loop flattening, collapsing the nested loops, removing the loop transitions (essentially creating a single loop with more iterations but overall fewer clock cycles).
4. In the Directives tab:
   a. Select loop Product.
   b. Right-click and select Insert Directive.
   c. In the Directives Editor dialog box, activate the Directives drop-down menu at the top and select PIPELINE.
   d. Click OK. With the default options, an initiation interval (II) of 1 (one new loop iteration per clock) will be the default.

   The Directive pane should show the following optimization directives. (The new directive is highlighted.)

5. Click the Run C Synthesis toolbar button to synthesize the design to RTL.

   During synthesis, the information reported in the Console pane shows loop flattening was performed on loop Row and that the default initiation internal target of 1 could not be achieved on loop Product due to a dependency.

   @I [XFORM-541] Flattening a loop nest 'Row' (matrixmul.cpp:54) in function 'matrixmul'.
   
   @I [SCHED-61] Pipelining loop 'Product'.
   @W [SCHED-68] Unable to enforce a carried dependency constraint (II = 1, distance = 1) between 'store' operation (matrixmul.cpp:60) of variable 'tmp_8' on array 'res' and 'load' operation ('res_load', matrixmul.cpp:60) on array 'res'.
   @I [SCHED-61] Pipelining result: Target II: 1, Final II: 2, Depth: 2.
The synthesis report (Figure 147) shows that although the Product loop is pipelined with an interval of 2, the interval of top-level loop is not pipelined.

![Figure 147: Matrixmul Initial Pipeline Report](image)

The reason the top-level loop is not pipelined is that loop flattening only occurred on loop Row. There was no loop flattening of loop Col into the Product loop. To understand why loop flattening was unable to flatten all nested loops, use the Analysis perspective.

6. Open the **Analysis** perspective.

7. In the **Performance View**, expand loops **Row_Col** and **Product**.

8. Select the **write** operation in state C1.

9. Right-click and select **Goto Source** to see the view in **Figure 148**.

The write operation in state C1 is due to the code that sets res to zero before the Product loop. Because res is a top-level function argument, it is a write to a port in the RTL: This operation must happen before the operations in loop Product are executed. Because it is not an internal operation but has an impact on the I/O behavior, this operation cannot be moved or optimized. This prevents the Product loop from being flattened into the Row_Col loop.
More importantly, it is worth addressing why only an II of 2 was possible for the Product loop.

The message SCHED-68 tells you:

```
@W [SCHED-68] Unable to enforce a carried dependency constraint (II = 1, distance = 1) between 'store' operation (matrixmul.cpp:60) of variable 'tmp_8' on array 'res' and 'load' operation ('res_load', matrixmul.cpp:60) on array 'res'.
```

- The issue is a carried dependency. This is a dependency between an operation in one iteration of a loop and an operation in a different iteration of the same loop. For example, an operation when k=1 and when k=2 (where k is the loop index).
- The first operation is a store (memory read operation) on array res on line 60.
- The second operation is a load (memory write operation) on array res on line 60.

From Figure 148 you can see line 60 is a read from array res (due to the += operator) and a write to array res. An array is mapped into a block RAM by default and the details in the Performance View can show why this conflict occurred.

The Performance view shows in which states the operations are scheduled. Figure 149 shows a number of copies of the schedule for the Product loop to highlight how this issue can be understood. The operations on the res array, a two-cycle read and write, are highlighted.
In the successful schedule, the next iteration of the Product loop appears as shown below. In this schedule, the initiation interval (II)=2 and the loop operations re-start every two cycles. There is no conflict between any block RAM accesses. (None of the highlighted cells overlap across iterations.)

The unsuccessful schedule shows why the loop cannot be pipelined with an II=1. In this case, the next iteration would need to start after 1 clock cycle. The write to the block RAM in the first iteration is still occurring when the second iteration tries to apply an address for a read operation. These addresses are different. Both cannot be applied to the block RAM at the same time.

Figure 149: Carried Dependency Analysis

You cannot pipeline the Product loop with an initiation interval of 1. The next lab exercise shows how re-writing the code can remove this limitation (any technique that does not write back to the same array/block RAM). In this lab exercise you optimize the code as it is.

The next step is to pipeline the loop above, the Col loop. This automatically unrolls the Product loop and creates more operators and hence more hardware resources, but it ensures there is no dependency between different iterations of the Product loop.

10. Return to the Synthesis perspective.
Step 4: Pipeline the Col Loop

1. Select the New Solution toolbar button to create a new solution, solution3.
2. Because solution2 already has a directive added, use the drop-down menu to select solution1 as the source for existing directives and constraints (solution1 has none).
3. Click Finish and accept the default solution name, solution3.
4. Open the C source code matrixmul.cpp to make it visible in the Information pane.
5. In the Directives tab:
   a. Select loop Col.
   b. Right-click and select Insert Directive
   c. In the Directives Editor dialog box activate the Directives drop-down menu at the top and select PIPELINE.
   d. Click OK. With the default options, an initiation interval (II) of 1 (one new loop iteration per clock) becomes the default.

The Directive pane, shown below, displays the following optimization directives (the new directive is highlighted).

![Figure 150: Col Pipeline Directive](image)

6. Click the Click the Run C Synthesis toolbar button to synthesize the design to RTL.

During synthesis, the information reported in the Console pane shows that loop Product was unrolled, loop flattening was performed on loop Row, and the default initiation internal target of 1 could not be achieved on loop Row_Col due resource limitations on the memory for array a.

@I [XFORM-502] Unrolling all sub-loops inside loop 'Col' (matrixmul.cpp:56) in function 'matrixmul' for pipelining.
@I [XFORM-501] Unrolling loop 'Product' (matrixmul.cpp:59) in function 'matrixmul' completely.
@I [XFORM-541] Flattening a loop nest 'Row' (matrixmul.cpp:54) in function 'matrixmul'.
Reviewing the synthesis report shows, as noted above, that the interval for loop Row_Col is only two: the target is to process one sample every cycle. Once again, you can use the Analysis perspective to highlight why the initiation target was not achieved.

7. Open the Analysis perspective.

8. In the Performance View, expand the Row_Col loop

The operations on array a (mentioned in the SCHED-69 message above) are highlighted in Figure 151. There are three read operations on array a. Two operations start in state C1 and a third read operation starts in state C2.

Arrays are implemented as block RAMs and arrays which are arguments to the function are implemented as block RAM ports. In both cases a block RAM can only have a maximum of two ports (for dual-port block RAM). By accessing array a through a single block RAM interface, there are not enough ports to be able to read all three values in one clock cycle.
Another way to view this resource limitation is to use the Resource pane.

9. Click the **Resource tab**.

10. Expand the memories to see the view shown in **Figure 152**.

In **Figure 152** the 2-cycle read operations in state C1 overlap with those starting in state C2 and so only a single cycle is visible: however, it is clear that this resource is used in multiple states.

In looking at this view, it is clear that even when the issue with port a is resolved, the same issue occurs with port b: it also has to perform 3 reads.

High-Level Synthesis can only report one schedule error or warning at a time, because, as soon as the first issue occurs, the actions to create an achievable schedule invalidates any other infeasible schedules.
High-Level Synthesis allows arrays to be partitioned, mapped together and re-shaped. These techniques allow the access to array to be modified without changing the source code.

11. Return to the **Synthesis** perspective.

### Step 5: Reshape the Arrays

1. Select the **New Solution** toolbar button or use the menu **Project > New Solution** to create a new solution, solution4.

2. Click **Finish** and accept the default solution name solution4.

Because the loop index for the Product loop is k, both arrays should be partitioned along their respective k dimension: the design needs to access more than two values of k in each clock cycle.

For array a, this is dimension 2 because its access patterns is \(a[i][k]\); for array b, this is dimension 1 because its access pattern is \(b[k][j]\).

Partitioning these arrays creates k arrays - in this case, k number ports. Alternatively, we can use re-shape instead of partition allowing one wide array (port) to be created instead of k ports.

After this transformation, the data in the block RAM outside this block must be reshaped in an identical manner: if this process is not done by HLS, the data must be arranged as:

- For array a: i elements, each of width data_word_size times k.
- For array b: j elements, each of width data_word_size times k.

3. Open the C source code **matrixmul.cpp** to make it visible in the Information pane.

4. In the **Directives** tab
   a. Select variable a.
   b. Right-click and select **Insert Directive.**
c. In the **Directives Editor** dialog box activate the **Directives** drop-down menu at the top and select **ARRAY_RESHAPE**.
d. Set the dimension to **2**.
e. Click **OK**.

5. Repeat this process for variable **b**, but set the **dimension to 1**.

The Directive pane should show the following optimization directives.

6. Click the **Run C Synthesis** toolbar button to synthesize the design to RTL.

The synthesis report shows the top-level loop Row_Col is now processing data at 1 sample per clock period (**Figure 154**).
The top-level module takes 12 clock cycles to complete.
The Row_Col loop outputs a sample after 3 cycles (iteration latency).
It then reads 1 sample every cycle (Initiation Interval).
After 9 iterations/samples (Trip count) it completes all samples.
$3 + 9 = 12$ clock cycles

The function can then complete and return to start to process the next set of data.

Now, change the block RAM interfaces to FIFO interfaces to allow for streaming data.

### Step 6: Apply FIFO Interfaces
1. Select the **New Solution** toolbar button to create a new solution.
2. Click **Finish** and accept the default solution name, solution5.
3. Open the C source code `matrixmul.cpp` to make it visible in the Information pane.
4. In the **Directives** tab
   a. Select **variable a**.
   b. Right-click and select **Insert Directive**.
c. In the **Directives Editor** dialog box activate the **Directives** drop-down menu at the top and select **INTERFACE**.

d. Click the **mode** drop-down menu to select **ap_fifo**.

e. Click **OK**.

5. Repeat this process for variables b and variable res.

The Directive pane displays the following optimization directives. (The new directives are highlighted).

![Figure 155: Matrixmul FIFO Directives](image)

6. Click the **Run C Synthesis** toolbar button to synthesizes the design to RTL.

**Figure 156** shows the Console display after synthesis runs.

![Figure 156: FIFO Synthesis Warning](image)

From the code shown in **Figure 157**, array res performs writes in the following sequence (MAT_B_COLS = MAT_B_ROWS = 3):
• Write to [0][0] on line 57.
• Then a write to [0][0] on line 60.
• Then a write to [0][0] on line 60.
• Then a write to [0][0] on line 60.
• Write to [0][1] on line 57 (after index J increments).
• Then a write to [0][1] on line 60.
• Etc.

Four consecutive writes to address [0][0] does not constitute a streaming access pattern; this is random access.

Examining the code in Figure 157 reveals that there are similar issues reading arrays a and b. It is impossible to use a FIFO interface for data access with the code as written. To use a FIFO interface, the optimization directives available in Vivado High-Level Synthesis are inadequate because the code currently enforces a certain order of reads and writes. Further optimization requires a re-write of the code, which you accomplish in Lab 2.

Before modifying the code, however, it is worth pipelining the function instead of the loops to contrast the difference in the two approaches.

**Step 7: Pipeline the Function**

1. Select the **New Solution** toolbar button to create a new solution, solution6.

---

_**IMPORTANT:** In this step, copy the directives from solution4 as this solution does not have FIFO interfaces specified._
2. Select **solution4** from both the drop down menus in the **Options** section. The Solution Wizard appears as shown in **Figure 158**.

![Solution Wizard](image)

**Figure 158: New Solution Based on Solution4 Directives**

3. Click **Finish** and accept the default solution name, **solution6**.

4. Open the C source code `matrixmul.cpp` to make it visible in the Information pane.

5. In the **Directives** tab:
   a. Select the pipeline directive on loop Col.
   b. Right-click and select **Remove Directive**.
   c. Select the top-level function **matrixmul**.
   d. Right-click and select **Insert Directive**.
e. In the Directives Editor dialog box activate the Directives drop-down menu at the top and select PIPELINE.

f. Click OK.

The Directives tab should appear as Figure 159.

![Figure 159: Directives for Solution6](image)

6. Click the Run C Synthesis toolbar button to synthesize the design to RTL.

7. Click the Compare Reports toolbar button.
   b. Add solution6.
   c. Click OK.

The comparison of solutions 4 and 6 is shown in Figure 160.
The design now completes in fewer clocks and can start a new transaction every 5 clock cycles. However, the area and resources have increased substantially because all the loops in the design were unrolled.

@I [XFORM-502] Unrolling all loops for pipelining in function 'matrixmul' (matrixmul.cpp:51).
@I [XFORM-501] Unrolling loop 'Row' (matrixmul.cpp:54) in function 'matrixmul' completely.
@I [XFORM-501] Unrolling loop 'Col' (matrixmul.cpp:56) in function 'matrixmul' completely.
@I [XFORM-501] Unrolling loop 'Product' (matrixmul.cpp:59) in function 'matrixmul' completely.

Pipelining loops allows the loops to remain rolled, thus providing a good means of controlling the area. When pipelining a function, all loops contained in the function are unrolled, which is a requirement for pipelining. The pipelined function design can process a new set of 9 samples every 5 clock cycles. This exceeds the requirement of 1 sample per second because the default behavior of High-Level Synthesis is to produce a design with the highest performance.

The pipelined function results in the best performance. However, if it exceeds the required performance, it might take multiple additional directives to slow the design down. Pipelining loops gives you an easy way to control resources, with the option of partially unrolling the design to meet performance.
Lab 2: C Code Optimized for I/O Accesses

In Lab 1, you were unable to use streaming interfaces. The nature of the C code, which specified multiple accesses to the same addresses, prevented streaming interfaces being applied.

- In a streaming interface, the values must be accessed in sequential order.
- In the code, the accesses were also port accesses, which High-Level Synthesis is unable to move around and optimize. The C code specified writing the value zero to port res at the start of every product loop. This may be part of the intended behavior. HLS cannot simply decide to change the specification of the algorithm.

The code intuitively captured the behavior of a matrix multiplication, but it prevented a required behavior in the hardware: streaming accesses.

This lab exercise uses an updated version of the C code you worked with in Lab 1. The following explains how the C code was updated.

Figure 161 shows the I/O access pattern for the code in Lab 1. Out of necessity the address values are shown in a small font.

As variables i, j and k iterate from 0 to 3, the lower part of Figure 161 shows the addresses generated to read a, b and write to res. In addition, at the start of each Product loop, res is set to the value zero.

![Figure 161: Lab 1 Matrix Multiplier Address Accesses](image)

To have a hardware design with sequential streaming accesses, the ports accesses can only be those shown highlighted in red. For the read ports, the data must be cached internally to ensure the design does not have to re-read the port. For the write port res, the data must be saved into a temporary variable and only written to the port in the cycles shown in red.

The C code in this lab reflects this behavior.

Step 1: Create and Open the Project

1. From the Vivado HLS command prompt used in Lab 1, change to the lab2 directory as shown in Figure 162.
2. Create a new Vivado HLS project by typing vivado_hls –f run_hls.tcl.
3. Open the Vivado HLS GUI project by typing `vivado_hls -p matrixmul_prj`.

4. Open the Source folder in the explorer pane and double-click `matrixmul.cpp` to open the code as shown in **Figure 163**.

![Figure 163: C Code with updated IO accesses](image)

Review the code and confirm the following:
• The directives from Lab 1, including the FIFO interfaces, are specified in the code as pragmas.

• For-loops have been added to cache the rol and column reads.

• A temporary variable is used for the accumulation and port res is only written to when the final result is computed for each value.

• Because the for-loops to cache the row and column would require multiple cycles to perform the reads, the pipeline directive has been applied to the Col for-loop, ensuring these cache for-loops are automatically unrolled.

**Synthesize the design and verify the RTL using co-simulation.**

5. Click the **Run C Synthesis** toolbar button to synthesize the design to RTL.

6. When synthesis completes, use the **Run C/RTL Cosimulation** toolbar button to launch the **Cosimulation Dialog** box.

7. Click **OK** to start RTL verification.

The design has been now been fully synthesized to read one sample every clock cycle using streaming FIFO interfaces.

**Conclusion**

In this tutorial, you:

• Learned how to analyze pipelined loops and understand exactly which limitations prevent optimizations targets from being achieved.

• The advantages and disadvantages of function versus loop pipelining.

• How unintended dependencies in the code can prevent hardware design goals from being realized and how they can be overcome by modifications to the source code.
Overview

The High Level Synthesis tool automates the process of RTL verification and allows you to use RTL verification to generate trace files that show the activity of the waveforms in the RTL design. You can use these waveforms to analyze and understand the RTL output. This tutorial covers all aspects of the RTL verification process.

To perform RTL verification, you use both the RTL output from High-Level Synthesis (Verilog, VHDL or SystemC) and the C test bench. RTL verification is often called “cosimulation” or “C/RTL cosimulation”; because both C and RTL are used in the verification.

This tutorial consists of three lab exercises.

Lab1
Perform RTL verification steps and understand the importance of the C test bench in verifying the RTL.

Lab2
Create RTL trace files and analyze them using the Vivado Design Suite.

Lab3
Create RTL trace files and analyze them using a third-party RTL simulator. This lab requires a license for Mentor Graphics ModelSim simulator. (You can use an alternative, third-party simulator with minor modifications to the steps).

Tutorial Design Description

You can download the tutorial design file from the Xilinx website. Refer to the information in
Obtaining the Tutorial Designs.

This tutorial uses the design files in the tutorial directory `Vivado_HLS_Tutorial\RTL_Verification`.

The sample design used in the lab exercise is a DUC (digital up converter) function. The purpose of this lab is to demonstrate and explain the features of RTL verification. There are no design goals for these lab exercises.

Lab 1: RTL Verification and the C test bench

This exercise explains the basic operations for RTL verification and highlights the importance of the C test bench.

IMPORTANT: The figures and commands in this tutorial assume the tutorial data directory `Vivado_HLS_Tutorial` is unzipped and placed in the location `C:\Vivado_HLS_Tutorial`.

If the tutorial data directory is unzipped to a different location, or on Linux systems, adjust the few pathnames referenced, to the location you have chosen to place the `Vivado_HLS_Tutorial` directory.

Step 1: Create and Open the Project

1. Open the Vivado HLS Command Prompt.
   a. On Windows use Start > All Programs > Xilinx Design Tools > Vivado 2014.1 > Vivado HLS > Vivado HLS 2014.1 Command Prompt (Figure 164).
   b. On Linux, open a new shell.

2. Using the command prompt window (Figure 165), change directory to the RTL Verification tutorial, lab1.

3. Execute the Tcl script to setup the Vivado HLS project, using the command `vivado_hls –f run_hls.tcl`, as shown in Figure 165.
4. When Vivado HLS completes, open the project in the Vivado HLS GUI using the command `vivado_hls -p duc_prj`, as shown in Figure 166.

```
C:\Uivado_HLS_Tutorial>cd RTL_Verification
C:\Uivado_HLS_Tutorial\RTL_Verification>cd lab1
C:\Uivado_HLS_Tutorial\RTL_Verification\lab1>vivado_hls -f run_hls.tcl
```

Figure 165: Setup the RTL Verification Tutorial Project

**Step 2: Perform RTL Verification**

1. Click the **Run C Synthesis** toolbar button to synthesize the design to RTL.

2. When synthesis completes, use the **Run C/RTL Cosimulation** toolbar button (Figure 167) to launch the **Cosimulation Dialog** box.

![Run C/RTL Cosimulation Toolbar button](image)

Figure 167: Run C/RTL Cosimulation Toolbar button

The Cosimulation Dialog box shown in Figure 168 opens.
The drop-down menu allows you to select the RTL simulator for HDL simulation. For this exercise, you use the default Vivado Simulator with Verilog RTL for cosimulation.

3. Click **OK** to start RTL verification.

When RTL Verification completes, the simulation report opens automatically (Figure 169). The report indicates if the simulation passed or failed. In addition, the report indicates the measured latency and interval.
RTL simulation completes in three steps. To better understand how the RTL verification process is performed, scroll up in the console window to confirm that the messages described below were issued.

First, the C test bench is executed to generate input stimuli for the RTL design.

```plaintext
@I [SIM-14] Instrumenting C test bench ...

< C simulation executes to generate input stimuli >
```

At the end of this phase, the simulation shows any messages generated by the C test bench. The output from the C function is not used in the C test bench at this stage, but any messages output by the test bench can be seen in the console.

```plaintext
@I [SIM-302] Generating test vectors ...

*** DUC hardware test PASSED ! ***
```

An RTL test bench with newly generated input stimuli is created and the RTL simulation is then performed.

```plaintext
@I [SIM-333] Generating C post check test bench ...
@I [SIM-12] Generating RTL test bench ...
...
@I [SIM-11] Starting SystemC simulation ...
```

Finally, the output from the RTL simulation is re-applied to the C test bench to check the results. Once again, you can see any message output by the C test bench in the console. Finally, RTL verification issues message SIM-1000 if the RTL verification passed.

```plaintext
SystemC: simulation stopped by user.
@I [SIM-316] Starting C post checking ...

*** DUC hardware test PASSED ! ***

@I [SIM-1000] *** C/RTL co-simulation finished: PASS ***
```

To fully understand why the C test bench should check the results and how message SIM-1000 is generated, you will modify the C test bench.
Step 3: Modify the C test bench

1. Expand the Test Bench folder in the Explorer pane (Figure 170).
2. Double-click duc_test.c to open the C test bench in the Information pane.

![Figure 170: RTL Test bench](image1)

3. Scroll to the end of the file to see the code shown in Figure 171.
4. Edit the return statement to match Figure 171 and ensure the test bench always returns the value 1.

![Figure 171: Modified RTL Test bench](image2)

5. Save the file.
6. Click the Run C Synthesis toolbar button to synthesize the design to RTL.
7. Click the Run C/RTL Cosimulation toolbar button to launch the Cosimulation Dialog box.
8. Leave the Cosimulation options at their default value and click **OK** to execute the RTL cosimulation.

When RTL cosimulation completes, the cosimulation report opens and says the verification has failed (**Figure 172**).

![Cosimulation Report Failure](image)

**Figure 172: Cosimulation Report Failure**

In **Figure 172**, you can see from the message printed to the console (DUC hardware test PASSED) that the results are correct, however, the verification report says the RTL verification failed.

If required, you can confirm the results are correct. To do this, compare the output files created by the RTL simulation with the golden results. The RTL simulation is executed in the simulation directory wrapc, which is inside the solution directory. **Figure 173** shows the solution directory, with the output files highlighted.
RTL Cosimulation only reports a successful verification when the test bench returns a value of 0 (zero). Modifying the test bench to return a non-zero value ensures RTL verification (and C simulation if it was performed) would always report a failure.

To ensure that the RTL results are automatically verified: the C test bench must always check the output from the C function to be synthesized and return a 0 (zero) if the results are correct OR return any other value if they are not correct.

When RTL Verification is performed, the same testing occurs in the test bench, and the output from the RTL block is automatically checked. This is why it is important for the C test bench to check the results and return a zero value only if they are correct (or return a non-zero value if they are incorrect).

9. Exit the Vivado HLS GUI and return to the command prompt.
Lab 2: Viewing Trace Files in Vivado

This exercise explains how to generate RTL trace files and how to view them using the Vivado Design Suite tools.

Step 1: Create an RTL Trace File using Xsim

1. From the Vivado HLS command prompt you used in Lab 1, change to the lab2 directory as shown in Figure 174.
2. Create a new Vivado HLS project by typing vivado_hls –f run_hls.tcl

3. Open the Vivado HLS GUI project by typing vivado_hls –p duc_prj.
4. Click the Run C Synthesis toolbar button to synthesize the design to RTL.
5. Click the Run C/RTL Cosimulation toolbar button to launch the Cosimulation Dialog box. In this case, you will produce a trace file you can open using the Vivado Simulator (Xsim). Therefore explicitly select Xsim.
6. In the Co-simulation Dialog window:
   a. Select Xsim from the Verilog/VHDL Simulator Selector (Figure 175).
   b. De-select SystemC.
   c. Select Verilog.
   d. Activate the Dump Trace drop-down menu and select the all option, to have the options shown in Figure 175.
   e. Click OK to execute RTL cosimulation.
When RTL verification completes, the cosimulation report automatically opens. The report shows that the Verilog simulation has passed (and the measured latency and interval). In addition, because the Dump Trace option was used with the Xsim simulator option and because Verilog was selected, two trace files are now present in the Verilog simulation directory. These are shown highlighted in Figure 176.
The next step is to view the trace files inside the Vivado Design Suite.

7. Exit the Vivado HLS GUI and return to the command prompt.

**Step 2: View the RTL Trace File in Vivado**

1. Launch the Vivado Design Suite (not Vivado HLS):
   a. On Windows use **Start > All Programs > Xilinx Design Tools > Vivado 2014.1 > Vivado 2014.1**
   b. On Linux, type vivado in the shell.

2. In the Vivado Tcl Console, enter the following commands, as shown in **Figure 177**. This example assumes the top-level tutorial directory is C:\Vivado_HLS_Tutorial:
   a.  `cd /Vivado_HLS_Tutorial/RTL_Verification/lab2/duc_prj/solution1/sim/verilog`
   b.  `current_fileset`
c. open_wave_database duc.wdb

d. open_wave_config duc.wcfg

Figure 177: Opening the Trace File in Vivado

You can then view the waveforms in the waveform viewer. Figure 178 shows the zoomed waveforms where the output data ports and their associated I/O protocol signals (output valid signals) are shown highlighted.

Figure 178: Analyzing the RTL Trace File

3. Exit and close the Vivado GUI.
4. Type exit to close the Vivado Tcl command prompt.

---

**Lab 3: Viewing Trace Files in ModelSim**

This exercise explains how you can generate and view RTL trace files and using the Mentor Graphics ModelSim RTL simulator. Other third-party simulators are supported, and similar process can be used if another RTL simulator is selected.

**CAUTION!** This lab exercise requires that the executable for ModelSim is defined in the system search path and that the required license to perform HDL simulation is available on the system.

---

**Step 1: Create an RTL Trace File using ModelSim**

1. From the Vivado HLS command prompt you used in Lab 2, change to the lab3 directory.
2. Create a new Vivado HLS project by typing `vivado_hls –f run_hls.tcl`.
3. Open the Vivado HLS GUI project by typing `vivado_hls –p duc_prj`.
4. Click the **Run C Synthesis** toolbar button to synthesize the design to RTL.
5. Click the **Run C/RTL Cosimulation** toolbar button to launch the Cosimulation Dialog box.

This exercise uses the Mentor Graphics ModelSim RTL simulator. The path to the simulator executable must be set in your system search path.

6. In the Co-simulation Dialog window:
   a. Select **ModelSim** from the Verilog/VHDL Simulator Selector.
   b. Unselect **SystemC**.
   c. Select **VHDL**.
   d. Activate the Dump Trace drop-down menu and select the all option, to have the options shown in Figure 179.
   e. Click **OK** to execute RTL cosimulation.
When RTL verification completes, the cosimulation report automatically opens, showing the VHDL simulation has passed (and the measured latency and interval). In addition, because the Dump Trace option was used with the ModelSim simulator option and because VHDL was selected, a trace file is now present in the VHDL simulation directory. The trace file is shown highlighted in **Figure 180**.
The next step is to view the trace files inside ModelSim.

7. Exit the Vivado HLS GUI and return to the command prompt.

**Step 2: View the RTL Trace File in ModelSim**

1. Launch the Mentor Graphics ModelSim RTL Simulator.
2. Click the menu **File > Open**.
3. Select **Log Files** as the file type (**Figure 181**).
4. Navigate to the VHDL simulation directory and select duc.wlf.
5. Click **Open**.

![Figure 181: ModelSim Open File WLF](image)

6. Add the signals to the trace window and adjust to see a view similar to **Figure 182**.
7. Exit and close the ModelSim RTL simulator.

**Conclusion**

In this tutorial, you learned how to:

- Perform RTL verification on a design synthesized from C and the importance of the test bench in this process.
- Create and open waveform trace files using the Vivado Design Suite.
- Create and open waveform trace files using a third-party HDL simulator (ModelSim) and view the trace file created by RTL verification.
Overview

You can package the RTL from High-Level Synthesis and use it inside IP Integrator. This tutorial demonstrates how to take HLS IP and use it in IP Integrator as part of a larger design.

This tutorial consists of a single lab exercise.

Lab1

Complete the steps to generate two HLS blocks for the IP catalog and use them in a design with Xilinx IP, an FFT. You validate and verify the final design using an RTL test bench.

Tutorial Design Description

You can download the tutorial design file from the Xilinx Website. Refer to the information in
Obtaining the Tutorial Designs.

This tutorial uses the design files in the tutorial directory Vivado_HLS_Tutorial\Using_IP_with_IPI.

The design blocks in this tutorial process the data for a complex FFT.

- The Xilinx FFT IP block only operates on complex data. Although you can perform an FFT of real data on a complex data set with all imaginary components set to zero, it can be done more efficiently by pre-processing the data.

- The front-end HLS block in this lab applies a Hamming windowing function to the 1024 (N) real data samples and sends even/odd pairs to an N/2-point XFFT as though they are complex data.

- The back-end HLS block takes bit-reverse ordered data, puts it in natural order and applies an O(N) transformation to FFT output to extract the spectral data for the N-point real data set. Note, the first output pair packs the 0th and 512th (purely real) spectral data point into the real and imaginary parts, respectively.

- The designs are fully-pipelined, streaming designs for high throughput; intended for continuous processing of data, but with throttling capability (stalls if input stalls).

- AXI4 Streaming interfaces are used to connect all blocks in IP Integrator (IPI).

Lab 1: Integrate HLS IP with a Xilinx IP Block

This lab exercise shows how two HLS IP blocks are combined with a Xilinx IP FFT in IP Integrator and the design verified in the Vivado Design Suite.

IMPORTANT: The figures and commands in this tutorial assume the tutorial data directory Vivado_HLS_Tutorial is unzipped and placed in the location C:\Vivado_HLS_Tutorial.

If the tutorial data directory is unzipped to a different location, or on Linux systems, adjust the few pathnames referenced, to the location you have chosen to place the Vivado_HLS_Tutorial directory.

Step 1: Create Vivado HLS IP Blocks

Create two HLS blocks for the Vivado IP Catalog using the provide Tcl script. The script runs HLS C-synthesis, RTL co-simulation and package the IP for the two HLS designs (hls_real2xfft and hls_xfft2real).

1. Open the Vivado HLS Command Prompt.
   b. On Linux, open a new shell.
2. Using the command prompt window, change the directory to Vivado_HLS_Tutorial\Using_IP_with_IPI\lab1\hls_designs (Figure 184).
3. Type `vivado_hls -f run_hls.tcl` to create the HLS IP (Figure 184).

Figure 184: Create the HLS Design for IPI

When the script completes, there are two Vivado HLS project directories, `fe_vhls_prj` and `be_vhls_prj`, which contain the HLS IP, including the Vivado IP Catalog archives for use in Vivado designs.

- The “front-end” IP archive is located at `fe_vhls_prj/IPXACTExport/impl/ip/`
- The “back-end” IP archive is located at `be_vhls_prj/IPXACTExport/impl/ip/`

The remainder of this tutorial exercise shows how the Vivado HLS IP blocks can be integrated into a design (in IP Integrator) and verified.

**Step 2: Create a Vivado Design Suite Project**

1. Launch the Vivado Design Suite (not Vivado HLS):
   a. On Windows use **Start > All Programs > Xilinx Design Tools > Vivado 2014.1 > Vivado 2014.1**
   b. On Linux, type `vivado` in the shell.
2. From the Welcome screen, click **Create New Project** (Figure 185).
3. Click **Next** on the first page of the **Create a New Vivado Project** wizard.

4. Click the **ellipsis** button to the right of the **Project location text entry box** and browse to the tutorial directory (**Figure 186**).
5. Click **Next** to move to the **Project Type** page of the wizard.
   a. Select **RTL Project**.
   b. Select **Do not specify sources at this time** (if not the default).
   c. Click **Next**.
6. On the Default Part page, under Specify, click **Boards** and select the **ZYNQ-7 ZC702 Evaluation Board**, as shown in Figure 187.
7. On the **New Project Summary Page**, click **Finish** to complete the new project setup. The Vivado workspace populates and appears as shown in **Figure 188**.
Step 3: Add HLS IP to an IP Repository

1. In the Project Manager area of the Flow Navigator pane, click **IP Catalog**.
2. The IP Catalog appears in the main pane of the workspace. Click the **IP Settings** icon.

3. In the IP Settings dialog, click **Add Repository**.

4. In the IP Repositories dialog:
Using HLS IP in IP Integrator

a. Browse to the tutorial directory, Using_IP_with_IPI\lab1.

b. Click the Create New Folder icon.

c. Enter “vivado_ip_repo” in the resulting dialog (Figure 191).

d. Click OK.

e. Click Select to close the IP Repository window.

![Image of IP Repositories dialog]

Figure 191: Create a New IP Repository

5. Back in the IP Setting dialog:

a. Click Add IP.

b. In the Select IP to Add to Repository dialog box, browse to the location of the HLS IP lab1/hls_designs/fe_vhls_prj/IPXACTExport/impl/ip/.

c. Select the xilinx_com_hls_hls_real2xfft_1_0.zip file (Figure 192).

d. Click OK.
6. Follow the same procedure to add the 2nd HLS IP package to the repository: `xilinx_com_hls_hls_xfft2real_1_0.zip`.

7. The new HLS IP should now show up in the IP Setting dialog (Figure 193).

8. Click **OK** to exit the dialog box.
A Vivado HLS IP category now appears in the IP Catalog and, if expanded, the HLS IP displays (Figure 194).

![IP Catalog with HLS IP](image)

**Figure 194: IP Catalog with HLS IP**

**Step 4: Create a Block Design for RealFFT**

1. Click **Create Block Diagram** under IP Integrator in the Flow Navigator.
   a. In the resulting dialog box, name the design `RealFFT`.
   b. Click **OK**.
The upper-right pane now has a Diagram tab. Add a Xilinx FFT IP block to the design and customize it.

2. In the Diagram tab click the Add IP link in the “get started” message (Figure 196).
   a. In the Search box type “fourier”.
   b. Press Enter.
The Xilinx IP block FFT is now instantiated in the design, as shown in **Figure 197**.

3. Double-click the new **Fast Fourier Transform IP Symbol** to open the Re-customize IP dialog box.
4. On the **Configuration** tab (**Figure 198**):
   a. Change the Transform Length to 512.
   b. Select **Pipelined, Streaming I/O** in the **Architecture Choice** section.

   ![Figure 198: Xilinx FFT Configuration](image)

5. Select the **Implementation** tab (**Figure 199**):
   a. Select **ARESETN** (active low) in the Control Signals group.
   b. Verify that **Non Real Time** is selected as Throttle Scheme.
   c. Click **OK** to exit the Re-customize IP dialog box.
Add one instance of each of the HLS generated blocks to the design.

6. Right-click in any space in the canvas and select **Add IP** (Figure 200).
7. Type “hls” into the Search text entry box.
   a. Highlight both IPs (Click the control key and select both)
   b. Press Enter.

The design block now as three IP blocks are shown in Figure 201.

![Figure 201: RealFFT IP Blocks](image)

The next step is to connect HLS blocks to the FFT block and ports.

8. Hover the cursor over the “m_axis_dout” interface connector of Hls_real2xfft block until pencil cursor appears.
   a. Left-click and hold down the mouse button to start a connection.
   b. Drag the connection line to “S_AXIS_DATA” port connector of FFT block and release (when green check mark appears next to it).

9. In a similar fashion, connect the FFT’s “M_AXIS_DATA” interface to the “s_axis_din” interface of the Hls_xfft2real block.

The two connections are shown in Figure 202.
Figure 202: Connecting Ports on the IP Blocks

To create I/O ports for the design, make some external connections.

10. Right-click the "s_axis_din" interface connector on Hls_real2xfft block and select Make External (Figure 203).

Figure 203: Make External Connections
Give the new interface port a clearly unique name.

a. Click **port symbol** to highlight it.
b. In the **External Interface Properties** pane (Figure 204).
c. Double-click in the **Name** text entry box to highlight “s_axis_din”.
d. Type in “real2xfft_din” and press **Enter**.

**IMPORTANT:** Property changes might not take effect if this re-naming step is not done.

11. In a similar manner to the previous step:

a. Make the “m_axis_dout” interface of Hls_xfft2real block external and rename it “xfft2real_dout”
b. Right-click **aclk** connector of Hls_real2xfft block and select **Make External**.
c. Right-click **aresethn** connector of Hls_real2xfft block and select **Make External**.
12. Tie the `ap_start` ports of both HLS blocks high
   a. Right-click canvas, select **Add IP**.
   b. Type “const” into **Search text** entry box.
   c. Select **Constant IP**.
   d. Press **Enter**.
   e. Double-click **Constant IP Symbol** (Figure 205) and verify that the settings for Const Width and Const Val are both ‘1’ and click **OK** to close Re-customize IP dialog box.

   ![Constant IP Properties](image)

   **Figure 205: Constant IP Properties**

   f. Connect `ap_start` of both HLS blocks to the Constant block (Figure 206).
13. Make the remaining connections.
   a. Click and drag from the aclk connector of FFT and Hls_xfft2real blocks to the aclk external port (or aclk connector on Hls_real2xfft block or anywhere on “wire” connecting them).
   b. Connect aresetn of FFT and Hls_xfft2real blocks to aresetn network.
   c. The XFFT configuration interface is left unconnected, as this design always operates in the default mode of the core.

14. Click the **Regenerate** icon to clean up and reorganize the Block Design.
15. Validate the Block Design by clicking the **Validate Design** icon on the toolbar.

16. Click **File > Save Block Design.**

17. Close the Block Design.

18. The next step is to generate output products.

   a. In the **Sources** tab of Project Manager pane (Figure 209), right-click **RealFFT.bd** and select **Generate Output Products.**

   b. Click **OK** in the resulting dialog to initiate the generation of all output products.
19. Create an HDL Wrapper.

   a. In the Sources tab of the Project Manager pane, right-click RealFFT.bd and select Create HDL Wrapper. (This is the same procedure and menu as described in the previous step.)

   b. Click OK and let Vivado manage the wrapper.

**Step 5: Verify the Design**

The next step in creating the final design is to verify design with the HDL test bench provided in the lab exercise: realfft_rtl_tb.v.

1. Right-click Simulation Sources in Sources tab of Project Manager pane (Figure 210).

2. Select Add Sources.
3. Select **Add or Create Simulation Sources** in the Add Sources dialog.

4. Click **next**.

5. In the Add Sources dialog box, click the **Add Files** button highlighted in Figure 212.
6. Browse to the file `realfft_rtl_tb.v` in the tutorial directory `Using_IP_with_IPI\lab1\verilog_tb`.

7. Select it and click **OK**.

8. Select the checkbox **Copy sources into the project** (**Figure 212**).
Figure 212: Copy Design Sources

**Note**: When you copy the design source files into the project, edits to the file(s) are not automatically propagated to the original source file.

9. Click **Finish**.

10. Click **Run Simulation** in the **Flow Navigator** (Figure 213).
11. Once the simulation has started, click the **Run All** icon to complete simulation.

**Figure 214: Run The Simulation to Conclusion**

**Conclusion**

In this tutorial, you learned:

- How to create Vivado HLS IP using a Tcl script.
- How to import a design using IP integrator (IPI) and include both Xilinx IP and the Vivado IP blocks.
- How to verify the design in IPI.
Chapter 10  Using HLS IP in a Zynq Processor Design

Overview

A common use of High-Level Synthesis design is to create an accelerator for a CPU – to move code that executes on the CPU into the FPGA programmable logic to improve performance. This tutorial shows how you can incorporate a design created with High-Level Synthesis into a Zynq device.

This tutorial consists of two lab exercises.

Lab 1

You create and configure a simple HLS design to work with the CPU on a Zynq device. The HLS design used in this lab is simple to allow the focus of the tutorial to be on explaining the connections to the CPU and how to configure the software drivers created by High-Level Synthesis to control the device and manage interrupts.

Lab 2

This lab illustrates a common high performance connection scheme for connecting hardware accelerator blocks that consume data originating in the CPU memory and/or producing data destined for it in a streaming manner. The lab highlights the software requirements to avoid cache coherency issues.

Tutorial Design Description

You can download the tutorial design file can be downloaded from the Xilinx Website. Refer to the information in
Locating the Tutorial Design Files.

This tutorial uses the design files in the tutorial directory `Vivado_HLS_Tutorial\Using_IP_with_Zynq`.

The sample design is a simple multiple accumulate block. The focus of this tutorial exercise is the methodology, connections and integration of the software drivers. (The tutorial does not focus on the logic in the design itself.)

Lab 1: Implement Vivado HLS IP on a Zynq Device

This lab exercise integrates both the High-Level Synthesis IP and the software drivers created by HLS to control the IP in a design implemented on a Zynq device.

**IMPORTANT:** The figures and commands in this tutorial assume the tutorial data directory `Vivado_HLS_Tutorial` is unzipped and placed in the location `C:\Vivado_HLS_Tutorial`.

If the tutorial data directory is unzipped to a different location, or on Linux systems, adjust the few pathnames referenced, to the location you have chosen to place the `Vivado_HLS_Tutorial` directory.

Step 1: Create a Vivado HLS IP Block

Create two HLS blocks for the Vivado IP Catalog using the Tcl script provided. The script runs HLS C-synthesis, runs RTL co-simulation, and packages the IP for the two HLS designs (hls_real2xfft and hls_xfft2real).

1. Open the Vivado HLS Command Prompt.
   a. On Windows use **Start > All Programs > Xilinx Design Tools > Vivado 2014.1 > Vivado HLS > Vivado HLS 2014.1 Command Prompt** (Figure 215).
   b. On Linux, open a new shell.

![Figure 215: Vivado HLS Command Prompt](image)

2. Using the command prompt window, change the directory to `Vivado_HLS_Tutorial\Using_IP_with_Zynq\lab1\hls_macc` (Figure 216).
3. Type `vivado_hls -f run_hls.tcl` to create the HLS IP (Figure 216).
Figure 216: Create the HLS Design

When the script completes, there is a Vivado HLS project directory vhls_prj, which contains the HLS IP, including the Vivado IP Catalog archive for use in Vivado designs. The remainder of this tutorial exercise shows how the Vivado HLS IP blocks can be integrated into a Zynq design using IP Integrator.

Step 2: Create a Vivado Zynq Project

1. Launch the Vivado Design Suite (not Vivado HLS):
   b. On Linux, type vivado in the shell.
2. From the Welcome screen, click Create New Project (Figure 217).
3. In the New Project wizard:
   a. Click **Next**.
   b. In the Project Location text entry box, browse to the location of the tutorial file directory and click **Next** (Figure 218).
   c. On the Project Type page, select “Do not specify sources at this time” (if it is not the default).
   d. Click **Next**.
4. On the Default Part page:
   a. Click **Boards**.
   b. Select the **ZYNQ-7 ZC702 Evaluation Board** (Figure 219).

![Figure 219: Specify the Vivado Project Details](image)
c. Click **Next**.
d. Click **Finish** on the New Project Summary Page.

The project workspace opens as shown in **Figure 220**.

**Step 3: Add HLS IP to the IP Catalog**

1. In the Project Manager area of the Flow Navigator pane, click **IP Catalog**.

The IP Catalog appears in the main pane of the workspace.

2. Click the **IP Settings** icon (**Figure 222**).
3. In the IP Settings dialog, click **Add Repository**.

4. In the IP Repositories dialog box:
   a. Browse to the tutorial directory location and click the **Create New Folder** icon.
   b. Enter “vivado_ip_repo” in the resulting dialog (Figure 223).
   c. Click **OK**.
   d. Click **Select** to close the IP Repository.

![Figure 222: Open the IP Catalog Settings](image)

![Figure 223: IP Repository](image)
5. Returning to the IP Setting dialog box:
   a. Click **Add IP**.
   b. In the Select IP to Add to Repository dialog, browse to the location of the HLS IP: `Using_IP_with_Zynq/lab1/hls_macc/vhls_prj/solution1/impl/ip/`.
   c. Select the IP Catalog package `Xilinx_com_hls_hls_macc_1_00.a.zip` file (**Figure 224**).
   d. Click **OK**.

   ![Figure 224: Add IP to the Repository](image)

6. The new HLS IP should now appear in the IP Settings dialog box.
7. Click **OK** to exit the dialog box.

8. There is now a Vivado HLS IP category in the IP Catalog and, if expanded, the Hls_macc IP displays (Figure 226).

   ![HLS IP in the Repository](image)

   **Figure 225: HLS IP in the Repository**

   ![HLS IP in the IP Catalog](image)

   **Figure 226: HLS IP in the IP Catalog**

**Step 4: Creating an IP Integrator Block Design of the System**

1. In the IP Integrator area of the Flow Navigator, click **Create Block Design** and enter "Zynq_Design" in the dialog box.
The Block Design view opens in the main pane, with a new Diagram tab, containing a blank Block Design canvas.

2. Click the **Add IP link** under the title bar, which pops up an IP search dialog.
   a. Type in “proce” into the Search text entry box.
   b. Select the **ZYNQ7 Processing System** item and press **Enter**.
An IP symbol for the ZYNQ7 Processing System appears on the canvas.

3. Double-click the ZYNQ IP symbol to open its Re-customize IP dialog.
   a. Click the Presets icon and select ZC702 (Figure 229).
4. Click **MIO Configuration** in the Page Navigator pane.
   a. Expand the **Application Processor Unit** tree view.
   b. Unselect **Timer 0** (or any other timer if they are selected).

5. Click **Interrupts** in the Page Navigator pane.
   c. Select **Fabric Interrupts** and expand its tree view.
d. Select **IRQ_F2P[15:0]** and click **OK** to close the Re-customize IP dialog box.

![Re-customize IP](image)

### ZYNQ7 Processing System (5.01)

<table>
<thead>
<tr>
<th>Interrupt Port</th>
<th>ID</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>IRQ_F2P[15:0]</td>
<td>91:04</td>
<td>Enables 16-bit shared interrupt port</td>
</tr>
<tr>
<td>Core0_nIRQ0</td>
<td>28</td>
<td>Enables Fast private interrupt signal</td>
</tr>
<tr>
<td>Core0_nIRQ1</td>
<td>31</td>
<td>Enables private interrupt signal for Core0</td>
</tr>
<tr>
<td>Core1_nIRQ0</td>
<td>28</td>
<td>Enables Fast private interrupt signal</td>
</tr>
<tr>
<td>Core1_nIRQ1</td>
<td>31</td>
<td>Enables private interrupt signal for Core1</td>
</tr>
<tr>
<td>PS-PL Interrupt Ports</td>
<td></td>
<td></td>
</tr>
<tr>
<td>IRQ_P2P_DMA0</td>
<td>Enables shared interrupt signal 0</td>
<td></td>
</tr>
<tr>
<td>IRQ_P2P_DMA1</td>
<td>Enables shared interrupt signal 1</td>
<td></td>
</tr>
<tr>
<td>IRQ_P2P_DMA2</td>
<td>Enables shared interrupt signal 2</td>
<td></td>
</tr>
</tbody>
</table>

**Figure 231: Zynq Processor Interrupt Configuration**

IPI provides Designer Assistance to automate certain tasks, such as making the correct external connections to DDR memory and Fixed I/O for the ZYNQ PS7.

6. Click the **Run Block Automation** link under the title bar (**Figure 232**).
   a. Select **/processing_system7_1**.
   b. Ensure **Apply Board Presets** is **Unselected**. If this remains selected it will re-apply the timers which were disable in step 4 and result in additional ports on the Zynq block in **Figure 232**
   c. Click **OK** to complete in the resulting dialog box.
7. Add HLS IP to the design by right-clicking in an open space of canvas and by selecting **Add IP** from the context menu.
   a. Type “hls” in the Search text entry box and press **Enter** to add it to design (**Figure 233**).
Designer assistance is also available to automate the interconnection of IP blocks.

8. Click the **Run Connection Automation** link at the top of the canvas.

9. Select `/hls_macc_1/S_AXI_HLS_MACC_PERIPH_BUS` and click **OK** in the resulting dialog box to automatically connect the HLS IP to the M_AXI_GP0 interface of the PS7.

This adds an AXI Interconnect (instance: `processing_system7_1_axi_periph`), a Proc Sys Reset block (instance: `proc_sys_reset`) and makes all necessary AXI related connections to create the design shown in Figure 234.
The only remaining connection necessary is from the HLS interrupt port to the PS7 IRQ_F2P port.

10. Bring the cursor over the interrupt pin on the hls_macc_1 IP symbol.
    a. When the cursor changes to pencil shape, click and drag to the IRQ_F2P[0:0] port of the PS7 and release, completing the connection.

11. Bring the **Address Editor** tab forward and confirm that the hls_macc_1 peripheral has been assigned a master address range. If it has not, click the **Auto Assign Address** icon.

![Address Editor](Figure 235: Address Editor)

The final step in the Block Diagram design entry process is to validate the design.

12. Click the **Validate Design** icon in the toolbar.

13. Upon successful validation, save (control-s) the Block Design.

**Step 5: Implementing the System**

Before proceeding with the system design, you must generate implementation sources and create an HDL wrapper as the top-level module for synthesis and implementation.

1. Return to the Project Manager view by clicking on **Project Manager** in the Flow Navigator.

2. In the Sources browser in the main workspace pane, a Block Diagram object named Zynq_Design is at the top of the Design Sources tree view (**Figure 236**). Right-click this object and select **Generate Output Products**.

3. In the resulting dialog box, click **Generate** to start the process of generating the necessary source files.
4. Right-click the Zynq_Design object again, select Create HDL Wrapper, and click OK to exit the resulting dialog box.

The top-level of the Design Sources tree becomes the Zynq_Design_wrapper.v file. The design is now ready to be synthesized, implemented, and to have an FPGA programming bitstream generated.

5. Click Generate Bitstream to initiate the remainder of the flow.

6. In the dialog that appears after bitstream generation has completed, select Open Implemented Design and click OK.

**Step 6: Developing Software and Running it on the ZYNQ System**

You are now ready to export the design to Xilinx SDK. In SDK, you create software that runs on a ZC702 board (if available). A driver for the HLS block was generated during HLS export of the Vivado IP Catalog package. This driver must be made available in SDK so that the PS7 software can communicate with the block.

1. From the Vivado File menu select Export > Export Hardware for SDK.

   **Note:** Both the IPI Block Design and the Implemented Design must be open in the Vivado workspace for this step to complete successfully.

2. In the Export Hardware for SDK dialog box (Figure 237), ensure that the Include Bitstream and Launch SDK options are enabled and click OK.
3. SDK opens. If the Welcome page is open, close it.

4. From the SDK File menu, select **New > Application Project**.
   a) In the New Project dialog enter a project name: Zynq_Design_Test
   b) Click **Next**.
   c) Select the **Hello World** template.
   d) Click **Finish**.
5. Power up the ZC702 board and test the Hello World application:
   b. Ensure the board has all the connections to allow you to download the bit stream on the FPGA device. Refer to the documentation that accompanies the ZC702 development board.

7. Click XilinxTools > Program FPGA (or toolbar icon).

Notice that the Done LED (DS3) is now on.

8. Setup a Terminal in the tab at bottom of workspace:
   a) Click the Connect icon (Figure 239).
b) Select **Connection Type > Serial**.

c) Select the COM port to which the USB UART cable is connected (generally *not* COM1 or COM3) On Windows, if you are not sure, open the Device Manager and identify the port with the Silicon Labs driver under Ports (COM & LPT).

d) Change the Baud Rate to 115200 (**Figure 240**).

e) Click **OK** to exit the **Terminal Settings** dialog box.
10. Switch to the **Terminal** tab and confirm that “Hello World” was received.

![Figure 241: Run the Application Project](image)

![Figure 242: Console Output](image)
Step 7: Modify software to communicate with HLS block

The completely modified source file is available in the arm_code directory of the tutorial file set. The modifications are discussed in detail below.

1. Open the helloworld.c source file.

2. Several BSP (and standard C) header files need to be included:

   ```
   #include <stdlib.h>  // Standard C functions, e.g. exit()
   #include <stdbool.h> // Provides a Boolean data type for ANSI/ISO-C
   #include "xparameters.h" // Parameter definitions for processor peripherals
   #include "xscugic.h"     // Processor interrupt controller device driver
   #include "XHls_macc.h"   // Device driver for HLS HW block
   ```

3. Define variables for the HLS block and interrupt controller instance data. The variables will be passed to driver API calls as handles in the respective hardware.

   ```
   // HLS macc HW instance
   XHls_macc HlsMacc;
   // Interrupt Controller Instance
   XScuGic ScuGic;
   ```

4. Define global variables to interface with the interrupt service routine (ISR).

   ```
   volatile static int RunHlsMacc = 0;
   volatile static int ResultAvailHlsMacc = 0;
   ```

5. Define a function to wrap all run-once API initialization function calls for the HLS block.

   ```
   int hls_macc_init(XHls_macc *hls_maccPtr)
   {
       XHls_macc_Config *cfgPtr;
       int status;

       cfgPtr = XHls_macc_LookupConfig(XPAR_XHLS_MACC_0_DEVICE_ID);
       if (!cfgPtr) {
           print("ERROR: Lookup of accelerator configuration failed.\n\r");
           return XST_FAILURE;
       }

       status = XHls_macc_CfgInitialize(hls_maccPtr, cfgPtr);
       if (status != XST_SUCCESS) {
           print("ERROR: Could not initialize accelerator.\n\r");
           return XST_FAILURE;
       }

       return status;
   }
   ```

6. Define a helper function to wrap the HLS block API calls required to enable its interrupt and start the block.

   ```
   void hls_macc_start(void *InstancePtr)
   {
       XHls_macc *pAccelerator = (XHls_macc *)InstancePtr;
       XHls_macc_InterruptEnable(pAccelerator,1);
   }
   ```
An interrupt service routine is required in order for the processor to respond to an interrupt generated by a peripheral.

Each peripheral with an interrupt attached to the PS must have an ISR defined and registered with the PS’s interrupt handler.

The ISR is responsible for clearing the peripheral’s interrupt and, in this example, setting a flag that indicates that a result is available for retrieval from the peripheral. In general, ISRs should be designed to be lightweight and as fast as possible, essentially doing the minimum necessary to service the interrupt. Tasks such as retrieving the data should be left to the main application code.

```c
void hls_macc_isr(void *InstancePtr){
    XHls_macc *pAccelerator = (XHls_macc *)InstancePtr;

    //Disable the global interrupt
    XHls_macc_InterruptGlobalDisable(pAccelerator);
    //Disable the local interrupt
    XHls_macc_InterruptDisable(pAccelerator, 0xffffffff);

    // clear the local interrupt
    XHls_macc_InterruptClear(pAccelerator, 1);

    ResultAvailHlsMacc = 1;
    // restart the core if it should run again
    if(RunHlsMacc){
        hls_macc_start(pAccelerator);
    }
}
```

7. Define a routine to setup the PS interrupt handler and register the HLS peripheral’s ISR.

```c
int setup_interrupt()
{
    //This functions sets up the interrupt on the ARM
    int result;
    XScuGic_Config *pCfg =
    XScuGic_LookupConfig(XPAR_SCUGIC_SINGLE_DEVICE_ID);
    if (pCfg == NULL){
        print("Interrupt Configuration Lookup Failed\n\r");
        return XST_FAILURE;
    }

    result = XScuGic_CfgInitialize(&ScuGic, pCfg, pCfg->CpuBaseAddress);
    if(result != XST_SUCCESS){
        return result;
    }

    // self-test
    result = XScuGic_SelfTest(&ScuGic);
    if(result != XST_SUCCESS){
        return result;
    }

    // Initialize the exception handler
    Xil_ExceptionInit();
}
```
8. Define a software model of the HLS hardware functionality with which you can compare reference results.

```c
void sw_macc(int a, int b, int *accum, bool accum_clr)
{
    static int accum_reg = 0;
    if (accum_clr)
        accum_reg = 0;
    accum_reg += a * b;
    *accum = accum_reg;
}
```

9. Modify main() to use the HLS device driver API and the functions defined above to test the HLS peripheral hardware.

```c
int main()
{
    print("Program to test communication with HLS MACC peripheral in PL\n\r");
    int a = 2, b = 21;
    int res_hw;
    int res_sw;
    int i;
    int status;

    //Setup the matrix mult
    status = hls_macc_init(&HlsMacc);
    if(status != XST_SUCCESS){
        print("HLS peripheral setup failed\n\r");
        exit(-1);
    }

    //Setup the interrupt
    status = setup_interrupt();
    if(status != XST_SUCCESS){
        print("Interrupt setup failed\n\r");
        exit(-1);
    }
```
//set the input parameters of the HLS block
XHls_macc_SetA(&HlsMacc, a);
XHls_macc_SetB(&HlsMacc, b);
XHls_macc_SetAccum_clr(&HlsMacc, 1);

if (XHls_macc_IsReady(&HlsMacc))
    print("HLS peripheral is ready. Starting... ");
else {
    print("!!! HLS peripheral is not ready! Exiting...\n\r");
    exit(-1);
}

if (0) { // use interrupt
    hls_macc_start(&HlsMacc);
    while(!ResultAvailHlsMacc)
        // spin
    res_hw = XHls_macc_GetAccum(&HlsMacc);
    print("Interrupt received from HLS HW.\n\r");
} else { // Simple non-interrupt driven test
    XHls_macc_Start(&HlsMacc);
    do {
        res_hw = XHls_macc_GetAccum(&HlsMacc);
    } while (!XHls_macc_IsReady(&HlsMacc));
    print("Detected HLS peripheral complete. Result received.\n\r");
}

//call the software version of the function
sw_macc(a, b, &res_sw, false);

printf("Result from HW: %d; Result from SW: %d\n\r", res_hw, res_sw);
if (res_hw == res_sw) {
    print("*** Results match ***\n\r");
    status = 0;
} else {
    print("!!! MISMATCH !!!\n\r");
    status = -1;
}

cleanup_platform();
return status;

10. Save (control-s) the modified source file, and SDK automatically attempts to re-build the
application executable. If the build fails, fix any outstanding issues.

Run the new application on the hardware and verify that it works as expected. Ensure that a TCF
hardware server is running, that the FPGA is programmed and a terminal session is connected to
the UART. Then Launch on Hardware, as you did for the previous Hello World application code.

Upon success, the Terminal session looks similar to Figure 243.
Lab 2: Streaming data between the Zynq CPU and HLS Accelerator Blocks

This lab illustrates a common high-performance connection scheme for connecting hardware accelerator blocks that consume data originating in the CPU memory and/or producing data destined for it, in a streaming manner.

- This tutorial uses the same Vivado HLS and XFFT IP blocks created in Lab 1 of the tutorial “Using HLS IP in IP Integrator”. In this lab exercise these blocks are connected to the HP0 Slave AXI4 port on a Zynq7 processing system via an AXI DMA IP core.
- The hardware accelerator blocks are free-running and do not require drivers; as long as data is pushed in and pulled out by the CPU (often simply referred to as the Processing System or PS).
- The lab highlights the software requirements to avoid cache coherency issues.

**Step 1: Generate the HLS IP**

1. From the Vivado HLS command prompt used in Lab 1, change to the lab2 directory as shown in **Figure 244**.
2. Run Vivado HLS to create two HLS IP blocks by typing `vivado_hls -f run_hls.tcl`.

![Figure 244: Setup for Zynq Lab 2](image)
When the script completes, there are two Vivado HLS project directories, fe_vhls_prj and be_vhls_prj, which contain the HLS IP, including the Vivado IP Catalog archives for use in Vivado designs.

- The “front-end” IP archive is located at fe_vhls_prj/IPXACTExport/impl/ip/
- The “back-end” IP archive is located at be_vhls_prj/IPXACTExport/impl/ip/

**Step 2: Create a Vivado Design Suite Project**

1. Launch the Vivado Design Suite (**not** Vivado HLS):
   a. On Windows use **Start > All Programs > Xilinx Design Tools > Vivado 2014.1 > Vivado 2014.1**
   b. On Linux, type vivado in the shell.
2. From the Welcome screen, select **Create New Project**.
3. Click **Next** on the first page of the Create a New Vivado Project wizard.
4. Click the ellipsis button to the right of the Project location text entry box and browse to the lab2 tutorial directory.
5. Click **Next** to move to the Project Type page of the wizard.
   a. Select **RTL Project** and click **Next**.
   b. Do not specify sources at this time (if not the default); just click **Next**.
   c. Do not add any Existing IP; just click **Next**.
   d. Do not add any constraints; just click **Next**.
6. On the Default Part page click **Boards** under Specify and select the **ZYNQ-7 ZC702 Evaluation Board**. Click **Next**.
7. On the New Project Summary Page, click **Finish** to complete the new project setup.

**Step 3: Add HLS IP to an IP Repository**

1. In the Project Manager area of the Flow Navigator pane, click **IP Catalog**.
2. The IP Catalog appears in the main pane of the workspace.
   a. Click the **IP Settings** icon.
3. In the IP Settings dialog box, click **Add Repository**.
4. In the IP Repositories dialog box:
   a. Browse to the Lab 2 tutorial directory lab2.
   b. Click the **Create New Folder** icon.
   c. Enter “vivado_ip_repo” in the resulting dialog.
   d. Click **OK**.
Using HLS IP in a Zynq Processor Design

5. On returning to the IP Setting dialog box:
   a. Click **Add IP**.
   b. In the Select IP to Add to Repository dialog box, browse to the location of the HLS IP:
      lab2/hls_designs/fe_vhls_prj/IPXACTExport/impl/ip/ or, if using IP created in previous tutorial, browse to the corresponding path.
   c. Select the xilinx_com_hls_hls_real2xfft_1_00_a.zip file.
   d. Click **OK**.
5. Follow the same procedure to add the 2nd HLS IP package, in directory
   lab2/hls_designs/be_vhls_prj/IPXACTExport/impl/ip/, to the repository:
   xilinx_com_hls_hls_xfft2real_1_00_a.zip.
6. The new HLS IP now appears in the IP Setting dialog box.
7. Click **OK** to exit the dialog box.
8. There is now a Vivado HLS IP category in the IP Catalog and, if expanded, the HLS IP displays.

**Step 4: Create a Top-level Block Design**

1. Click **Create Block Diagram** under IP Integrator in the Flow Navigator.
   a. In the resulting dialog box, name the design Zynq_RealFFT.
   b. Click **OK**.
2. In the Diagram tab, click the **Add IP** link in the “get started” message.
   a. In the Search box, type “fourier”.
   b. Press **Enter**.
3. Double-click the new **Fast Fourier Transform IP** symbol to open the Re-customize IP dialog box. On the Configuration tab:
   a. Change the Transform Length to 512.
   b. Change the Target Clock Frequency to 100 MHz.
   c. In the **Architecture Choice** section, select **Pipelined, Streaming I/O**
4. Select the **Implementation** tab:
   a. Select ARESETN (active low) in the Control Signals group
   b. Verify that Bit/Digit Reversed Order is selected under Output Ordering Options
   c. Verify that Non Real Time is selected as Throttle Scheme.
   d. Click **OK** to exit Re-customize IP dialog
5. Add one instance of each of the HLS generated blocks to the design.
a. Right-click in any space in the canvas and select **Add IP**.

b. Type “hls” into the Search text entry box.

c. Highlight both IPs (Click the control key and select both)

d. Press **Enter**.

Because the output AXI4-Stream interface of the hls_xfft2real block does not include a TKEEP signal, it cannot be directly connected to the AXI DMA (which will be added later). For that reason, you add a Xilinx AXI4-Stream Subset converter: this block configures automatically.

6. Right-click in any space in the canvas and select **Add IP**.

a. Type “subset” into the Search text entry box.

b. Click **Enter**.

7. Connect the HLS blocks to the FFT block.

a. Hover the cursor over the “m_axis_dout” interface connector of the Hls_real2xfft block until a pencil cursor appears.

b. Left-click and hold down the mouse button to start a connection.

c. Drag the connection line to the “S_AXIS_DATA” input port connector of the FFT block and release when a green check mark appears next to it.

8. In a similar fashion:

a. Connect the FFT’s “M_AXIS_DATA” interface to the “s_axis_din” input interface of the Hls_xfft2real block.

b. Connect the m_axis_dout pin of the hls_xfft2real_1 component to the S_AXIS pint of the axis_subset_converter_1 component

9. Now put the data processing blocks into their own level of hierarchy.

a. Select everything in the current digram by entering **Ctrl+A**.

b. Right-click the canvas and select **Create Hierarchy** from the context menu.
c. In the Create Hierarchy dialog box, enter RealFFT as the Cell name.
d. Ensure that the **Move ‘4’ selected blocks to new hierarchy** option is checked, as shown in Figure 246.

e. Click **OK**.
The diagram will look as Figure 247.

![Figure 247: New Hierarchy Block](image)

Add pins to the RealFFT hierarchical block so that you can connect it at the top-level

10. Double-click the **RealFFT** block to open its diagram.

![Figure 248: RealFFT Diagram](image)

11. Right-click the s_axis_din pin of the hls_real2xfft_1 block and select **Create Interface Pin** from the context menu.
12. In the Create Interface Pin dialog box, change the Interface name to realfft_s_axis_din.
   a. Accept all other defaults and click **OK**.

13. Right-click the **aclk** pin of the hls_real2xfft_1 block and select **Create Pin** from the context menu.
   a. Click **OK** to accept all defaults in the Create Pin dialog.
Using HLS IP in a Zynq Processor Design

Figure 251: Create a Clock Pin

Once you create this clock pin, the RealFFT diagram appears.

Figure 252: RealFFT Diagram with Interface Pin and clock pin

14. Following the procedures in steps 11 to 13:

   a. Create an interface pin called ‘realfft_m_axis_dout’ connected to the M_AXIS pin of the axis_subset_converter_1 component.

   b. Create a pin for aresetn (from any one of the blocks).
After this step, the RealFFT diagram appears.

![RealFFT Diagram with all pins](image)

**Figure 253: RealFFT Diagram with all pins**

Finalize RealFFT block internal connections. The ap_start pins for the HLS blocks are tied HIGH, and the aclk and arestn pins on all blocks are tied together.

15. Right-click the canvas and select **Add IP** from the context menu.
   a. Type 'const' into the search box and press **Enter**.
   b. Double-click the **xlconstant_1** component and verify that the Const Val field in the Customize IP dialog is set to ‘1’.

![Create A Constant 1 Tie-Off](image)

**Figure 254: Create A Constant 1 Tie-Off**

16. Following techniques covered in Lab1 of this tutorial:
   a. Connect the output pin of xlconstant_1 to the ap_start pin of hls_real2xfft_1.
b. Connect the output pin of xlconstant_1 to the ap_start pin of hls_xfft2real_1.

17. Similarly, connect all remaining component aclk and aresetn pins to the RealFFT block diagram aclk and aresetn pins respectively.

Leave the S_AXIS_CONFIG input interface of xfft_1 unconnected. For this tutorial, the default operating modes suffice. Also, leave all other output pins of the components unconnected. The final RealFFT diagram appears with the connections shown in Figure 255.

![Figure 255: Final RealFFT Diagram](image)

18. Close the RealFFT diagram tab and return to the top-level Zynq_RealFFT diagram.

19. Create the Zynq system.
   a. Right-click the canvas of the top-level diagram and select **Add IP** from the context menu.
   b. Type ‘proce’ in the search box, select **ZYNQ7 Processing System** and press **Enter**.
   c. Double-click the **processing_system7_1** component to enter the Re-customize IP wizard for the ZYNQ7.
   d. Click the **Presets** button near the top of the wizard screen, select the **ZC702 Development Board Template**, and click **OK**.
   e. Click **PS-PL Configuration** in the Page Navigator pane on the left of the wizard.
   f. Expand the HP Slave AXI Interface category and check the box for the S AXI HP0 interface, leaving the S AXI HP0 DATA WIDTH at 64.
20. Note the Designer Assistance Available notification at the top of the screen.
   a. Run Block Automation on /processing_system7_1.
   b. Click **OK** in the resulting dialog box.

21. Add AXI DMA IP to allow the PS to stream data to/from the RealFFT block via its HP0 Slave AXI interface
   a. Right-click the canvas and select **Add IP** from the context menu.
b. Type ‘direct’ into the search box and select AXI Direct Memory Access from the menu and press Enter.

22. Double-click the axi_dma_1 component to open its Re-customize IP dialog and make the following changes (Figure 259):
   a. Disable the Scatter Gather Engine (deselect the option).
   b. Set the Memory Map Data Width to 64 for both Read and Write channels.
   c. Set the Stream Data Width to 16 for the Read channel (MM2S).
   d. Leave the Stream Data Width at 32 for the Write channel (S2MM).
   e. Set the Max Burst Size to 128 for both channels.
   f. Enable Allow Unaligned Transfers for both channels.

![Figure 258: Configuring the AXI Direct Memory Access](image)

23. Note that Designer Assistance is again available. Run Connection Automation on /axi_dma_1/S_AXI_LITE and click OK in the resulting dialog box.

After running Design Assistance, the diagram appears similar to the one shown in Figure 260.
24. Run Connection Automation on /processing_system7_1/S_AXI_HP0 and click OK to accept the default connection in the dialog box.

**Note:** the Connection Automation only connects one of the AXI DMA components M_AXI_* ports through the axi_mem_intercon component.

25. Double-click the axi_mem_intercon component to re-customize it.
   a. Change the Number of Slave Interfaces from 1 to 2 (Figure 260).
   b. Click OK.
26. Make a connection between the M_AXI_S2MM port on axi_dma_1 component and S01_AXI port on the axi_mem_intercon component.

27. Connect the clocks and reset ports.
   a. Connect the axi_mem_intercon S01_ACLK and S01_ARESETN ports to the appropriate nets already present in the diagram (processing_system7_1_fclk_clk0 and proc_sys_reset_peripheral_aresetn, respectively).
   b. Connect the m_axi_s2mm_aclk port of the axi_dma_1 component to the clock network.

28. Connect the RealFFT block to rest of the system.
   a. Make a connection between the realfft_s_axis_din input of the RealFFT block and the M_AXIS_MM2S output of the axi_dma_1 component.
   b. Make a connection between the realfft_m_axis_dout output of the RealFFT block and the S_AXIS_S2MM input of the axi_dma_1 component.
   c. Connect the aclk and aresten pin of the RealFFT block to the existing networks.

29. Finalize the IPI block diagram design.
   a. Select the Address Editor tab and click the **Auto Assign Address** icon.
30. To view the completed design, run Validate Design by clicking the icon in the toolbar (Figure 263).

Step 5: Implementing the System

Before proceeding with the system design, you must generate implementation sources and create an HDL wrapper as the top-level module for synthesis and implementation.

1. Return to the Project Manager view by clicking **Project Manager** in the Flow Navigator.

2. In the Sources browser in the main workspace pane, a Block Diagram object named Zynq_RealFFT appears at the top of the Design Sources tree view. Right-click this object and select **Generate Output Products**.

3. In the resulting dialog box, click **OK** to start the process of generating the necessary source files.
4. Right-click the Zynq_RealFFT object again, select Create HDL Wrapper, and click OK to exit the resulting dialog box.

The top-level of the Design Sources tree becomes the Zynq_RealFFT_wrapper.v file. You are now ready to synthesize, implement, and generate an FPGA programming bitstream for the design.

5. Click Generate Bitstream to initiate the remainder of the flow.

6. In the dialog that appears after bitstream generation has completed, select Open Implemented Design and click OK.

**Step 6: Setup SDK and test the ZYNQ System**

You are now ready to export the design to Xilinx SDK. In SDK, you create software to be run on a ZC702 board (if available). A driver for the HLS block was generated during HLS export of the Vivado IP Catalog package and must be made available in SDK for the PS7 software to communicate with the block.

1. From the Vivado File menu select Export > Export Hardware for SDK.

   **Note:** Both the IPI Block Design and the Implemented Design must be open in the Vivado workspace for this step to complete successfully.

2. In the Export Hardware for SDK dialog box, ensure that the Include Bitstream and Launch SDK options are checked, and click OK.

3. SDK opens. If the Welcome page is open, close it.

4. Create a Hello World application (also creates BSP).
   a. Select File > New > Application Project.
   b. Enter the project name Zynq_RealFFT_Test.
   c. Click Next.
   d. Select Hello World (if it is not the default).
   e. Click Finish.

5. Power up the ZC702 board and program the FPGA.

   Ensure the board has all the connections to allow you to download the bit stream on the FPGA device. Refer to the documentation that accompanies the ZC702 development board.
   a. Select XilinxTools > Program FPGA. The Done LED (DS3) goes on.

6. Set up a Terminal in the tab at bottom of workspace:
   a. Click the Connect icon.
   b. Select Connection Type > Serial.
   c. Select the COM port to which the USB UART cable is connected (generally not COM1 or COM3). On Windows, if you are not sure, open the Device Manager and identify the port with the Silicon Labs driver under Ports (COM & LPT).
d. Change the Baud Rate to 115200.

e. Click **OK** to exit Terminal Settings dialog box.

f. Check that terminal is connected by message in tab title bar.

7. Right-click application project **Zynq_Design_Test** in the Explorer pane
   a. Select **Run As > Launch on Hardware**.

8. Switch to the Terminal tab and confirm that “Hello World” was received.

9. This project uses the C math library (libm), so you must adjust the build settings to link to it.
   a. Right-click the **zynq_realfft_test project** in the Project Explorer pane and select **C/C++ Build Settings** (Figure 264).
      b. Add the ARM gcc linker libraries.
         i. In the Tool Settings tab, select ‘ARM gcc linker’ > Libraries.
          
         ii. Click the **Add** icon.
Step 7: Modify software to communicate with HLS block

The completely modified source file is available in the arm_code directory of the tutorial file set. The modifications are discussed in detail below.

1. Open the helloworld.c source file.
2. Several BSP (and standard C) header files must be included:
   
   ```
   #include <stdlib.h> // Std C functions, e.g. exit()
   #include <math.h>   // libm header: sqrt(), cos(), etc
   ```

   c. Enter ‘m’ in the text box in the Enter Value dialog box and click **OK**.

   ![Figure 264: C/C++ Build Settings](image)

   ![Figure 265: Library Setting](image)

   d. Click **OK** to exit the Properties for zynq_realfft_test dialog box.

   ![Figure 264: C/C++ Build Settings](image)

   ![Figure 265: Library Setting](image)
3. Define the (real data) transform length of the FFT:

#define REAL_FFT_LEN 1024

4. Define a custom complex data type with 16-bit real and imaginary members:

typedef struct {
    short re;
    short im;
} complex16;

5. Declare helper functions before the definition of main(); they will be defined later.

Note: The init_dma() function wraps up all run-once, initialization AXI DMA driver API calls and checks that hardware initialization is successful before returning or exiting on an error condition. The generate_waveform() function fills an array with a simple, periodic waveform to be used as input stimulus for the RealFFT accelerator.

int init_dma(XAxiDma *axiDma);
void generate_waveform(short *signal_buf, int num_samples);

6. Modify main() to generate and send input data to the RealFFT accelerator and receive the spectral data from it via the AXI DMA engine. Sections of particular importance will be discussed in detail.

// Program entry point
int main()
{
a. Declare an XAxiDma instance that will be used as a handle to the AXI DMA hardware:

   // Declare a XAxiDma object instance
   XAxiDma axiDma;

b. Declare variable for local data storage:

   // Local variables
   int i, j;
   int status;
   static short realdata[4*REAL_FFT_LEN];
   volatile static complex16 realspectrum[REAL_FFT_LEN/2];

c. Run platform and DMA initialization functions:

   // Initialize the platform
init_platform();
print("---------------------------------------\n\r");
print("- RealFFT PL accelerator test program -\n\r");
print("---------------------------------------\n\r");

// Initialize the (simple) DMA engine
status = init_dma(&axiDma);
if (status != XST_SUCCESS) {
    exit(-1);
}

d. Generate a stimulus waveform:

    // Generate a waveform to be input to FFT
    for (i = 0; i < 4; i++)
        generate_waveform(realdata + i * REAL_FFT_LEN, REAL_FFT_LEN);

e. Before making the DMA transfer request, the buffer containing the data must be flushed
   from the processor's data cache. Without this step, the DMA might pull stale data from
   the DRAM.

    // *IMPORTANT* - flush contents of 'realdata' from data cache to memory
    // before DMA. Otherwise DMA is likely to get stale or uninitialized data
    Xil_DCacheFlushRange((unsigned)realdata, 4 * REAL_FFT_LEN * sizeof(short));

f. Request DMA transfer from PS to PL. Enough data to fill the front-end block and the FFT
   processing pipelines must be sent in order for spectral data to be ready when the PL to
   PS transfer is requested. Therefore, four data sets are sent before the first output set is
   requested:

    // DMA enough data to push out first result data set completely
    status = XAxiDma_SimpleTransfer(&axiDma, (u32)realdata,
        4 * REAL_FFT_LEN * sizeof(short), XAXIDMA_DMA_TO_DEVICE);

    // Do multiple DMA xfers from the RealFFT core's output stream and
    // display data for bins with significant energy. After the first frame,
    // there should only be energy in bins around the frequencies specified
    // in the generate_waveform() function - currently bins 191~193 only
    for (i = 0; i < 8; i++) {

    g. Request DMA transfer of a frame of FFT spectral data from PL to PS then poll for
       completion of the transfer before proceeding.

        // Setup DMA from PL to PS memory using
        // AXI DMA's 'simple' transfer mode
        status = XAxiDma_SimpleTransfer(&axiDma, (u32)realspectrum,
REAL_FFT_LEN / 2 * sizeof(complex16), XAXIDMA_DEVICE_TO_DMA);

// Poll the AXI DMA core
do {
    status = XAxiDma_Busy(&axiDma, XAXIDMA_DEVICE_TO_DMA);
} while(status);

h. Before attempting to use the spectral data, the processor’s data cache copy of the buffer must be invalidated to avoid use of stale data.

    // Data cache must be invalidated for 'realspectrum' buffer after DMA
    Xil_DCacheInvalidateRange((unsigned)realspectrum,
        REAL_FFT_LEN / 2 * sizeof(complex16));

i. Push another set of stimulus data to the PL in order to start the accelerator processing the next frame:

    // DMA another frame of data to PL
    if (!XAxiDma_Busy(&axiDma, XAXIDMA_DMA_TO_DEVICE))
        status = XAxiDma_SimpleTransfer(&axiDma, (u32)realdata,
            REAL_FFT_LEN * sizeof(short), XAXIDMA_DMA_TO_DEVICE);
        printf("Frame #%d received:\n", status);

j. Do something to verify that the accelerator is functioning. In this case, the spectral data is scanned for bins that contain significant energy. The expectation is to detect only energy in bins around the single tone (192) generated by the generate_waveform() function.

    // Detect energy in spectral data above a set threshold
    for (j = 0; j < REAL_FFT_LEN / 2; j++) {
        // Convert the fixed point (s.15) values into floating point values
        float real = (float)realspectrum[j].re / 32767.0f;
        float imag = (float)realspectrum[j].im / 32767.0f;
        float mag = sqrtf(real * real + imag * imag);
        if (mag > 0.00390625f) {
            printf("Energy detected in bin %3d - ",j);
            printf("{%8.5f, %8.5f}; mag = %8.5f\n", real, imag, mag);
        }
    }
    printf("End of frame.\n");
}

printf("***************\n");
printf("* End of test *\n");
printf("***************\n");
7. Define the helper function that generates the waveform data sets. This version simply fills a
buffer with a single tone with 192 cycles per num_samples data window with values in a S.15
fixed point format.

```c
void generate_waveform(short *signal_buf, int num_samples)
{
    const float cycles_per_win = 192.0f;
    const float phase = 0.0f;
    const float ampl = 0.9f;
    int i;
    for (i = 0; i < num_samples; i++) {
        float sample = ampl * 
            cosf((i * 2 * M_PI * cycles_per_win / (float)num_samples) + phase);
        signal_buf[i] = (short)(32767.0f * sample);
    }
}
```

8. Define a routine to set up the and initialize the AXI DMA engine, wrapping all driver API calls
that only need to be run once at startup.

```c
int init_dma(XAxDma *axiDmaPtr)
{
    XAxDma_Config *CfgPtr;
    int status;
    // Get pointer to DMA configuration
    CfgPtr = XAxDma_LookupConfig(XPAR_AXIDMA_0_DEVICE_ID);
    if(!CfgPtr){
        print("Error looking for AXI DMA config\n\r");
        return XST_FAILURE;
    }
    // Initialize the DMA handle
    status = XAxDma_CfgInitialize(axiDmaPtr,CfgPtr);
    if(status != XST_SUCCESS){
        print("Error initializing DMA\n\r");
        return XST_FAILURE;
    }
    //check for scatter gather mode - this example must have simple mode only
    if(XAxDma_HasSg(axiDmaPtr)){
        print("Error DMA configured in SG mode\n\r");
    }
}
```
return XST_FAILURE;
}
//disable the interrupts
XAxidma_IntrDisable(axiDmaPtr, XAXIDMA_IRQ_ALL_MASK,XAXIDMA_DEVICE_TO_DMA);
XAxidma_IntrDisable(axiDmaPtr, XAXIDMA_IRQ_ALL_MASK,XAXIDMA_DMA_TO_DEVICE);

return XST_SUCCESS;
}

9. Save the modified source file. As soon as you save the file, SDK automatically attempts to re-build the application executable. If the build fails, fix any outstanding issues.

10. Run the new application on the hardware and verify that it works as expected. Ensure that the FPGA is programmed and a terminal session is connected to the UART. Then Launch on Hardware, as done for the previous Hello World application code.
Chapter 11  Using HLS IP in System Generator for DSP

Overview
The RTL created by High-Level Synthesis can be packaged as IP and used inside System Generator for DSP (Vivado). This tutorial shows how this process is performed and demonstrates how the design can be used inside System Generator for DSP.

This tutorial consists of a single lab exercise.

Lab1 Description
Generate a design using Vivado HLS and package the design for use with System Generator for DSP. Then include the HLS IP into a System Generator for DSP design and execute an RTL simulation.

Tutorial Design Description
You can download the tutorial design file from the Xilinx Website. Refer to the information in
Obtaining the Tutorial Designs.

This tutorial uses the design files in the tutorial directory
Vivado_HLS_Tutorial\Using_IP_with_SysGen.

The sample design is a FIR filter that uses streaming interfaces modeled with the High-Level Synthesis hls::stream class. The design is fully pipelined at the function level. The optimization directives are embedded into the C code as pragmas.

---

Lab 1: Package HLS IP for System Generator

This lab exercise integrates the High-Level Synthesis IP into System Generator for DSP.

**IMPORTANT:** The figures and commands in this tutorial assume the tutorial data directory **Vivado_HLS_Tutorial** is unzipped and placed in the location **C:\Vivado_HLS_Tutorial**.

*If the tutorial data directory is unzipped to a different location, or on Linux systems, adjust the few pathnames referenced, to the location you have chosen to place the **Vivado_HLS_Tutorial** directory.*

---

Step 1: Create a Vivado HLS IP Block

Create two HLS blocks for the Vivado IP Catalog using the provided Tcl script. The script runs HLS C-synthesis, runs RTL co-simulation, and package the IP for the two HLS designs (hls_real2xfft and hls_xfft2real).

1. Open the Vivado HLS Command Prompt.
   a. On Windows, go to **Start > All Programs > Xilinx Design Tools > Vivado 2014.1 > Vivado HLS > Vivado HLS 2014.1 Command Prompt**
   b. On Linux, open a new shell.

   ![Vivado HLS Command Prompt](image)

   **Figure 266: Vivado HLS Command Prompt**

2. Using the command prompt window, change the directory to **Vivado_HLS_Tutorial\Using_IP_with_SysGen\lab1.**

3. Type **vivado_hls –f run_hls.tcl** to create the HLS IP.
A key aspect of the Tcl script used to create this IP is the command `export_design –format sysgen`. This command creates an IP package for System Generator. When the script completes there is a Vivado HLS project directories fir_prj, which contains the HLS IP, including the IP package for use in a System Generator for DSP design.

The remainder of this tutorial exercise shows how to integrate the Vivado HLS IP block into a System Generator design.

**Step 2: Open the System Generator Project**

1. Open System Generator for DSP.
   a. On Windows use the desktop icon.
   b. On Linux, open a new shell and type `sysgen`.

2. When Matlab invokes, click the **Open** toolbar button.

3. Navigate to the tutorial directory Vivado_HLS_Tutorial\Using_IP_with_SysGen\lab1 and select the file fir_sysgen.mdl.
When System Generator invokes, all blocks and ports except the HLS IP are already instantiated in the design.

4. Right-click in the canvas and select Xilinx BlockAdd.

5. Type “hls” in the Add Block field.

6. Select Vivado HLS.
7. Double-click the **Vivado HLS** block to open the Vivado HLS dialog box.
8. Navigate to the fir_prj project and select the solution1 folder.

**IMPORTANT:** System Generator for DSP uses the location of the solution folder to identify the IP.

9. Click **OK** to load the IP block.

The FIR IP block is instantiated into the design.

10. Connect the design I/O ports to the ports on the FIR IP block.
11. Ensure the simulation stop time says 300.
12. Click the Run button on the toolbar to execute simulation.
13. Double-click the Scope block to view the simulation waveforms.

Conclusion
In this tutorial, you learned:
- How to create Vivado HLS IP using a Tcl script.
- How to import an HLS design as IP into System Generator for DSP.