

Vivado Design Suite Tutorial

Logic Simulation

UG937 (v2019.1) June 4, 2019



Revision History

Section	Revision Summary
06/04/2019 Version 2019.1	
System Verilog Feature	Added new chapter.

Table of Contents

Revision History	2
Vivado Simulator Overview.....	5
Introduction	5
Tutorial Description	6
Locating Tutorial Design Files	7
Software and Hardware Requirements	8
Lab 1: Running the Simulator in Vivado IDE	9
Introduction	9
Step 1: Creating a New Project	9
Step 2: Adding IP from the IP Catalog	15
Step 3: Running Behavioral Simulation	21
Conclusion	23
Lab 2: Debugging the Design	24
Introduction	24
Step 1: Opening the Project	24
Step 2: Displaying Signal Waveforms	25
Step 3: Using the Analog Wave Viewer	27
Step 4: Working with the Waveform Window	29
Step 5: Changing Signal Properties	32
Step 6: Saving the Waveform Configuration	34
Step 7: Re-Simulating the Design	36
Step 8: Using Cursors, Markers, and Measuring Time	37
Step 9: Debugging with Breakpoints	40
Step 10: Relaunch Simulation	46
Conclusion	48
Lab 3: Running Simulation in Batch Mode.....	49
Introduction	49
Step 1: Preparing the Simulation	49



Step 2: Building the Simulation Snapshot	51
Step 3: Manually Simulating the Design	53
Conclusion	54
System Verilog Feature	55
Introduction	55
Creating an Example Design	55
Functional Coverage	62
Random Constraint	67
Legal Notices	70
Please Read: Important Legal Notices	70



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

Introduction

This Xilinx[®] Vivado[®] Design Suite tutorial provides designers with an in-depth introduction to the Vivado simulator.



VIDEO: You can also learn more about the Vivado simulator by viewing the quick take video at [Vivado Logic Simulation](#).

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



- [Designing FPGAs Using the Vivado Design Suite 1 Training Course](#)
- [Designing FPGAs Using the Vivado Design Suite 2 Training Course](#)
- [Designing FPGAs Using the Vivado Design Suite 2 Training Course](#)

The Vivado simulator is a Hardware Description Language (HDL) simulator that lets you perform behavioral, functional, and timing simulations for VHDL, Verilog, and mixed-language designs. The Vivado simulator environment includes the following key elements:

1. `xvhdl` and `xvlog`: Parsers for VHDL and Verilog files, respectively, that store the parsed files into an HDL library on disk.
2. `xelab`: HDL elaborator and linker command. For a given top-level unit, `xelab` loads up all sub-design units, translates the design units into executable code, and links the generated executable code with the simulation kernel to create an executable simulation snapshot.
3. `xsim`: Vivado simulation command that loads a simulation snapshot to effect a batch mode simulation, or a GUI or Tcl-based interactive simulation environment.
4. Vivado Integrated Design Environment (IDE): An interactive design-editing environment that provides the simulator user-interface.

Tutorial Description

This tutorial demonstrates a design flow in which you can use the Vivado simulator for performing behavioral, functional, or timing simulation from the Vivado Integrated Design Environment (IDE).



IMPORTANT: Tutorial files are configured to run the Vivado simulator in a Windows environment. To run elements of this tutorial under the Linux operating system, some file modifications may be necessary.

You can run the Vivado simulator in both Project Mode (using a Vivado design project to manage design sources and the design flow) and in Non-Project mode (managing the design more directly). For more information about Project Mode and Non-Project Mode, refer to the *Vivado Design Suite User Guide: Design Flows Overview* ([UG892](#)).

Figure 1 shows a block diagram of the tutorial design.

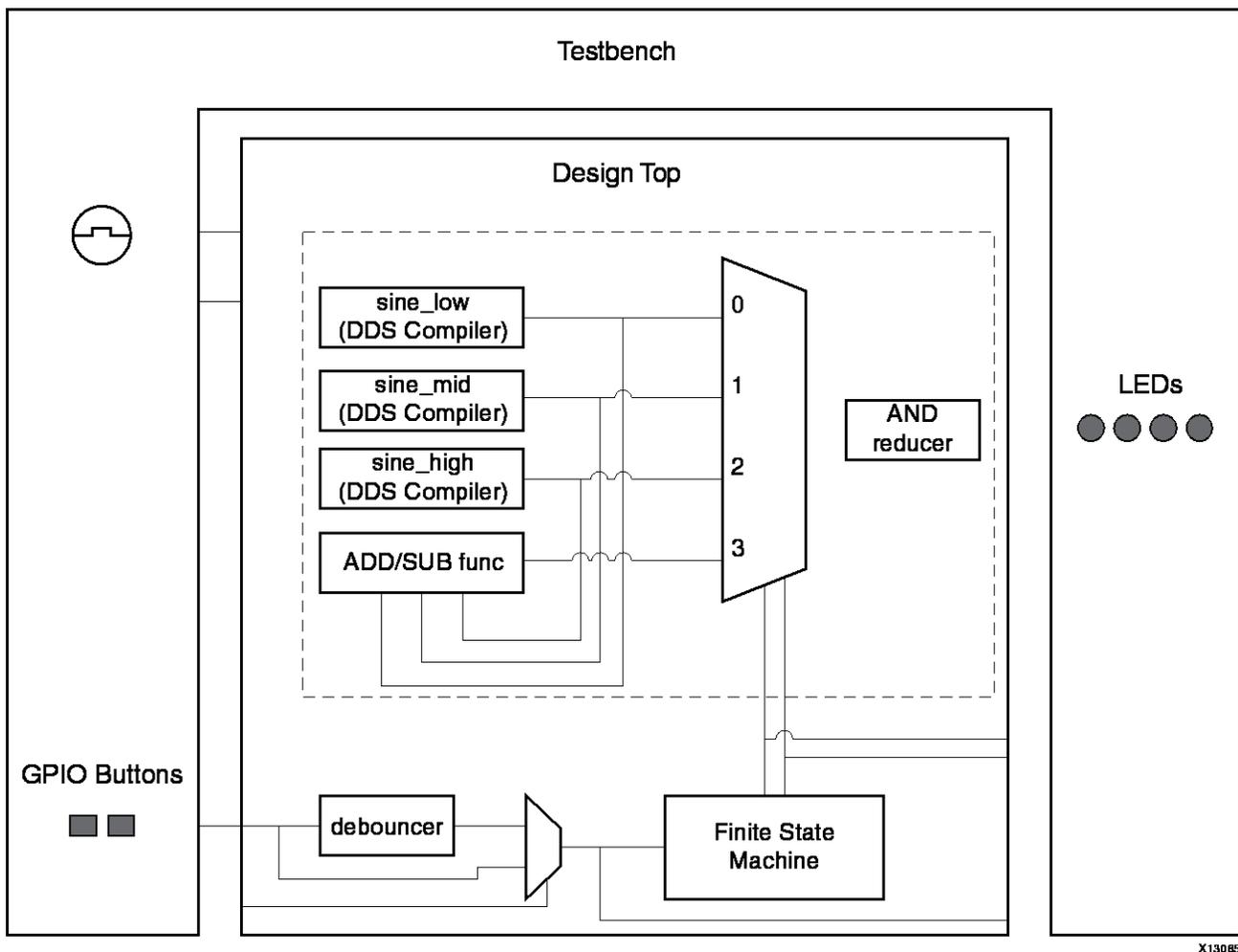


Figure 1: Tutorial Design

The tutorial design consists of the following blocks:

- A sine wave generator that generates high, medium, and low frequency sine waves; plus an amplitude sine wave (`sinegen.vhd`).
- DDS compilers that generate low, middle, and high frequency waves: (`sine_low.vhd`, `sine_mid.vhd`, and `sine_high.vhd`).
- A Finite State Machine (FSM) to select one of the four sine waves (`fsm.vhd`).
- A debouncer that enables switch-selection between the raw and the debounced version of the sine wave selector (`debounce.vhd`).
- A design top module that resets FSM and the sine wave generator, and then multiplexes the sine select results to the LED output (`sinegen_demo.vhd`).
- A simple testbench (`testbench.v`) to initiate the sine wave generator design that:
 - Generates a 200 MHz input clock for the design system clock, `sys_clk_p`.
 - Generates GPIO button selections.
 - Controls raw and debounced sine wave select.

Note: For more information about testbenches, see *Writing Efficient Testbenches* ([XAPP199](#)).

Locating Tutorial Design Files

There are separate project files and sources for each of the labs in this tutorial. You can find these at the link provided below or under **Support > Documentation > Development Tools (Product Type) > Hardware Development (Product Category) > Vivado Design Suite – HLx Editions (Product) > Tutorials (Doc Type)** on the Xilinx.com website.

1. Download the [reference design files](#).
2. Extract the zip file contents into any write-accessible location.

This tutorial refers to the extracted file contents of `ug937-design-files` directory as `<Extract_Dir>`.



RECOMMENDED: You modify the tutorial design data while working through this tutorial. Use a new copy of the design files each time you start this tutorial.

The following table describes the contents of the `ug937-design-files.zip` file.

Directories/Files	Description
<code>/completed</code>	Contains the completed files, and a Vivado 2019.x project of the tutorial design for reference. (x denotes the latest version of Vivado 2019 IDE)
<code>/scripts</code>	Contains the scripts you run during the tutorial.
<code>/sim</code>	Contains the <code>testbench.v</code> file.
<code>/sources</code>	Contains the HDL files necessary for the functional simulation.
<code>readme.txt</code>	<code>readme.txt</code> is a readme file about the contents and version history of this tutorial design.

Software and Hardware Requirements

This tutorial requires that the 2019.1 Vivado Design Suite software release is installed. The following partial list describes the operating systems that the Vivado Design Suite supports on x86 and x86-64 processor architectures:

Microsoft Windows Support:

- Windows 8.1 Professional (32-bit and 64-bit), English/Japanese
- Windows 7 and 7 SP1 Professional (32-bit and 64-bit), English/Japanese

Linux Support:

- Red Hat Enterprise Workstation 6.4 and 6.5 (32-bit and 64-bit)
- SUSE Linux Enterprise 11 (32-bit and 64-bit)
- Cent OS 6.4 and 6.5 (64-bit)

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

Introduction

In this lab, you create a new Vivado Design Suite project, add HDL design sources, add IP from the Xilinx IP catalog, and generate IP outputs needed for simulation. Then you run a behavioral simulation on an elaborated RTL design.

Step 1: Creating a New Project

The Vivado Integrated Design Environment (IDE) (Figure 2) lets you launch simulation from within design projects, automatically generating the necessary simulation commands and files.

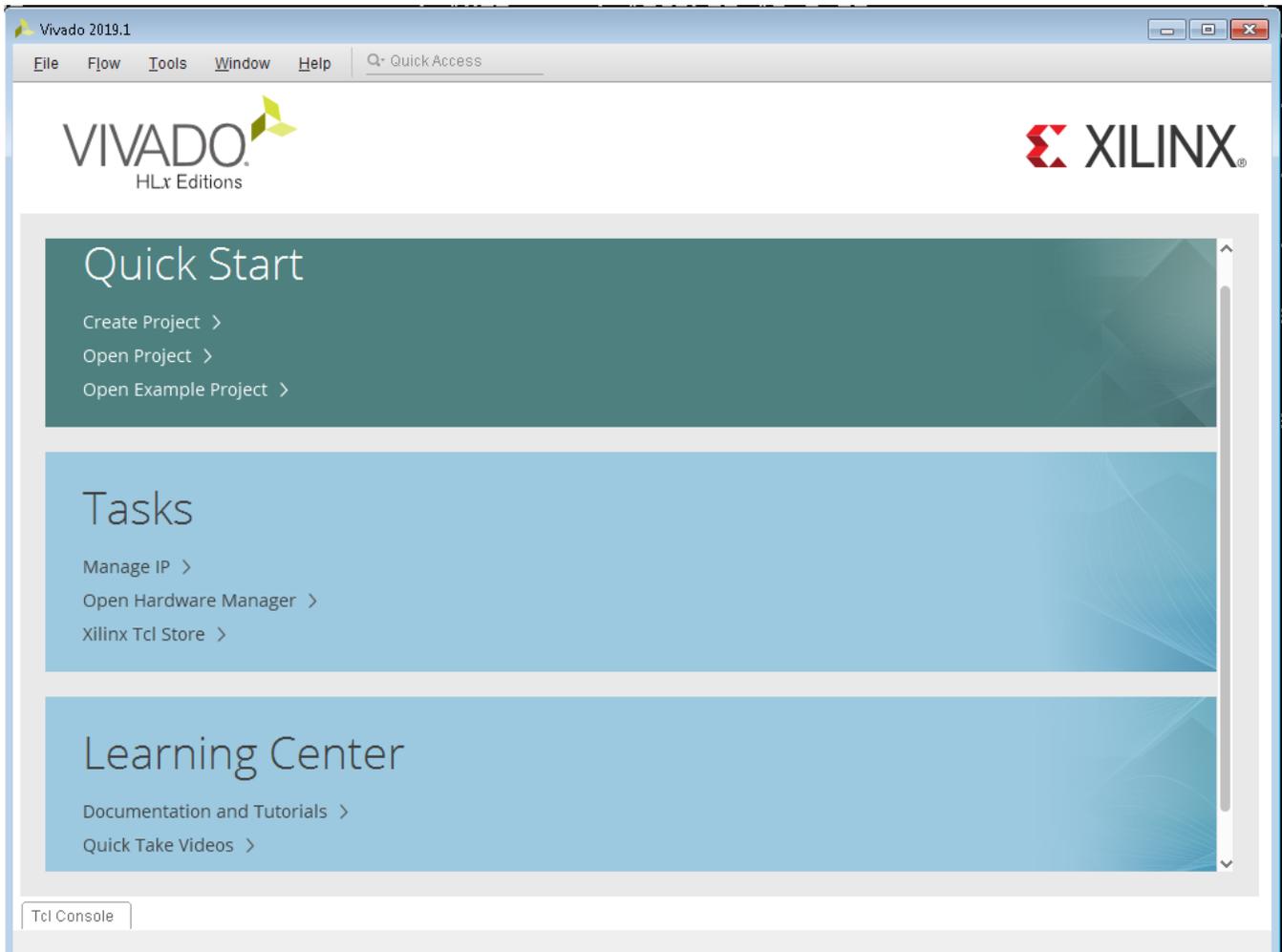


Figure 2: Vivado IDE - Getting Started Page

Create a new project for managing source files, add IP to the design, and run behavioral simulation.

1. On Windows, launch the Vivado IDE:

Start > All Programs > Xilinx Design Tools > Vivado 2019.x > Vivado 2019.x

(**x** denotes the latest version of Vivado 2019 IDE)

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

2. In the Vivado IDE Getting started page, click **Create Project**.

3. In the **New project** dialog box, click **Next** and enter a project name: `project_xsim`.

4. For the **Project Location**, browse to the folder containing the extracted tutorial data, `<Extract_Dir>`. Make sure to check the **Create project subdirectory** option and click **Next**. (Figure 3).

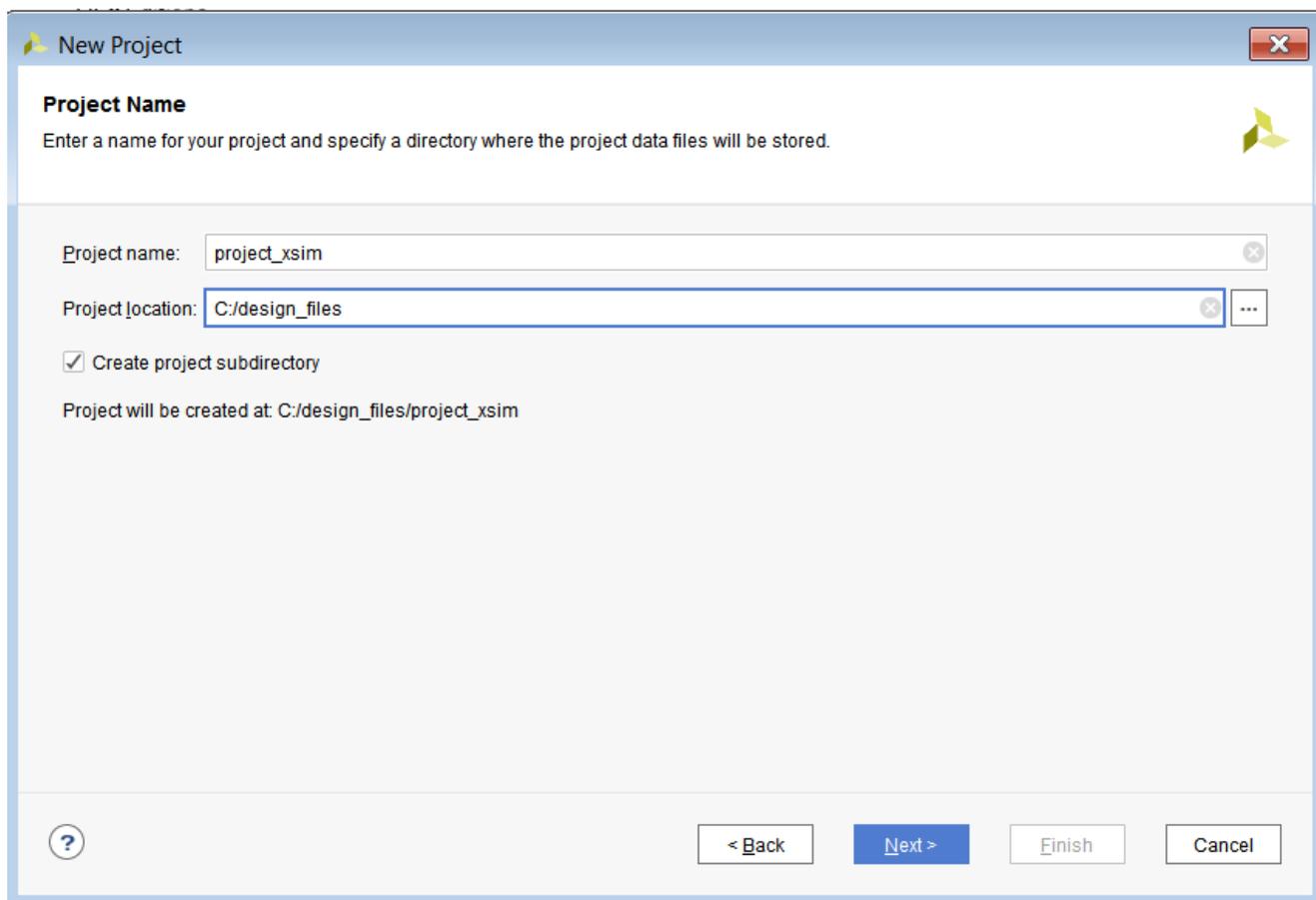


Figure 3: Create Project

Note: Create project subdirectory option is preselected.

5. In the Project Type dialog box, select **RTL Project** and click **Next**.

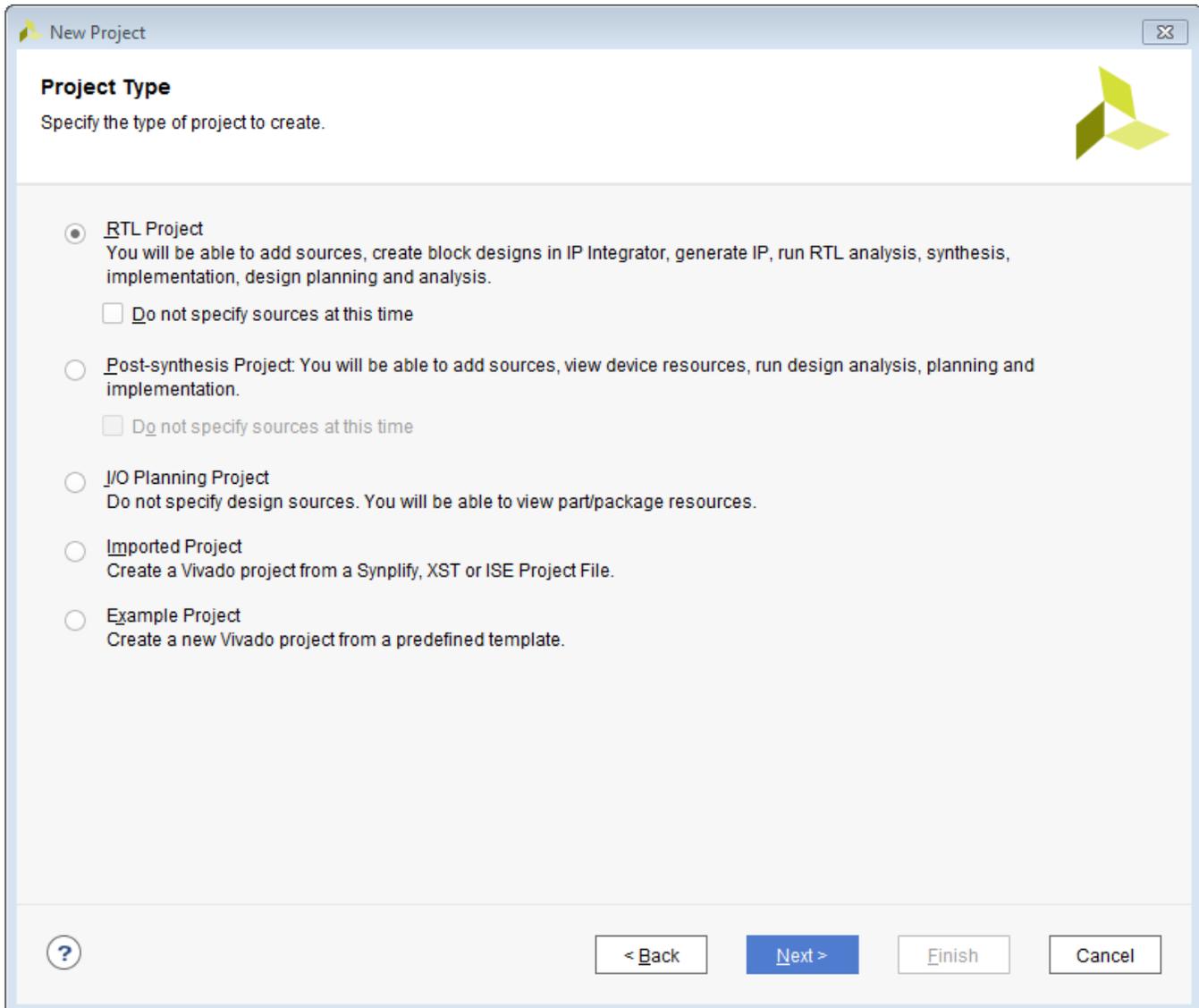


Figure 4: Selecting RTL Project

6. In the Add Source dialog box, click **Add Directories** and add the extracted tutorial design data:

- `<Extract_Dir>/sources`
- `<Extract_Dir>/sim`

Note: You can press the *Ctrl* key to click and select multiple files or directories.

7. Set the **Target Language** to **Verilog** to indicate the netlist language for synthesis.

8. Set the **Simulator Language** to **Mixed** as seen in [Figure 5](#).

The Simulator Language indicates which languages the logic simulator supports or requires. Vivado Design Site ensures the availability of simulation models of any IP cores in the design by using the available synthesis files to generate the required language-specific structural simulation model when

generating output targets. For more information on working with IP cores and the Xilinx IP Catalog, refer to the *Vivado Design Suite User Guide: Design with IP* ([UG896](#)). You can also work through the *Vivado Design Suite Tutorial: Designing with IP* ([UG939](#)).

9. Click **Next**.

10. Click **Next** to bypass the Add Constraints dialog box.

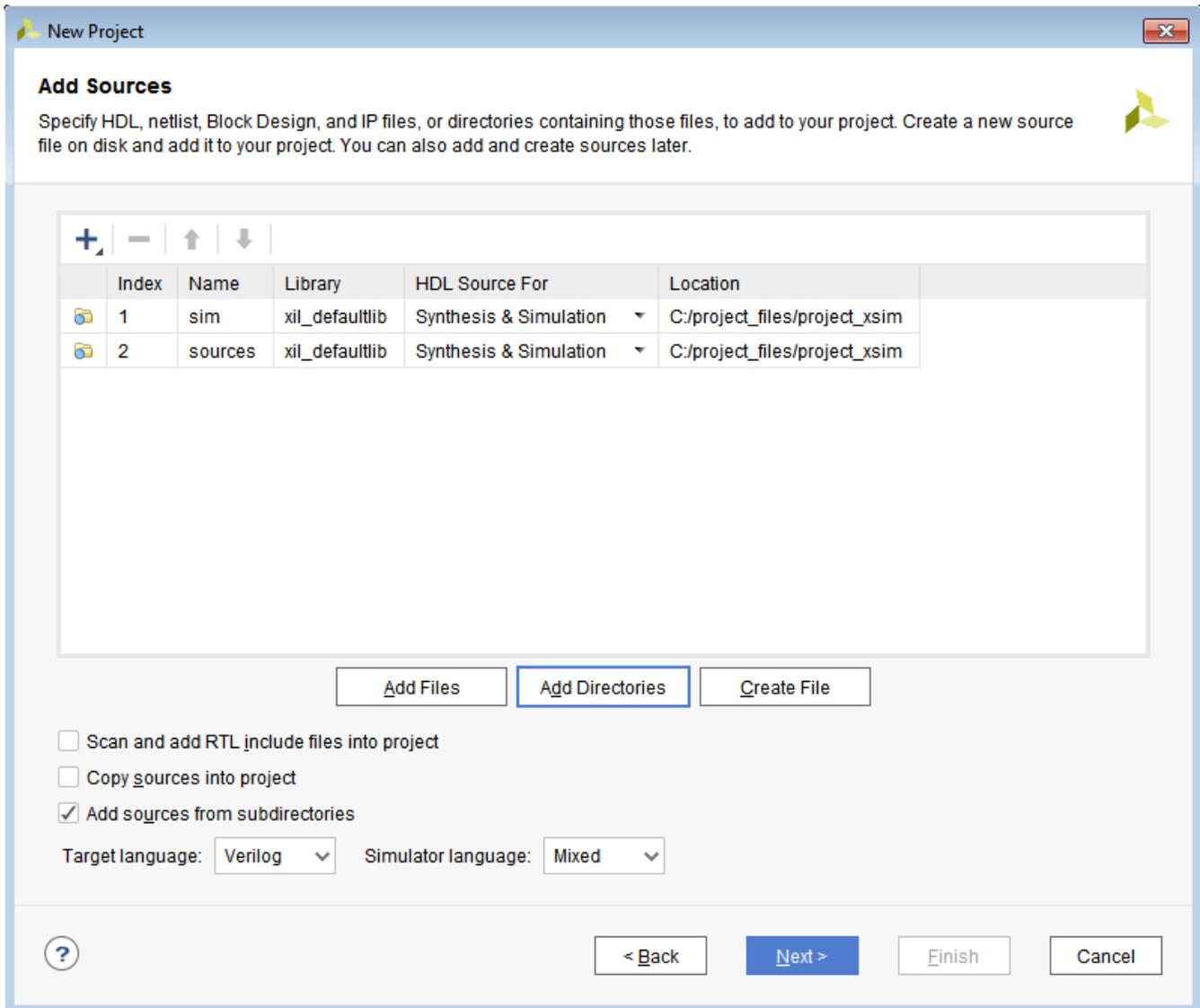


Figure 5: Add Sources

In the Default Part dialog box ([Figure 6](#)), select **Boards**, and then select either **Kintex-7 KC705 Evaluation Platform** for 7-Series or **Kintex-UltraScale KCU105 Evaluation Platform** for UltraScale devices and click **Next**.

Note: Add sources from subdirectories option is preselected.

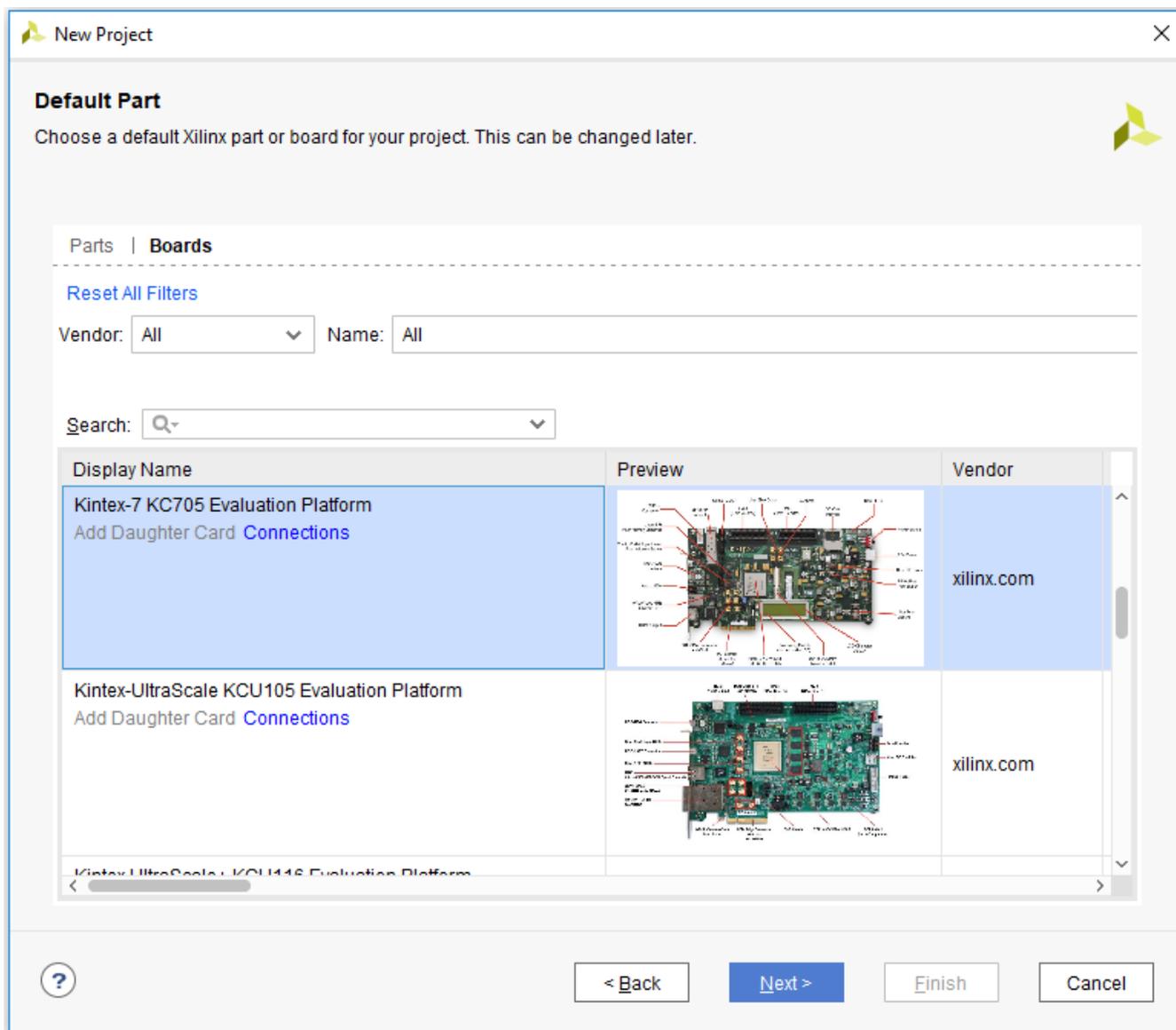


Figure 6: Specify Default Part or Board

11. Review the New Project Summary dialog box.
12. Click **Finish** to create the project.

Vivado opens the new project in the Vivado IDE, using the default view layout (Figure 7).

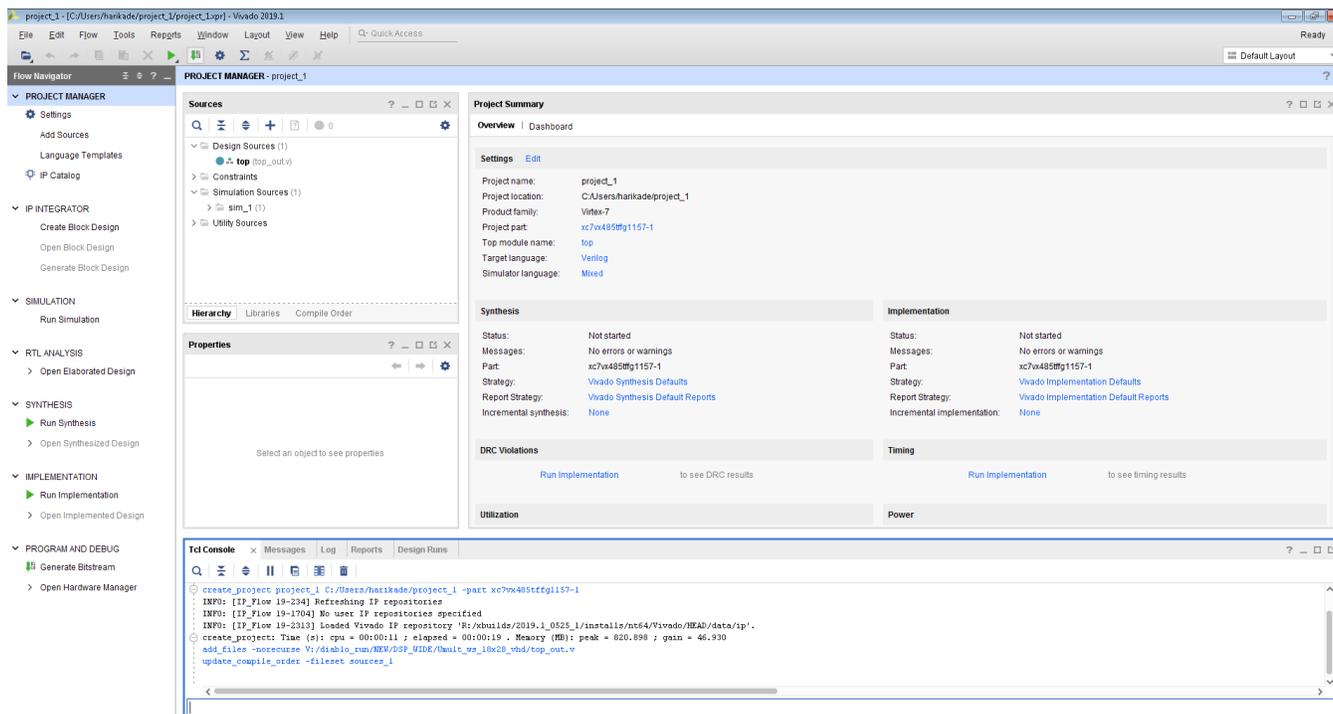


Figure 7: Vivado IDE - Default Layout

Step 2: Adding IP from the IP Catalog

The Sources window displays the source files that you have added during project creation. The **Hierarchy** tab displays the hierarchical view of the source files.

1. Click the > icon in the Sources window to expand the folders as shown in [Figure 8](#). Expand all > button can be used to view all the files at all levels of hierarchy.

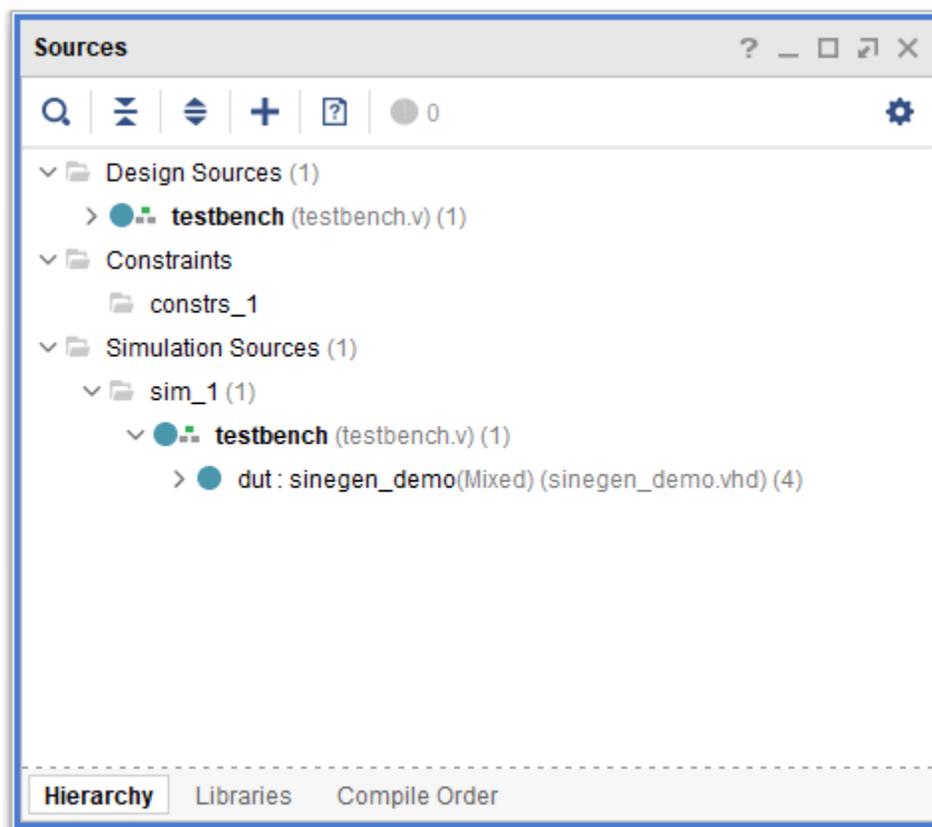


Figure 8: Sources window

Notice that the Sine wave generator (sinegen.vhd) references cells that are not found in the current design sources. In the Sources window, the missing design sources are marked by the missing source icon .

Note: The missing source icon is used to view only the missing sources. This is useful in viewing the missing sources in larger designs.

Now, add the `sine_high`, `sine_mid`, and `sine_low` modules to the project from the Xilinx IP Catalog.

Adding Sine High

1. In the Flow Navigator, select the **IP Catalog** button.

The IP Catalog opens in the graphical windows area. For more information on the specifics of the Vivado IDE, refer to the *Vivado Design Suite User Guide: Using the Vivado IDE* ([UG893](#)).

2. In the search field of the IP Catalog, type `DDS`.

The Vivado IDE highlights the DDS Compilers in the IP catalog.

3. Under any category, double-click the **DDS Compiler**.

The Customize IP wizard opens ([Figure 9](#)).

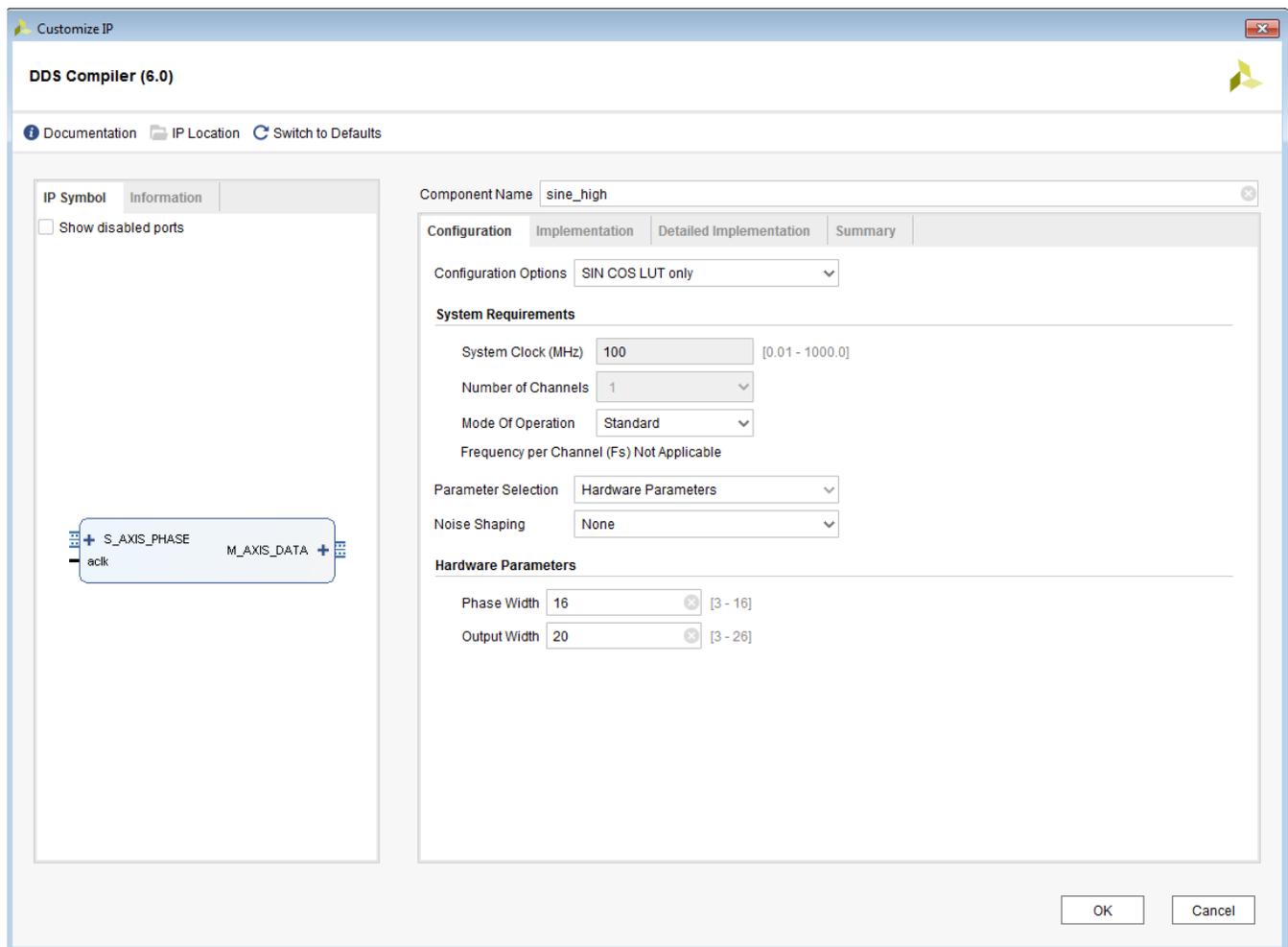
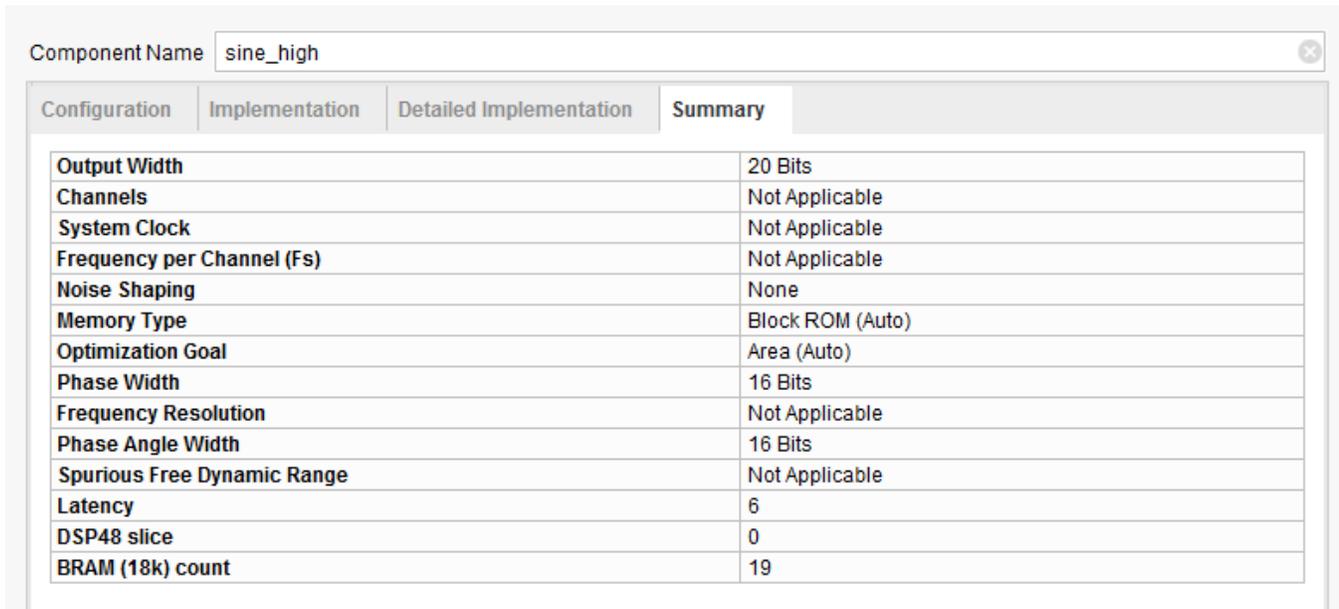


Figure 9: Customize IP - DDS Compiler

4. In the IP Symbol on the left, ensure that **Show disabled ports** is unchecked.
5. Specify the following on the **Configuration** tab:
 - o **Component Name:** type `sine_high`
 - o **Configuration Options:** select **SIN COS LUT only**

- **Noise Shaping:** select **None**
 - Under **Hardware Parameters**, set **Phase Width** to 16 and **Output Width** to 20
6. On the **Implementation** tab, set **Output Selection** to **Sine**
 7. On the **Detailed Implementation** tab, set **Control Signals** to **ARESETn (active-Low)**
 8. On the **Summary** tab, review the settings and click **OK** (Figure 10).
 9. In the Create Directory Dialog Box, Click **OK**.



Configuration	Implementation	Detailed Implementation	Summary
Output Width			20 Bits
Channels			Not Applicable
System Clock			Not Applicable
Frequency per Channel (Fs)			Not Applicable
Noise Shaping			None
Memory Type			Block ROM (Auto)
Optimization Goal			Area (Auto)
Phase Width			16 Bits
Frequency Resolution			Not Applicable
Phase Angle Width			16 Bits
Spurious Free Dynamic Range			Not Applicable
Latency			6
DSP48 slice			0
BRAM (18k) count			19

Figure 10: Sine High Summary

When the `sine_high` IP core is added to the design, the output products required to support the IP in the design must be generated. The Generate Output Products dialog box displays, as shown in [Figure 11](#).

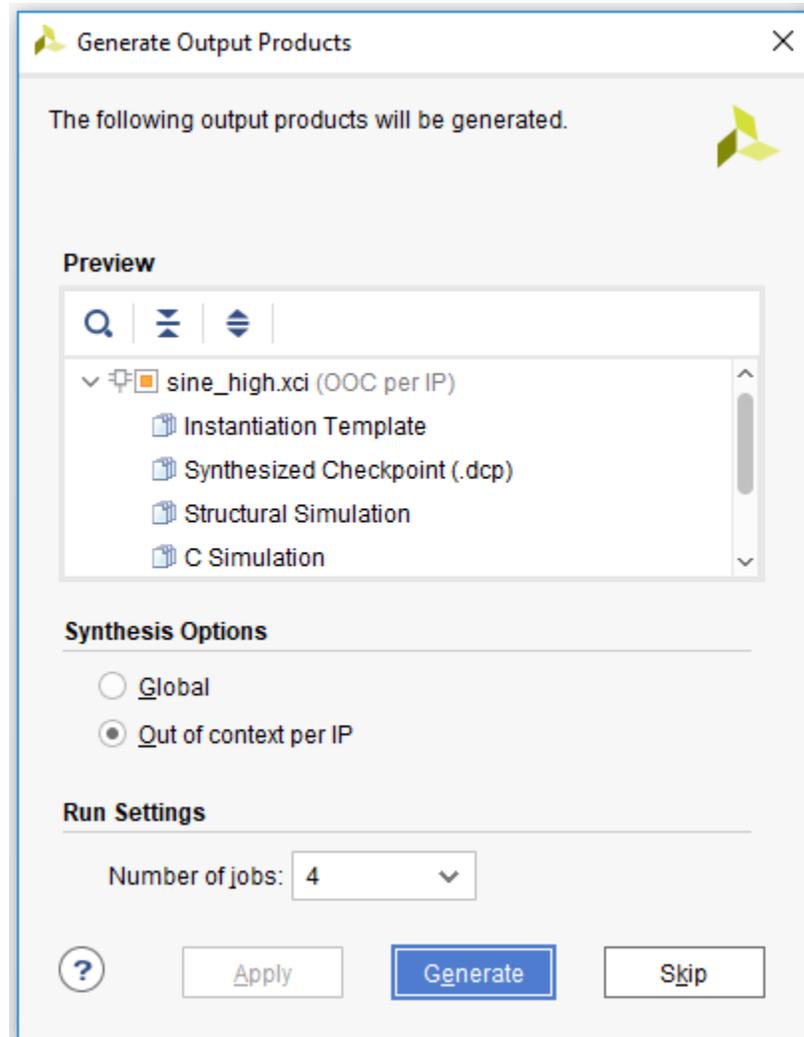


Figure 11: Generate Output Products

The output products allow the IP to be synthesized, simulated, and implemented as part of the design. For more information on working with IP cores and the Xilinx IP Catalog, refer to the *Vivado Design Suite User Guide: Design with IP* ([UG896](#)). You can also work through the *Vivado Design Suite Tutorial: Designing with IP* ([UG939](#)).

Click **Generate** to generate the default output products for `sine_high`.

Adding Sine Mid

1. In the **IP catalog**, double-click the **DDS Compiler** IP a second time.
2. Specify the following on the **Configuration** tab:
 - **Component Name:** type `sine_mid`
 - **Configuration Options:** select **SIN COS LUT only**
 - **Noise Shaping:** select **None**

- Under **Hardware Parameters**, set the **Phase Width** to 8, and the **Output Width** to 18
3. On the **Implementation** tab, set the **Output Selection** to **Sine**
 4. On the **Detailed Implementation** tab, set **Control Signals** to **ARESETn (active-Low)**
 5. Select the **Summary** tab, review the settings and click **OK** (Figure 12).

Component Name

Configuration	Implementation	Detailed Implementation	Summary
Output Width			18 Bits
Channels			Not Applicable
System Clock			Not Applicable
Frequency per Channel (Fs)			Not Applicable
Noise Shaping			None
Memory Type			Block ROM (Auto)
Optimization Goal			Area (Auto)
Phase Width			8 Bits
Frequency Resolution			Not Applicable
Phase Angle Width			8 Bits
Spurious Free Dynamic Range			Not Applicable
Latency			2
DSP48 slice			0
BRAM (18k) count			1

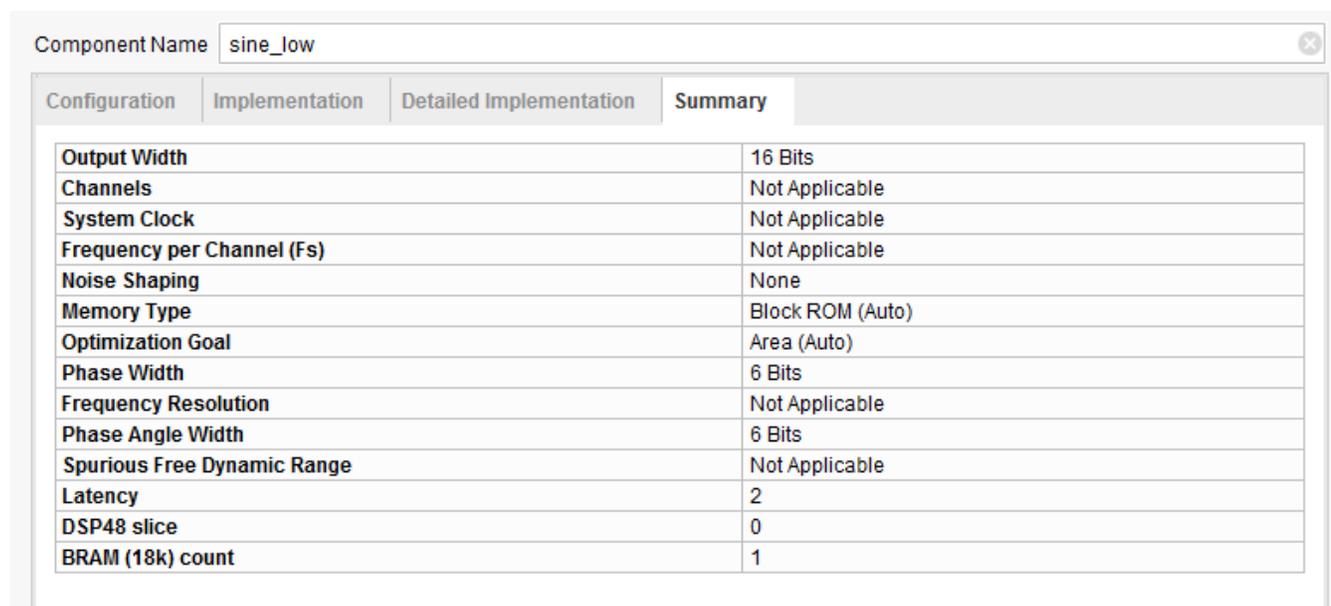
Figure 12: Sine Mid Summary

When the `sine_mid` IP core is added to the design, the Generate Output Products dialog box displays to generate the output products required to support the IP in the design.

6. Click **Generate** to generate the default output products for `sine_mid`. A dialog box opens saying that the Out of context module run was launched for generating output products. Click **OK**.

Adding Sine Low

1. In the **IP catalog**, double-click the **DDS Compiler** IP for the third time.
2. Specify the following on the **Configuration** tab:
 - **Component Name:** type `sine_low`
 - **Configuration Options:** select **SIN COS LUT only**
 - **Noise Shaping:** select **None**
 - Under **Hardware Parameters**, set the **Phase Width** to 6 and the **Output Width** to 16
3. On the **Implementation** tab, set the **Output Selection** to **Sine**.
4. On the **Detailed Implementation** tab, set **Control Signals** to **ARESETn (active-Low)**
5. Select the **Summary** tab, review the settings as seen in [Figure 13](#), and click **OK**.



Component Name: `sine_low`

Configuration	Implementation	Detailed Implementation	Summary
Output Width			16 Bits
Channels			Not Applicable
System Clock			Not Applicable
Frequency per Channel (Fs)			Not Applicable
Noise Shaping			None
Memory Type			Block ROM (Auto)
Optimization Goal			Area (Auto)
Phase Width			6 Bits
Frequency Resolution			Not Applicable
Phase Angle Width			6 Bits
Spurious Free Dynamic Range			Not Applicable
Latency			2
DSP48 slice			0
BRAM (18k) count			1

Figure 13: Sine Low Summary

When the `sine_low` IP core is added to the design, the Generate Output Products dialog box displays to generate the output products required to support the IP in the design.

6. Click **Generate** to generate the default output products for `sine_low`. A dialog box opens saying that the Out of context module run was launched for generating output products. Click **OK**.

Step 3: Running Behavioral Simulation

After you have created a Vivado project for the tutorial design, you set up and launch Vivado simulator to run behavioral simulation. Set the behavioral simulation properties in Vivado tools:

- In the Flow Navigator, Right-click **Simulation** and then click **Simulation Settings**. Alternatively, click Settings in the Flow Navigator under Project Manager to open the Settings window. Select Simulation from the Settings window. The following defaults are automatically set:
 - Simulation set:** select **sim_1**
 - Simulation top-module name:** set **testbench**
- In the **Elaboration** tab (Figure 14), ensure that the debug level is set to **typical**, which is the default value.

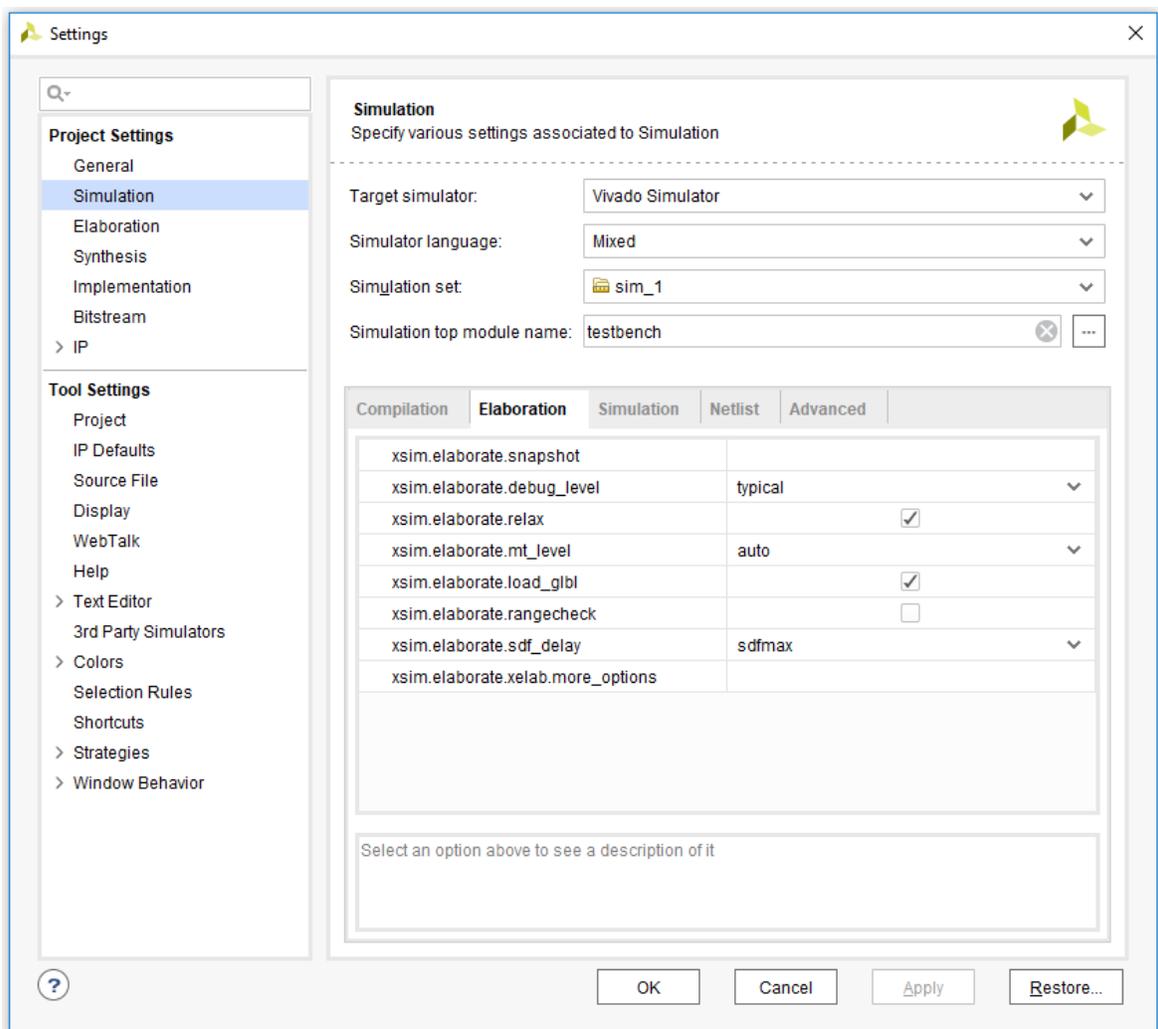


Figure 14: Simulation Settings: Elaboration

- In the **Simulation** tab, observe that the **Simulation Run Time** is 1000ns.

4. Click **OK**.

With the simulation settings properly configured, you can launch Vivado simulator to perform a behavioral simulation of the design.

5. In the Flow Navigator, click **Run Simulation > Run Behavioral Simulation**.

Functional and timing simulations are available post-synthesis and post-implementation. Those simulations are outside the scope of this tutorial.

When you launch the Run Behavioral Simulation command, the Vivado tool runs `xvlog` and `xvhdl` to analyze the design and `xelab` in the background to elaborate and compile the design into a simulation snapshot, which the Vivado simulator can run. When that process is complete, the Vivado tool launches `xsim` to run the simulation.

In the Vivado IDE, the simulator GUI opens after successfully parsing and compiling the design (Figure 15). By default, the top-level HDL objects display in the Waveform window.

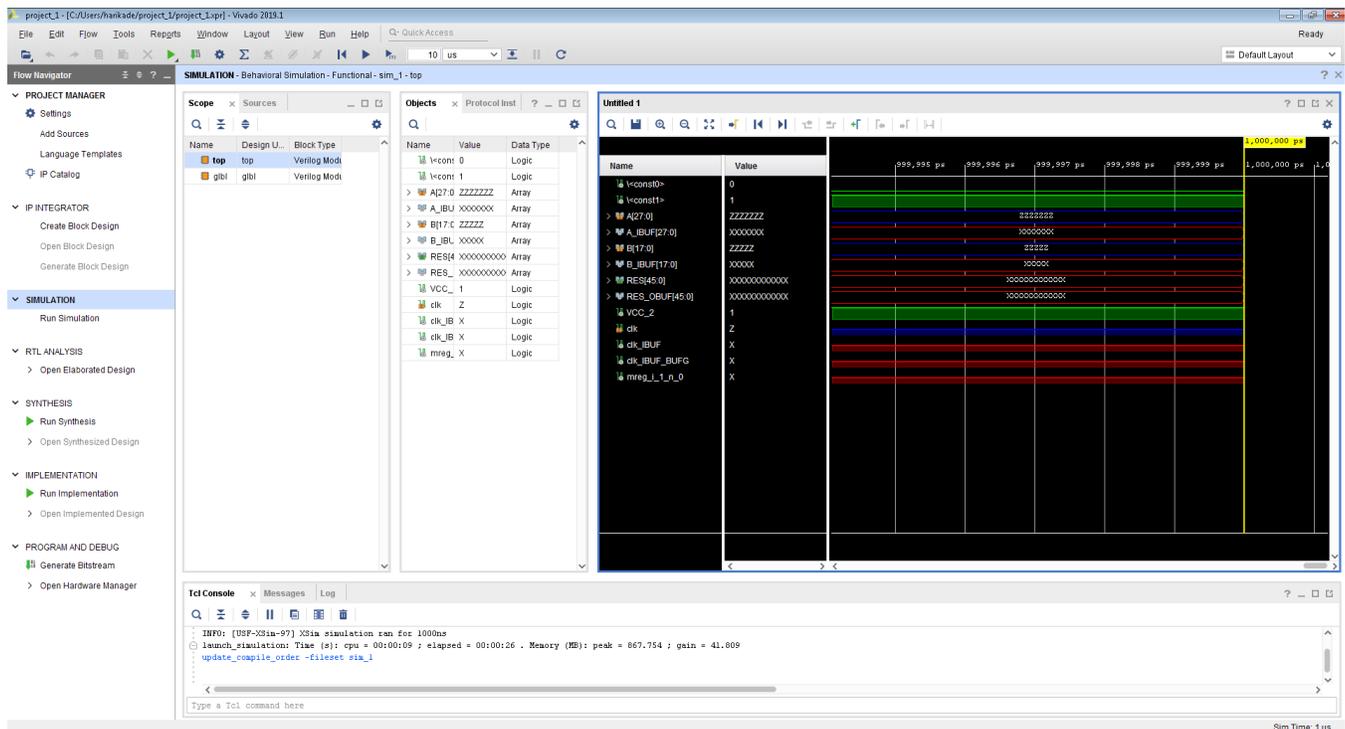


Figure 15: Vivado Simulation GUI

Conclusion

In this lab, you have created a new Vivado Design Suite project, added HDL design sources, added IP from the Xilinx IP catalog and generated IP outputs needed for simulation, and then run behavioral simulation on the elaborated RTL design.

This concludes Lab #1. You can continue Lab #2 at this time by starting at [Step 2: Displaying Signal Waveforms](#).

You can also close the simulation, project, and the Vivado IDE to start Lab #2 at a later time.

1. Click **File > Close Simulation** to close the open simulation.
2. Select **OK** if prompted to confirm closing the simulation.
3. Click **File > Close Project** to close the open project.
4. Click **File > Exit** to exit the Vivado tool.

Introduction

The Vivado simulator GUI contains the Waveform window, and Object and Scope Windows. It provides a set of debugging capabilities to quickly examine, debug, and fix design problems. See the *Vivado Design Suite User Guide: Logic Simulation* ([UG900](#)) for more information about the GUI components.

In this lab, you:

- Enable debug capabilities
- Examine a design bug
- Use debug features to find the root cause of the bug
- Make changes to the code
- Re-compile and re-launch the simulation

Step 1: Opening the Project

This lab continues from the end of Lab #1 in this tutorial. You must complete Lab #1 prior to beginning Lab #2. If you closed the Vivado IDE, or the tutorial project, or the simulation at the end of Lab #1, you must reopen them.

Start by loading the Vivado Integrated Design Environment (IDE).

Start > All Programs > Xilinx Design Tools > Vivado 2019.x > Vivado 2019.x

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

Note: As an alternative, click the **Vivado 2019.x** Desktop icon to start the Vivado IDE.

The Vivado IDE opens. Now, open the project from Lab #1, and run behavioral simulation.

1. From the main menu, click **File > Project > Open Recent** and select `project_xsim`, which you saved in Lab #1.
2. After the project has opened, from the Flow Navigator click **Run Simulation > Run Behavioral Simulation**.

The Vivado simulator compiles your design and loads the simulation snapshot.

Step 2: Displaying Signal Waveforms

In this section, you examine features of the Vivado simulator GUI that help you monitor signals and analyze simulation results, including:

- Running and restarting the simulation to review the design functionality, using signals in the Waveform window and messages from the testbench shown in the Tcl console.
- Adding signals from the testbench and other design units to the Waveform window so you can monitor their status.
- Adding groups and dividers to better identify signals in the Waveform window.
- Changing signal and wave properties to better interpret and review the signals in the Waveform window.
- Using markers and cursors to highlight key events in the simulation and to perform zoom and time measurement features.
- Using multiple waveform configurations.

Add and Monitor Signals

The focus of the tutorial design is to generate sine waves with different frequencies. To observe the function of the circuit, you monitor a few signals from the design. Before running simulation for a specified time, you can add signals to the wave window to observe the signals as they transition to different states over the course of the simulation.

By default, the Vivado simulator adds simulation objects from the testbench to the Waveform window. In the case of this tutorial, the following testbench signals load automatically:

- Differential clock signals (`sys_clk_p` and `sys_clk_n`). This is a 200 MHz clock generated by the testbench and is the input clock for the complete design.
- Reset signal (`reset`). Provides control to reset the circuit.
- GPIO buttons (`gpio_buttons[1:0]`). Provides control signals to select different frequency sine waves.
- GPIO switch (`gpio_switch`). Provides a control switch to enable or disable debouncer logic.
- LEDs (`leds_n[3:0]`). A placeholder bus to display the results of the simulation.

You add some new signals to this list to monitor those signals as well.

If necessary, in the Scopes window, click the  sign to expand the testbench. (It might be expanded by default.)

An HDL scope, or scope, is defined by a declarative region in the HDL code, such as a module, function, task, process, or named blocks in Verilog. VHDL scopes include entity/architecture definitions, blocks, functions, procedures, and processes.

3. In the Scopes window, click to select the **dut** object.

The current scope of the simulation changes from the whole testbench to the selected object. The Objects window updates with all the signals and constants of the selected scope, as shown in Figure 16.

4. From the Objects window, select signals `sine[19:0]` and `sineSel[1:0]` and add them into Wave Configuration window using one of the following methods:
 - Drag and drop the selected signals into the Waveform window.
 - Right-click on the signal to open the popup menu, and select **Add to Wave Window**.

Note: You can select multiple signals by holding down the CTRL key during selection.

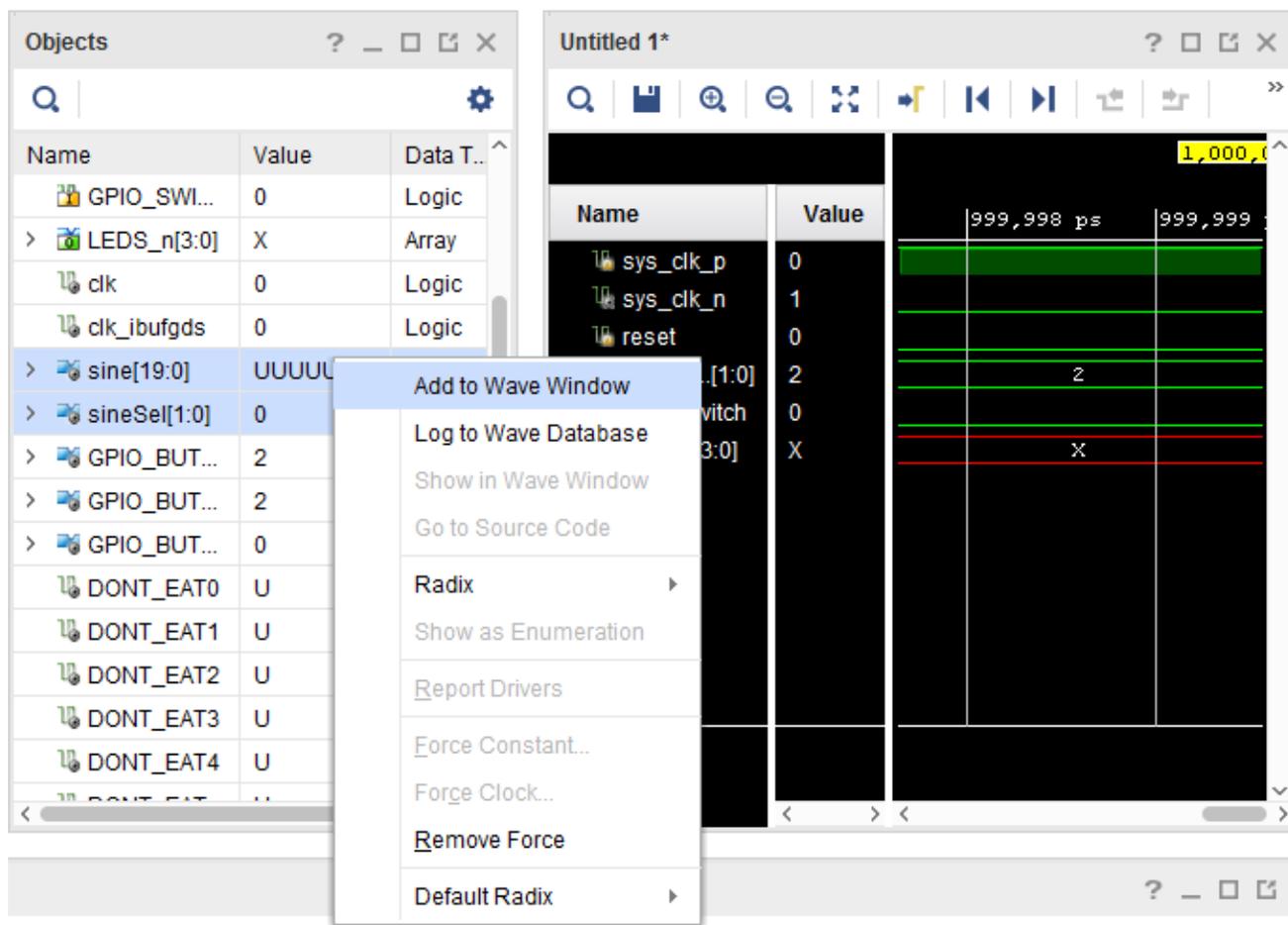


Figure 16: Add signals to Wave Window

Step 3: Using the Analog Wave Viewer

The sine[19:0] signals you are monitoring are analog signals, which you can view better in Analog wave mode. You can choose to display a given signal as Digital or Analog in the Waveform window.

1. In the Waveform window, select the **sine[19:0]** signal.
2. Right click to open the popup menu, and select **Waveform Style > Analog**, as shown in [Figure 17](#).
3. Right click to open the popup menu again, and select **Radix > Signed Decimal** as shown in [Figure 18](#).

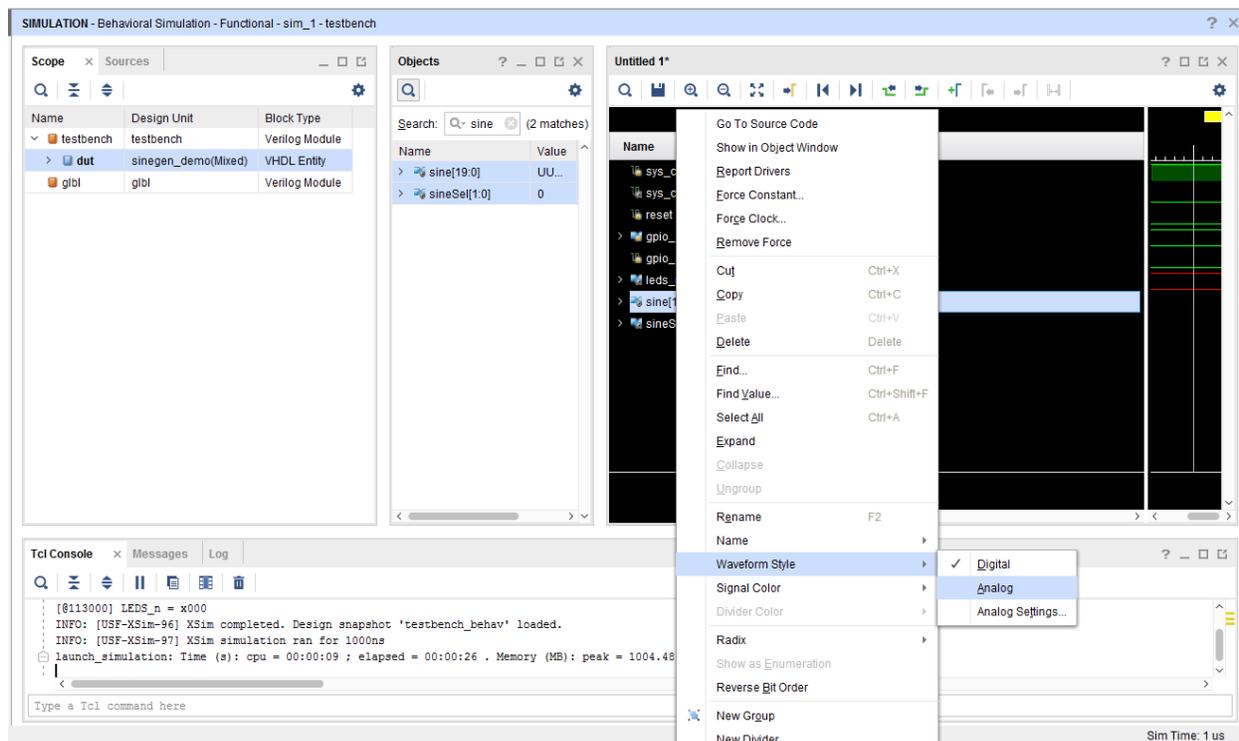


Figure 17: Enable Analog Waveform Style

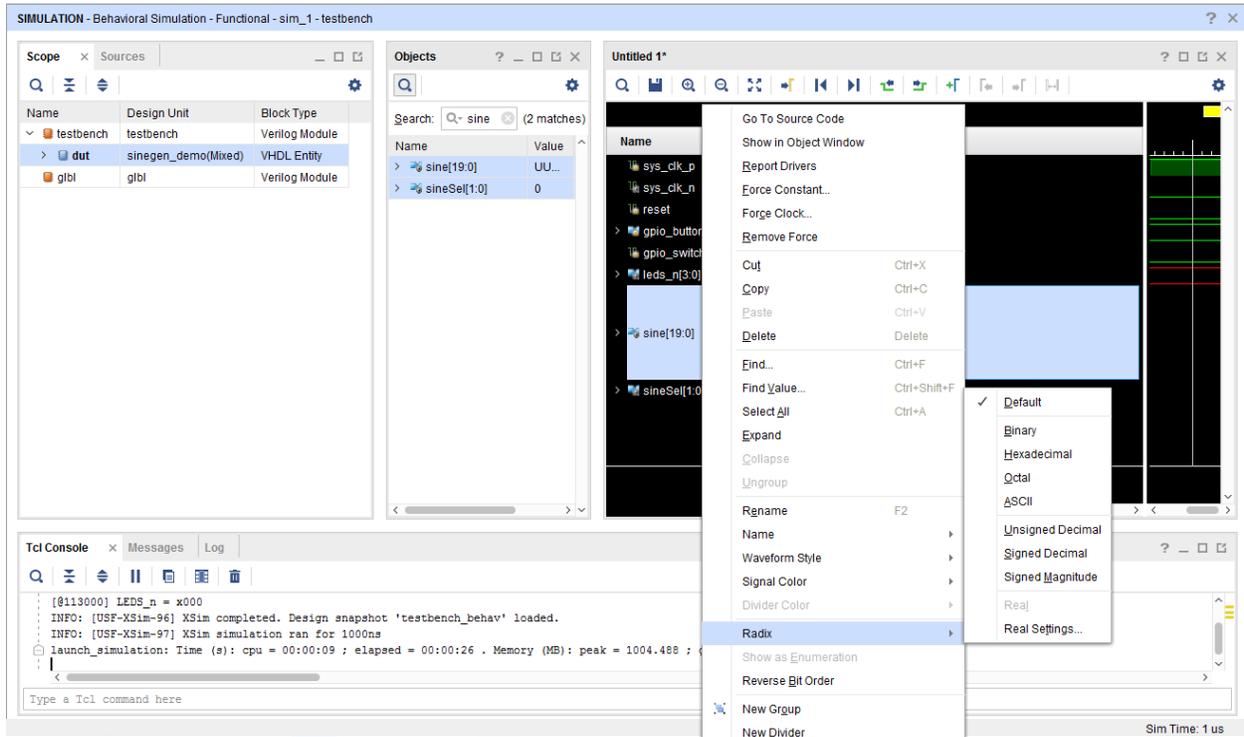


Figure 18: Enable Radix

Logging Waveforms for Debugging

The Waveform window lets you review the state of multiple signals as the simulation runs. However, due to its limited size, the number of signals you can effectively monitor in the Waveform window is limited. To identify design failures during debugging, you might need to trace more signals and objects than can be practically displayed in the Waveform window. You can log the waveforms for signals that are not displayed in the Waveform window, by writing them to the simulation waveform database (WDB). After simulation, you can review the transitions on all signals captured in the waveform database file.

In the Scope window, right-click on `dut` under `testbench`. Click **Log to Wave Database** from the options list. Select **Objects in Scope** option. The specified signals are written to a waveform database.

Using TCL Command

Enable logging of the waveform for the specified HDL objects by entering the following command in the Tcl console:

```
log_wave [get_objects /testbench/dut/*] [get_objects /testbench/dut/U_SINEGEN/*]
```

Note: See the *Vivado Design Suite Tcl Command Reference Guide* ([UG835](#)) for more information on the `log_wave` command.

This command enables signal dumping for the specified HDL objects, `/testbench/dut/*` and `/testbench/dut/U_SINEGEN/*`.

Note: * Symbol specifies all the HDL objects in a scope.

The `log_wave` command writes the specified signals to a waveform database, which is written to the simulation folder of the current project:

```
<project_name>/<project_name>.sim/sim_1/behavior
```

Step 4: Working with the Waveform Window

Now that you have configured the simulator to display and log signals of interest into the waveform database, you are ready to run the simulator again.

1. Run the simulation by clicking the **Run All** button ▶.

Observe the sine signal output in the waveform. The Wave window can be undocked from Main window layout to view it as standalone.

2. Click the **Float**  button in the title bar of the Waveform Configuration window.
3. Display the whole time spectrum in the Waveform window by clicking the **Zoom Fit** button .

Notice that the low frequency sine output is incorrect. You can view the waveform in detail by zooming  into the Waveform window. When you zoom into the waveform, you can use the horizontal and vertical scroll bars to pan down the full waveform.

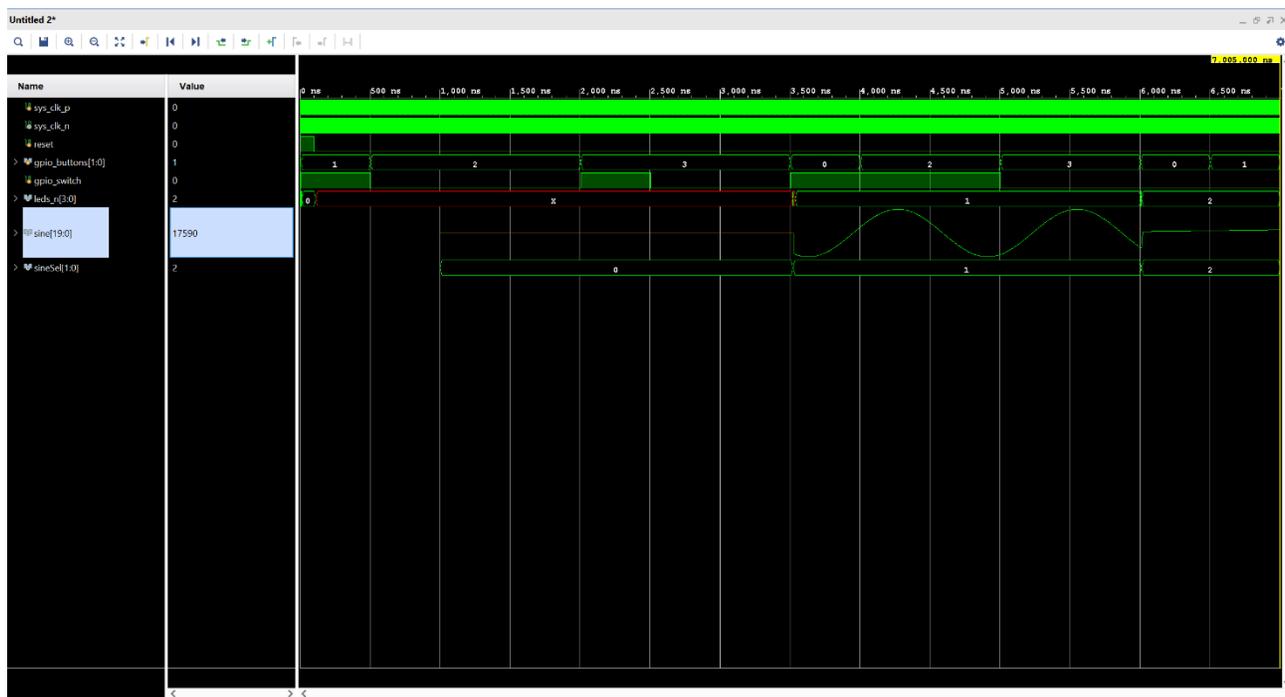


Figure 19: Design Bug - Wave View

As seen in [Figure 19](#), when the value of `sineSel` is 0, which indicates a low frequency sine selection, the analog `sine[19:0]` output is not a proper sine wave, indicating a problem in the design or the testbench.

Grouping Signals

Next, you add signals from other design units to better analyze the functionality of the whole design. When you add signals to the Waveform window, the limited size of the window makes it difficult to display all signals at the same time. Reviewing all signals would require the use of the vertical scroll bar, making the review process difficult.

You can group related signals together to make viewing them easier. With a group, you can display or hide associated signals to make the Waveform window less cluttered, and easier to understand.

1. In the Waveform window, select all signals in the testbench unit: `sys_clk_p`, `sys_clk_n`, `reset`, `gpio_buttons`, `gpio_switch`, and `leds_n`.

Note: Press and hold the **Ctrl** key, or **Shift** key, to select multiple signals.

2. With the signals selected right-click to open the popup menu and select **New Group**. Rename it as TB Signals.

Vivado simulator creates a collapsed group in the waveform configuration window. To expand the group, click **>** to the left of the group name.

3. Create another signal group called DUT Signals to group signals `sine[19:0]` and `sine_sel[1:0]`.

You can add or remove signals from a group as needed. Cut and paste signals from the list of signals in the Waveform window, or drag and drop a signal from one group into another.

You can also drag and drop a signal from the Objects window into the Waveform window, or into a group.

You can ungroup all signals, thereby eliminating the group. Select a group, right-click to open the popup menu and select **Ungroup**.

To better visualize which signals belong to which design units, add dividers to separate the signals by design unit.

Adding Dividers

Dividers let you create visual breaks between signals or groups of signals to more easily identify related objects.

1. In the Waveform window, right-click to open the popup menu and select **New Divider**. The Name dialog box opens to let you name the divider you are adding to the Waveform window.
2. Add two dividers named:
 - o Testbench
 - o SineGen
3. Move the SineGen divider above the DUT Signals group.



TIP: You can change divider names at any time by highlighting the divider name and selecting the **Rename** command from the popup menu, or change the color with **Divider Color**.

Adding Signals from Sub-modules

You can also add signals from different levels of the design hierarchy to study the interactions between these modules and the testbench. The easiest way to add signals from a sub-module is to filter objects and then select the signals to add to the Waveform view.

Add signals from the instantiated `sine_gen_demo` module (DUT) and the `sinegen` module (U_SINEGEN).

1. In the Scopes window, select and expand the `Testbench`, then select and expand `DUT`.

Simulation objects associated with the currently selected scope display in the Objects window.

By default, all types of simulation objects display in the Objects window. However, you can limit the types of objects displayed by selecting the object filters at the top of the Objects window.

Figure 20 shows the Objects window with the Input and Output port objects enabled, and the other object types are disabled. Move the cursor to hover over a button to see the tooltip for the object type.

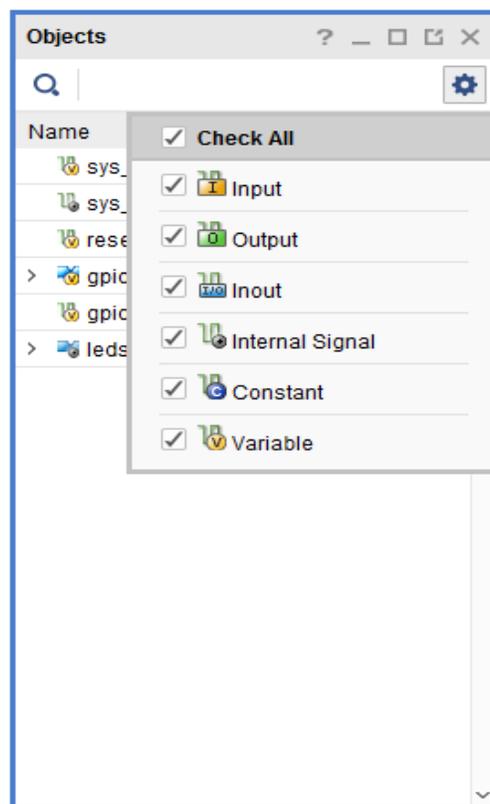


Figure 20: Object Filters

- Use the Objects window toolbar to enable and disable the different object types.
The types of objects that can be filtered in the Objects window include Input, Output, Inout ports, Internal Signals, Constants, and Variables.
- In the Scopes window, select the U_SINEGEN design unit.
- In the Waveform window, right-click in the empty space below the signal names, and use the **New Group** command to create three new groups called `Inputs`, `Outputs`, and `Internal Signals`.



TIP: If you create the group on top of, or containing, any of the current objects in the Waveform window, simply drag and drop the objects to separate them as needed.

- In the Objects window, select the Input filter to display the Input objects.
- Select the Input objects in the Objects window, and drag and drop them onto the Input group you created in the Waveform window.

Repeat steps 5 and 6 above to filter the Output objects and drag them onto the Output group, and filter the Internal Signals and drag them onto the Internal Signals group, as shown in [Figure 21](#).

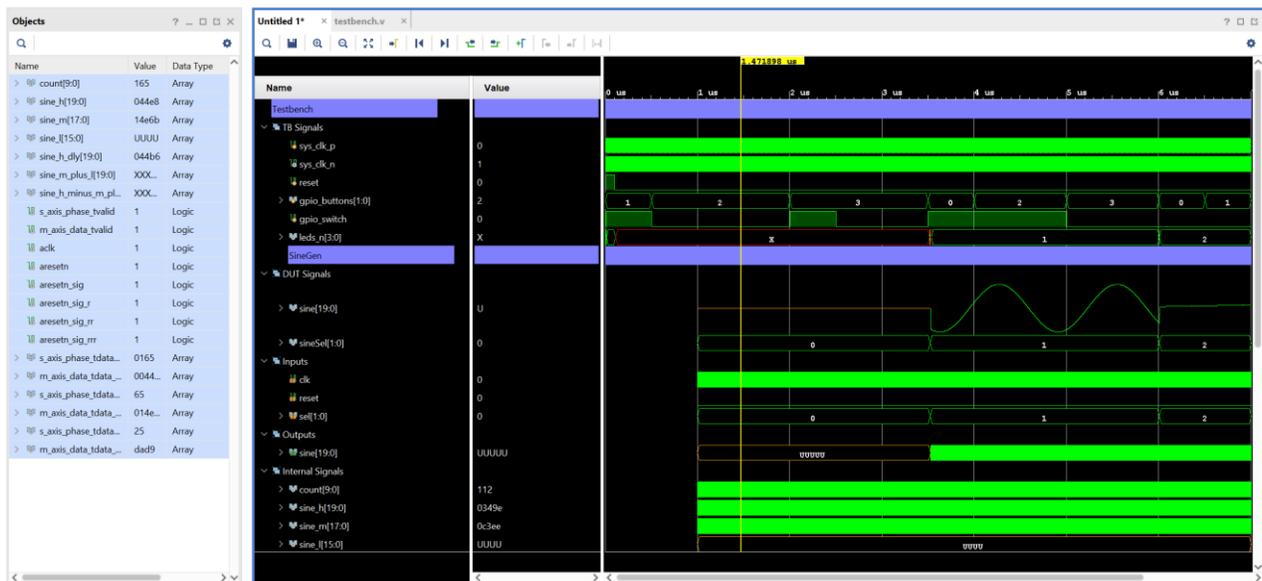


Figure 21: Configuring the Wave Window

Step 5: Changing Signal Properties

You can also change the properties of some of the signals shown in the Waveform window to better visualize the simulation results.

Viewing Hierarchical Signal Names

By default, the Vivado simulator adds signals to the waveform configuration using a short name with the hierarchy reference removed. For some signals, it is important to know to which module they belong.

1. In the Waveform window, hold **Ctrl** and click to select the `sine[19:0]` and `sineSel[1:0]` signals listed in the `DUT signals` group, under the `SineGen` divider.
2. Hold **Ctrl**, and click to select the `sine[19:0]` signals listed in the `Outputs` group, under the `SineGen` divider.
3. Right-click in the Waveform window to open the popup menu, and select the **Name > Long** command.

The displayed name changes to include the hierarchical path of the signal. You can now see that the `sine[19:0]` signals under the `DUT Signals` group refers to different objects in the design hierarchy than the `sine[19:0]` signals listed under the `Outputs` group.

See [Figure 22](#).

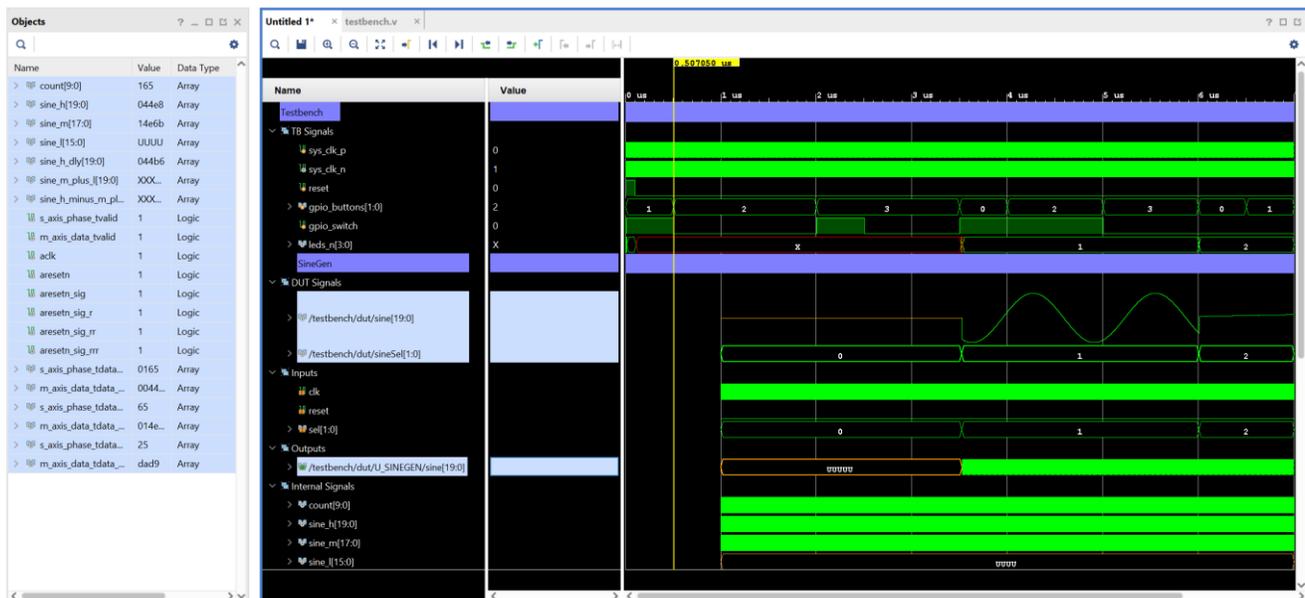


Figure 22: Long Signal Names

Viewing Signal Values

You can better understand some signal values if they display in a different radix format than the default, for instance, binary values instead of hexadecimal values. The default radix is Hexadecimal unless you override the radix for a specific object.

Supported radix values are Binary, Hexadecimal, Octal, ASCII, Signed and Unsigned decimal. You can set any of the above values as Default using Default Radix option.

1. In the Waveform window, select the following signals:

s_axis_phase_tdata_sine_high, s_axis_phase_tdata_sine_mid and s_axis_phase_tdata_sine_low.

2. Right-click to open the popup menu, and select **Radix > Binary**.

The values on these signals now display using the specified radix.

Step 6: Saving the Waveform Configuration

You can customize the look and feel of the Waveform window, and then save the Waveform configuration to reuse in future simulation runs. The Waveform configuration file defines the displayed signals, and the display characteristics of those signals.

1. In the Waveform window, click the **Settings**  button on the title bar menu.

The Waveform Options dialog box opens to the **General** tab.

2. Ensure the **Default Radix** is set to **Hexadecimal**.

This defines the default number format for all signals in the Waveform window. The radix can also be set for individual objects in the Waveform window to override the default.

3. Select the **Draw Waveform Shadow**, as shown in [Figure 23](#), to enable or disable the shading under the signal waveform.

By default, a waveform is shaded under the high transitions to make it easier to recognize the transitions and states in the Waveform window.

You can also enable or disable signal indices, so that each signal or group of signals is identified with an index number in the Waveform window.

4. Check or uncheck the **Show signal indices** check box to enable or disable the signal list numbering.

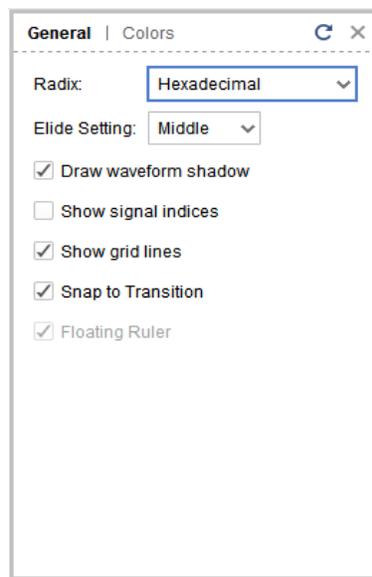


Figure 23: Waveform Options - General View

5. Check or uncheck the **Show grid lines** check box to enable or disable the grid lines in waveform window.
6. Check the **Snap to Transition** check box to snap the cursor to transition edges.
7. In the Waveform Options dialog box, select the **Colors** view.

Examine the Waveform Color Options dialog box. You can configure the coloring for elements of the Waveform window to customize the look and feel. You can specify custom colors to display waveforms of certain values, so you can quickly identify signals in an unknown state, or an uninitialized state.

The Waveform window configures with your preferences. You can save the current waveform configuration so it is available for use in future Vivado simulation sessions.

By default, the Vivado simulator saves the current waveform configuration setting as `testbench_behav.wcfg`.

8. In the Waveform window sidebar menu, select the **Save Wave Configuration** button .
9. Save the Wave Configuration into the project folder with the filename `tutorial_1.wcfg`.
10. Click **Yes**. The file is added to the project simulation fileset, `sim_1`, for archive purposes.



TIP: You can also load a previously saved waveform configuration file using the **File > Simulation Waveform > Open Configuration** command.

Working with Multiple Waveform Configurations

You can also have multiple Waveform windows, and waveform configuration files open at one time. This is useful when the number of signals you want to display exceeds the ability to display them in a single window. Depending on the resolution of the screen, a single Waveform window might not display all the signals of interest at the same time. You can open multiple Waveform windows, each with their own set of signals and signal properties, and copy and paste between them.

1. To add a new Waveform window, select **File > Simulation Waveform > New Configuration**.

An untitled Waveform window opens with a default name. You can add signals, define groups, add dividers, set properties and colors that are unique to this Waveform window.

2. Select signal groups in the first Waveform window by pressing and holding the **Ctrl** key, and selecting the following groups: `Inputs`, `Outputs`, and `Internal Signals`.
3. Right-click to open the popup menu, and select **Copy**, or use the shortcut **Ctrl+C** on the selected groups to copy them from the current Waveform window.
4. Select the new Waveform window to make it active.
5. Right-click in the Waveform window and select **Paste**, or use the shortcut **Ctrl+V** to paste the signal groups into the prior Waveform window.
6. Select **File > Simulation Waveform > Save Configuration** or click the **Save Wave Configuration** button, and save the waveform configuration to a file called `tutorial_2.wcfg`.

7. When prompted to add the waveform configuration to the project, select **No**.
8. Click the  icon to close the new Waveform window.

Step 7: Re-Simulating the Design

With the various signals, signal groups, dividers, and attributes you have added to the Waveform window, you are now ready to simulate the design again.

1. Click the **Restart** button  to reset the circuit to its initial state.
2. Click the **Run All** button .

The simulation runs for about 7005 ns. If you do not restart the simulator prior to executing the Run All command, the simulator runs continuously until interrupted.

3. After the simulation is complete, click the **Zoom Fit** button  to see the whole simulation timeline in the Waveform window. [Figure 24](#) shows the current simulation results.

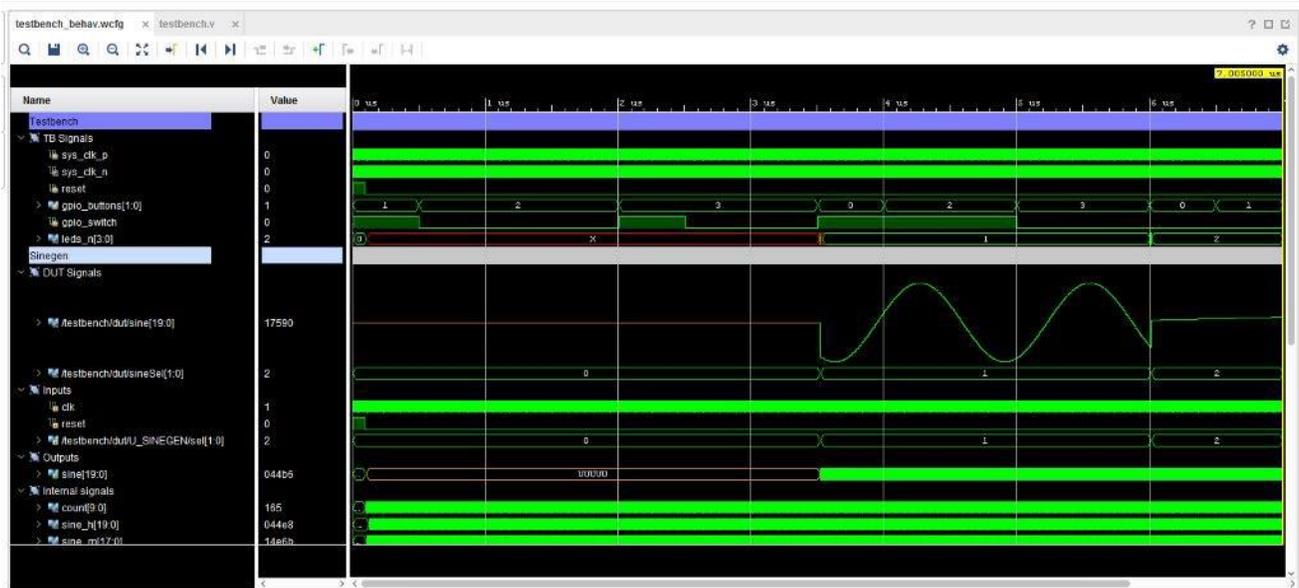


Figure 24: Simulation Waveform at Time 7005 ns

Step 8: Using Cursors, Markers, and Measuring Time

The Finite State Machine (U_FSM) module used in the top-level of the design generates three different sine-wave select signals for specific outputs of the `SineGen` block. You can identify these different wave selections better using Markers to highlight them.

1. In the Waveform window select the `/testbench/dut/sineSel[1:0]` signal, as shown in [Figure 25](#).

2. In the waveform sidebar menu, click the **Go to Time 0** button .

The current marker moves to the start of the simulation run.

3. Enable the **Snap to Transition** check box in the General tab of settings window to snap the cursor to transition edges.

4. From the waveform toolbar menu, click the **Next Transition** button .

The current marker moves to the first value change of the selected `sineSel[1:0]` signal, at 3.5225 microseconds.

5. Click the **Add Marker** button .

6. Search for all transitions on the `sineSel` signal, and add markers at each one.

With markers identifying the transitions on `sineSel`, the Waveform window should look similar to [Figure 25](#). As previously observed, the low frequency signals are incorrect when the `sinSel` signal value is 0.

You can also use the main Waveform window cursor to navigate to different simulation times, or locate value changes. In the next steps, you use this cursor to zoom into the Waveform window when the `sineSel` is 0 to review the status of the output signal, `sine[19:0]`, and identify where the incorrect behavior initiates. You also use the cursor to measure the period of low frequency wave control.

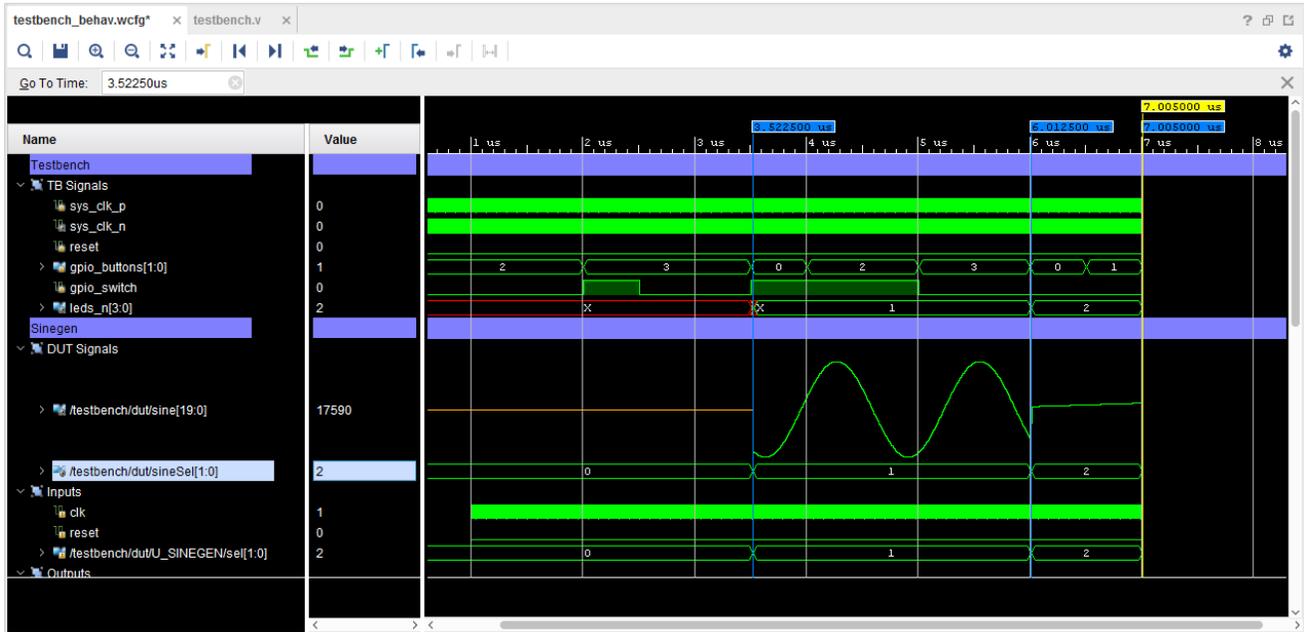


Figure 25: Using Markers



TIP: By default, the Waveform window displays the time unit in microseconds. However, you can use whichever measurement you prefer while running or changing current simulation time, and the Waveform window adjusts accordingly.

7. In the Waveform window, click the **Go to Time 0** button, then click the Zoom in button repeatedly to zoom into the beginning of the simulation run.
8. Continue to zoom in the Waveform window as needed, until you can see the `reset` signal asserted low, and you can see the waveform of the clock signals, `sys_clk_p` and `sys_clk_n`, as seen in [Figure 26](#).

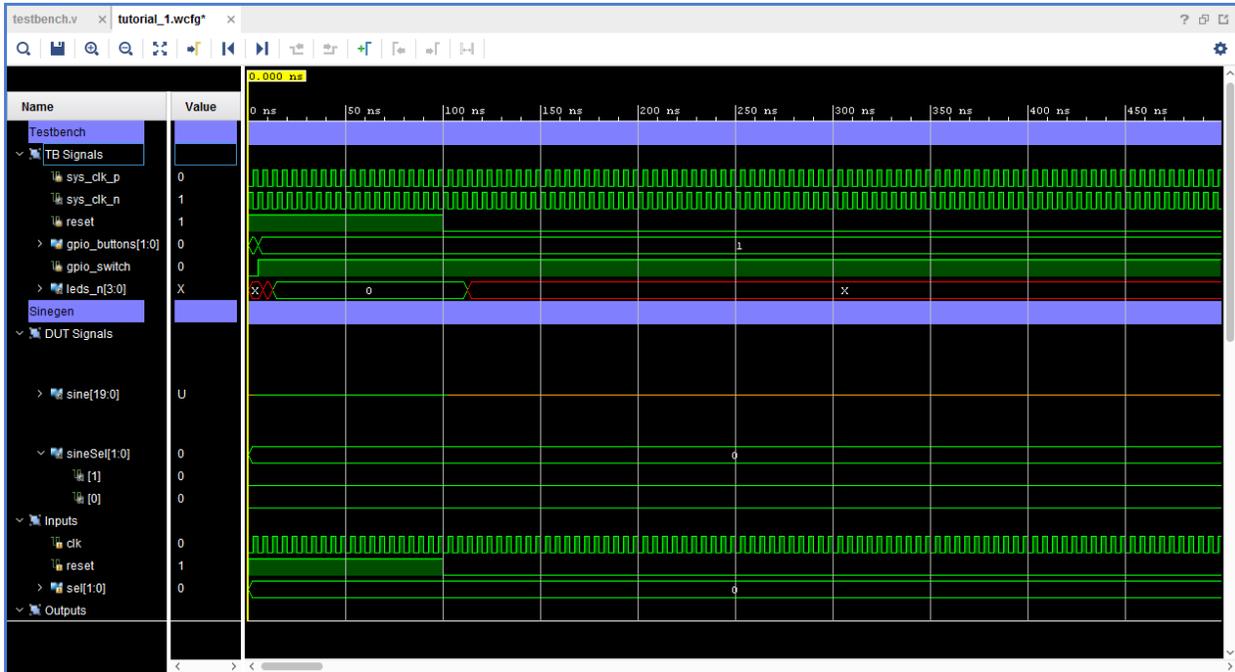


Figure 26: Viewing Reset and Clock Signals

The Waveform window zooms in or out around the area centered on the cursor.

- Place the main Waveform window cursor on the area by clicking at a specific time or point in the waveform.

You can also click on the main cursor, and drag it to the desired time.

- Because 0 is the initial or default FSM output, move the cursor to the first posedge of `sys_clk_p` after reset is asserted low, at time 102.5 ns, as seen in [Figure 27](#).

You can use the Waveform window to measure time between two points on the timeline.

- Place a marker at the time of interest, 102.5 ns, by clicking the **Add Marker** button .
- Click to select the marker.

The **Floating Ruler** option that is available in the **General** tab of waveform **Settings** displays a ruler at the bottom of the Waveform window useful for measuring time between two points. Use the floating ruler to measure the sineSel control signal period, and the corresponding `output_sine[19:0]` values during this time frame.

When you select the marker, a floating ruler opens at the bottom of the Waveform window, with time 0 on the ruler positioned at the selected marker. As you move the cursor along the timeline, the ruler measures the time difference between the cursor and the marker.



TIP: Enable the **Floating Ruler** checkbox from the **General** tab of **Waveform Settings**, if the ruler does not appear when you select the marker.

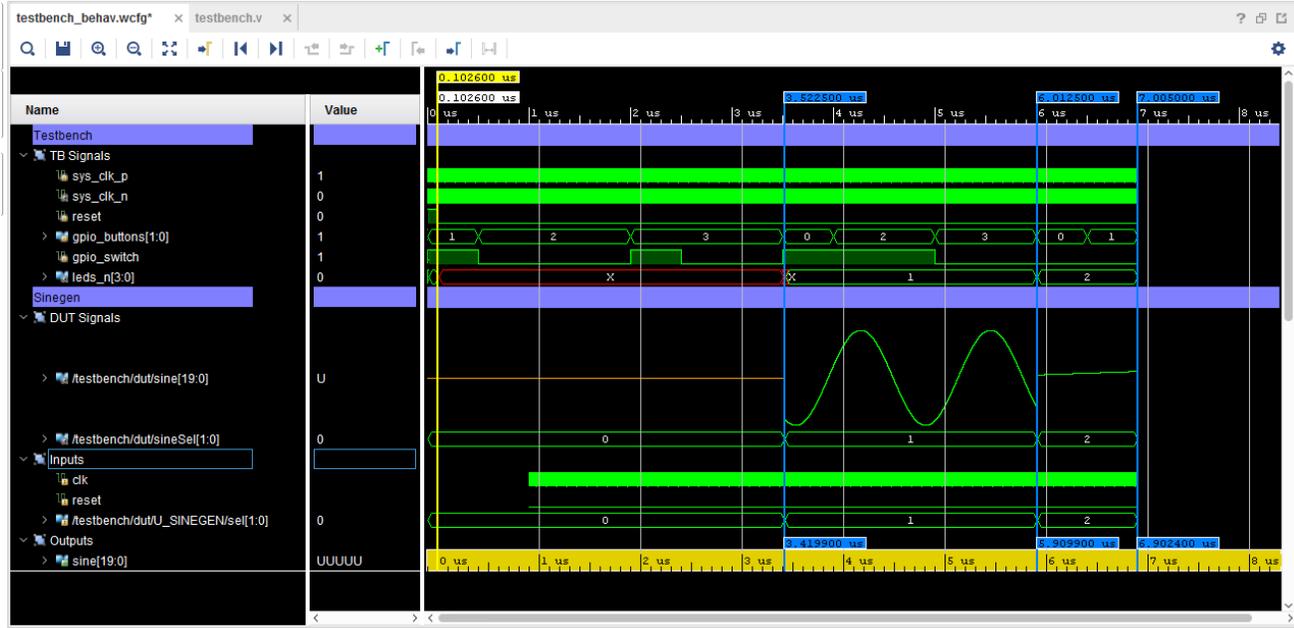


Figure 27: Measuring Time in the Waveform

You can move the cursor along the timeline in a number of ways. You can scroll the horizontal scroll bar at the bottom of the Waveform window. You can zoom out, or zoom fit to view more of the time line, reposition the cursor as needed, and then zoom in for greater detail.

13. Select `sineSel` from the list of signals in the Waveform window and use the **Next Transition** command to move to the specific transition of interest.

As shown in [Figure 27](#), the ruler measures a time period of 3.420 ns as the period that FSM selected the low frequency output.

Step 9: Debugging with Breakpoints

You have examined the design using cursors, markers, and multiple Waveform windows. Now you use Vivado simulator debugging features, such as breakpoints, and line stepping, to debug the design and identify the cause of the incorrect output.

1. First, open the tutorial design testbench to learn how the simulator generates each design input.
2. *Open the `testbench.v` file by double-clicking the file in the Sources window, if it is not already open.*

The source file opens in the Vivado IDE Text Editor, as shown in [Figure 28](#).

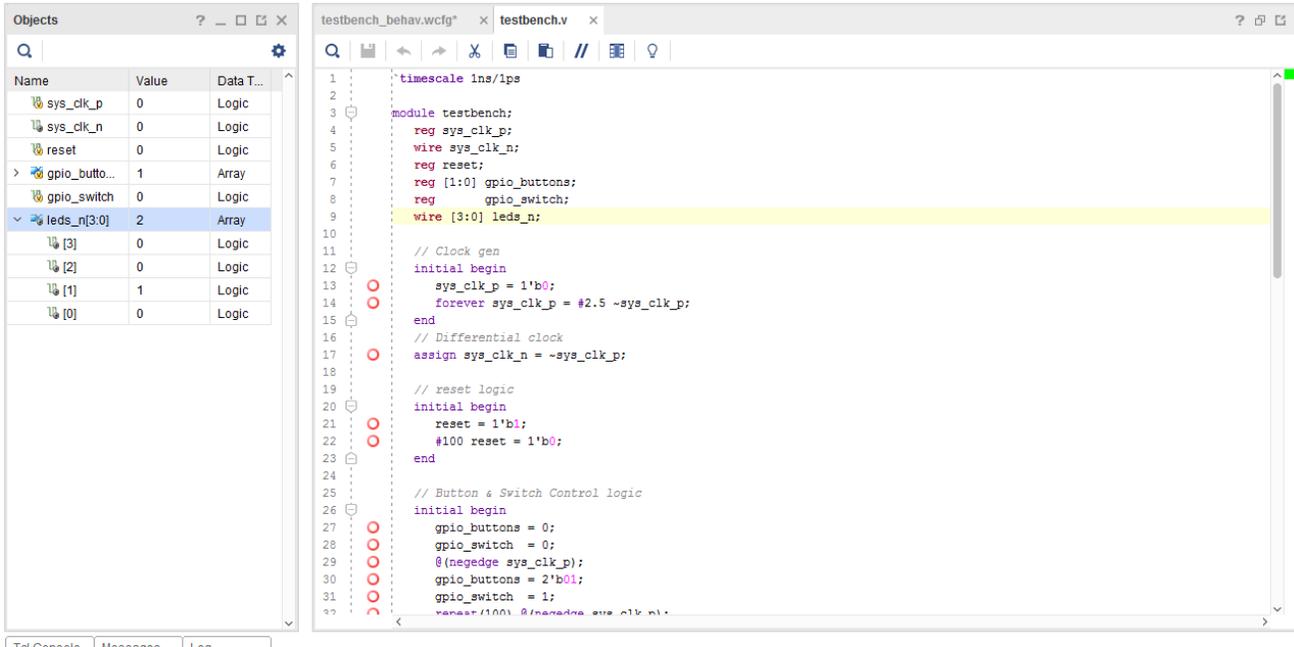


Figure 28: Integrated Text Editor

Note: You can also use **File > Text Editor > Open File** from the main menu, or **Open File** from the popup menu in the Sources window. You can also select an appropriate design object in the Scopes window or Objects window, right-click and select **Go to Source Code**.

Using Breakpoints

A breakpoint is a user-determined stopping point in the source code used for debugging the design. When simulating a design with set breakpoints, simulation of the design stops at each breakpoint to verify the design behavior. After the simulation stops, an indicator shows in the text editor next to the line in the source file where the breakpoint was set, so you can compare the Wave window results with a particular event in the HDL source.

You use breakpoints to debug the error with the low frequency signal output that you previously observed. The erroneous `sine[19:0]` output is driven from the `sineGen` VHDL block. Start your debugging with this block.

1. Select the **U_SINEGEN** scope in the Scopes window to list the objects of that scope in the Objects window.
2. In the Objects window, right-click `sine[19:0]` and use **Go to Source Code** to open the `sinegen.vhd` source file in the Text Editor.



TIP: If you do not see the `sine[19:0]` signal in the Objects window, make sure that the filters at the top of the Objects window are set properly to include Output objects.

Looking through the HDL code, the `clk`, `reset`, and `sel` inputs are correct as expected. Set your first breakpoint after the `reset` asserts low at line 137.

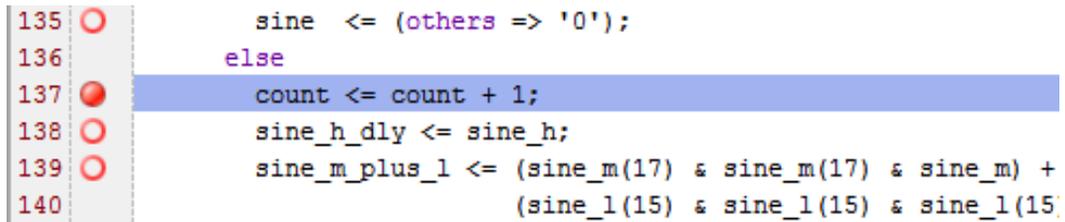
3. Scroll to line 137 in the file.

Add a breakpoint at line 137 in `sinegen.vhd`. Note that the breakpoint can be set only on the executable lines. Vivado simulator marks the executable lines with an empty red circle , on the left hand margin of the Text Editor, beside the line numbers.

Setting a breakpoint causes the simulator to stop at that point, every time the simulator processes that code, or every time the counter is incremented by one.

4. Click the red circle  in the left margin, to set a breakpoint, as shown in [Figure 29](#).

Observe that the empty circle becomes a red dot  to indicate that a breakpoint is set on this line. Clicking on the red dot  removes the breakpoint and reverts it to the empty circle .



```

135 ○ sine <= (others => '0');
136 ○ else
137 ● count <= count + 1;
138 ○ sine_h_dly <= sine_h;
139 ○ sine_m_plus_1 <= (sine_m(17) & sine_m(17) & sine_m) +
140 ○ (sine_l(15) & sine_l(15) & sine_l(15);

```

Figure 29: Setting a Breakpoint

Note: To delete all breakpoints in the file, right-click on one of the breakpoints and select **Delete All Breakpoints**.

Debugging in the Vivado simulator, with breakpoints and line stepping, works best when you can view the Tcl Console, the Waveform window, and the HDL source file at the same time, as shown in [Figure 30](#).

5. Resize the windows, and use the window Float command  or the New Vertical Group command to arrange the various windows so that you can see them all.
6. Click the **Restart** button  to restart the simulation from time 0.
7. Run the simulation by clicking the **Run All** button .

The simulation runs to time 102.5 ns, or near the start of first counting, and stops at the breakpoint at line 137. The focus within the Vivado IDE changes to the Text Editor, where it shows the breakpoint indicator  and highlights the line.

A message also displays in the Tcl console to indicate that the simulator has stopped at a specific time, displayed in picoseconds, indicating the line of source code last executed by the simulator.

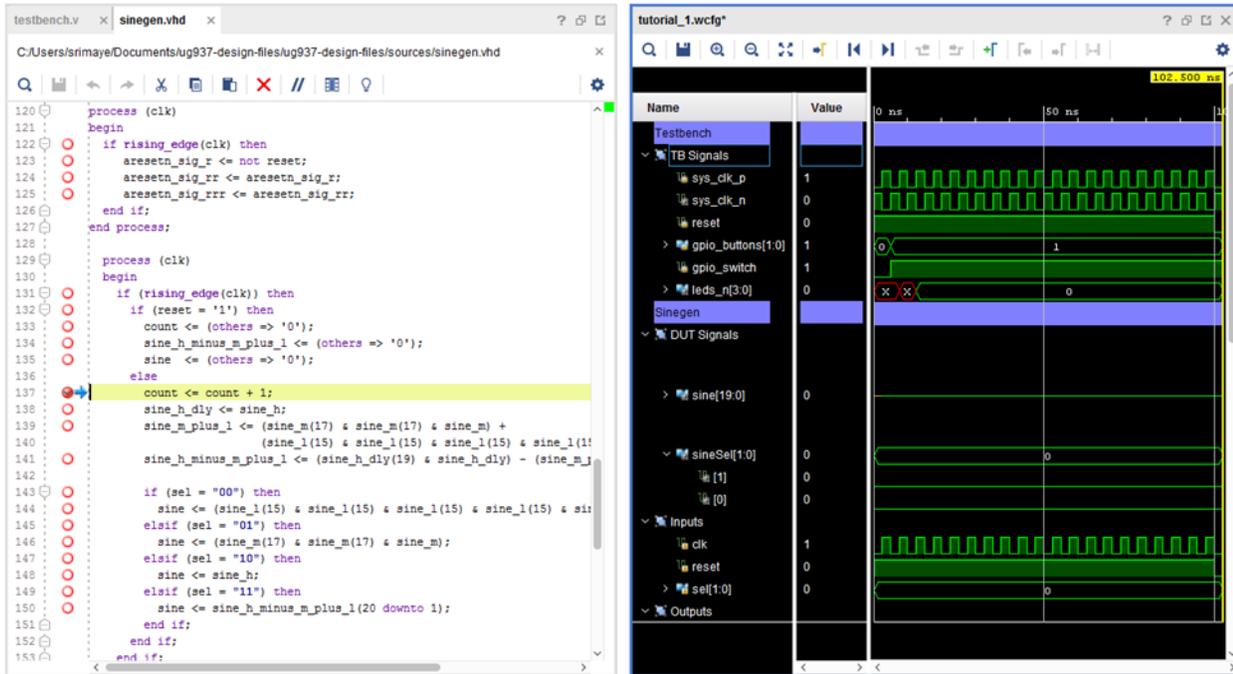


Figure 30: Arrange Windows for Debugging



TIP: When you have arranged windows to perform a specific task, such as simulation debug in this case, you can save the view layout to reuse it when needed. Use the **Layout > Save Layout As** command from the main menu to save view layouts. See the *Vivado Design Suite User Guide: Using the Vivado IDE (UG893)* for more information on arranging windows and using view layouts.

8. Continue the simulation by clicking the **Run All** button ▶.

The simulation stops again at the breakpoint. Take a moment to examine the values in the Waveform window. Notice that the `sine[19:0]` signals in the Outputs group are uninitialized, as are the `sine_l[15:0]` signals in the Internal signals group.

9. In the Text Editor, add another breakpoint at line 144 of the `sinegen.vhd` source file.

This line of code runs when the value of `sel` is 0. This code assigns, with bit extension, the low frequency signal, `sine_l`, to the output, `sine`.

10. In the Waveform window, select `sine_l[15:0]` in the Internal Signals group, and holding **Ctrl**, select `sine[19:0]` in the Outputs group.

These selected signals are highlighted in the Waveform window, making them easier for you to monitor.

11. Run the simulation by clicking the **Run All** button ▶.

Once again, the simulation stops at the breakpoint, this time at line 144.

Stepping Through Source Code

Another useful Vivado simulator debug tool is the *Line Stepping* feature. With line stepping, you can run the simulator one-simulation unit (line, process, task) at a time. This is helpful if you are interested in learning how each line of your source code affects the results in simulation.

Step through the source code line-by-line and examine how the low frequency wave is selected, and whether the DDS compiler output is correct.

1. On the Vivado simulator toolbar menu, click the **Step** button .

The simulation steps forward to the next executable line, in this case in another source file. The `fsm.vdh` file is opened in the Text Editor. You may need to relocate the Text Editor to let you see all the windows as previously arranged.

Note: You can also type the `step` command at the Tcl prompt.

2. Continue to Step through the design, until the code returns to line 144 of `sinegen.vhd`.

You have stepped through one complete cycle of the circuit. Notice in the Waveform window that while `sel` is 0, signal `sine_1` is assigned as a low frequency sine wave to the output `sine`. Also, notice that `sine_1` remains uninitialized.

3. For debug purposes, initialize the value of `sine_1` by entering the following `add_force` command in the Tcl console:

```
add_force /testbench/dut/U_SINEGEN/sine_1 0110011011001010
```

This command forces the value of `sine_1` into a specific known condition, and can provide a repeating set of values to exercise the signal more vigorously, if needed. Refer to the *Vivado Design Suite User Guide: Logic Simulation (UG900)* for more information on using `add_force`.

4. Continue the simulation by clicking the **Run All** button  a few more times.

In the Waveform window, notice that the value of `sine_1[15:0]` is now set to the value specified by the `add_force` command, and this value is assigned to the output signal `sine[19:0]` since the value of `sel` is still 0.

Trace the `sine_1` signal in the HDL source files, and identify the input for `sine_1`.

5. In the Text Editor, right-click to open the popup menu, and select the **Find in files** option to search for `sine_1`.
6. Select the **Match whole word** and **Enabled design sources** checkboxes, as shown in [Figure 31](#), and click **Find**.

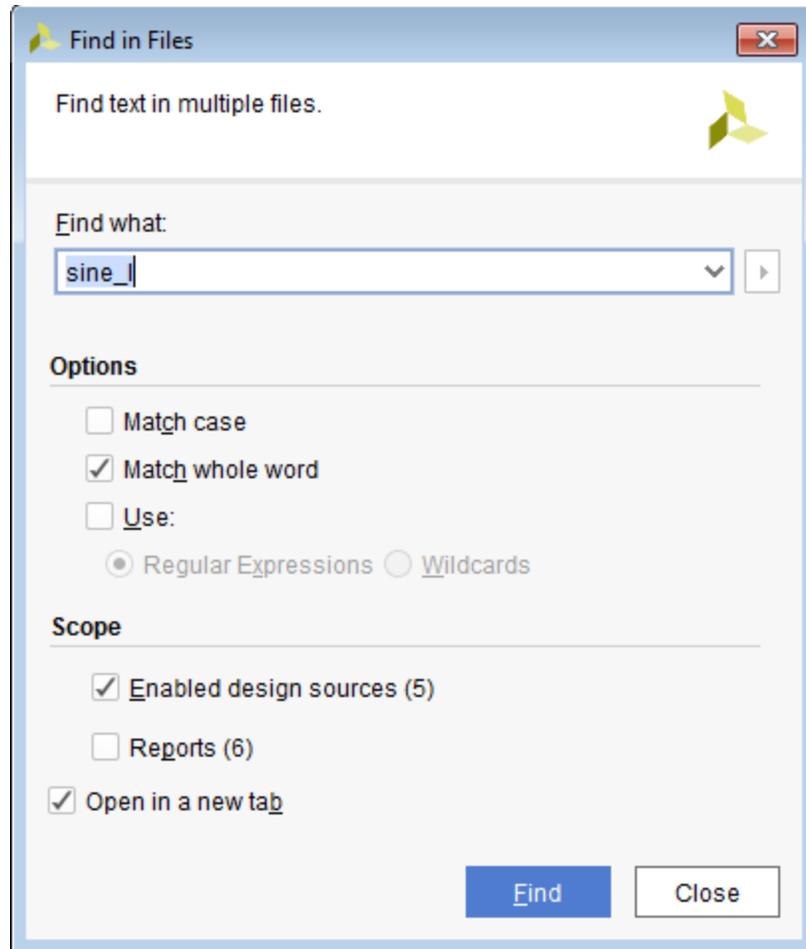


Figure 31: Find in Files

The Find in Files results display at the bottom of the Vivado IDE, with all occurrences of `sine_1` found in the `sinegen.vhd` file.

- Expand the Find in Files results to view the results in the `sinegen.vhd` file.

The second result, on line 111, identifies a problem with the design. At line 111 in the `sinegen.vhd` file, the `m_axis_data_tdata_sine_low` signal is assigned to `sine_1`. Since line 111 is commented out, the `sine_1` signal is not connected to the low frequency DDS compiler output, or any other input.

- Uncomment line 111 in the `sinegen.vhd` file, and click the **Save File** button.
- In the Tcl Console, remove the force on `sine_1`: `remove_forces -all`

Step 10: Relaunch Simulation

By using breakpoints and line stepping, you identified the problem with the low frequency output of the design and corrected it.

Since you modified the source files associated with the design, you must recompile the HDL source and build new simulation snapshot. Do not just restart the simulation at time 0 in this case but rebuild the simulation from scratch.

1. In `sinegen.vhd`, select one of the breakpoints, right-click and select **Delete All Breakpoints**.
2. Click the **Relaunch** button  on the main toolbar menu.

Note: If prompted to save the Wave Config file, click **yes**.

The Vivado simulator recompiles the source files with `xelab`, and re-creates the simulation snapshot. Now you are ready to simulate with the corrected design files. The relaunch button  will be active only after one successful run of Vivado Simulator using `launch_simulation`. If you run the simulation in a Batch/Scripted mode, the relaunch button would be greyed out.

3. Click the **Run All** button  (Figure 32) to run the simulation.

Observe the `sine[19:0]`, the analog signal in the waveform configuration. The low frequency sine wave looks as expected. The Tcl console results are:

```
[@3518000] LEDS_n = 0100
[@3523000] LEDS_n = 0001
[@3523000] LEDS_n = 0001
[@6008000] LEDS_n = 0101
[@6013000] LEDS_n = 0010
[@6013000] LEDS_n = 0010
$finish called at time : 7005 ns : File "ug937/sim/testbench.v" Line 63
```

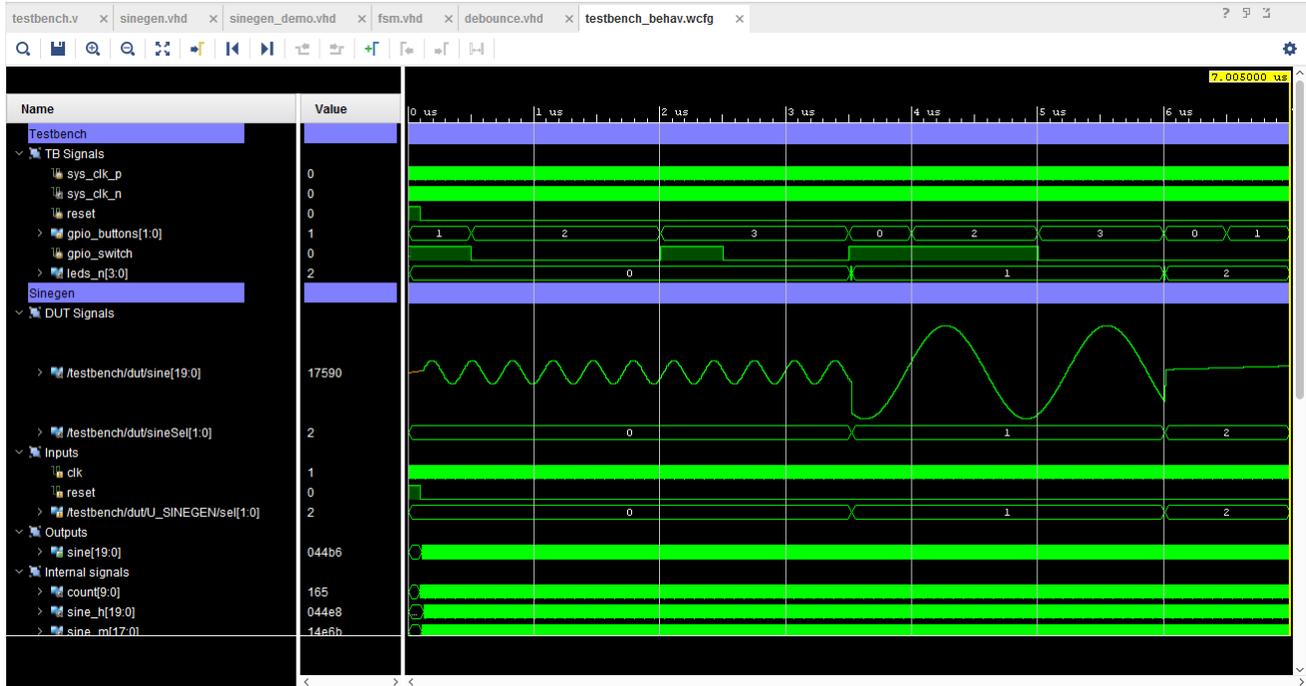


Figure 32: Corrected Low Frequency Output

Conclusion

After reviewing the simulation results, you may close the simulation, and close the project. This completes Lab #2. Up to this point in the tutorial, between Lab #1 and Lab #2, you have:

- Run the Vivado simulator using the Project Mode flow in Vivado IDE
- Created a project, added source files, and added IP
- Added a simulation-only file (`testbench.v`)
- Set simulation properties and launched behavioral simulation
- Added signals to the Waveform window
- Configured and saved the Waveform Configuration file
- Debugged the design bug using breakpoints and line stepping.
- Corrected an error, re-launched simulation, and verified the design

In Lab # 3, you will examine the Vivado simulator batch mode.

Lab 3: Running Simulation in Batch Mode

Introduction

You can use the Vivado simulator *Non-Project Mode* flow to simulate your design without setting up a project in Vivado Integrated Design Environment (IDE).

In this flow, you:

- Prepare the simulation project manually by creating a Vivado simulator project script.
 - Create a simulation snapshot file using the Vivado simulator `xelab` utility.
 - Start the Vivado simulator GUI by running the `xsim` command with the resulting snapshot.
-

Step 1: Preparing the Simulation

The Vivado simulator Non-Project Mode flow lets you simulate your design without setting up a project in the Vivado IDE.

You can compile the HDL files in a design, and create a simulation snapshot by either:

- Creating a Vivado simulator project script, specifying all HDL files to be compiled, and using the `xelab` command to create a simulation snapshot, or
- Using specific Vivado simulator parser commands, `xvlog` and `xvhdl`, to parse individual source files and write the parsed files into an HDL library on disk, and then using `xelab` to create a simulation snapshot from the parsed files

Creating the Vivado Simulator Project File

A Vivado simulator project script specifies design source files and libraries to parse and compile for simulation. This method is useful to create a simulation project script that can be run repeatedly over the course of project development.

The format for a Vivado simulator project script (prj file) is as follows:

```
verilog | vhdl | sv <library_name> {<file_name>.v|.vhd}
```

Where:

- `verilog | vhdl | sv` specifies whether the design source is a Verilog, VHDL, or SV file.
- `<library_name>`: Specifies the library into which you may compile the source file. If unspecified, the default library for compilation is `work`.
- `<file_name>.v|.vhd`: Specifies the name of the design source file to compile.



IMPORTANT: While you can specify one or more Verilog source files on a single command line, you can only specify one VHDL source on a single command line.

In this step, you build a Vivado simulator project script by editing an existing project script to add missing source files. The command lines for the project script should be constructed using the syntax described above.

1. Browse to the `<Extract_Dir>/scripts` folder.
2. Open the `simulate_xsim.prj` project script with a text editor.
3. Add the following commands to the project script:

```
vhdl xil_defaultlib "../sources/sinegen.vhd"  
vhdl xil_defaultlib "../sources/debounce.vhd"  
vhdl xil_defaultlib "../sources/fsm.vhd"  
vhdl xil_defaultlib "../sources/sinegen_demo.vhd"  
verilog xil_defaultlib "../sim/testbench.v"
```

4. Save and close the file.

You do not need to list the sources based on any specific order of dependency. The `xelab` command resolves the order of dependencies, and automatically processes the files accordingly.



TIP: For your reference, a completed version of the tutorial files can be found in the `ug937-design-files/completed` folder.

Manually Parsing Design Files

As an alternative to creating a Vivado simulator project script, you can compile individual design source files directly from the command line using the `xvlog` or `xvhdl` commands to parse the design sources and write them to an HDL library. You could use this method for simple simulation runs, or to define a shell script and makefile compilation flow.

Parse individual or multiple Verilog files using the `xvlog` command with the following syntax format:

```
xvlog [options] <verilog_file | list_of_files>
```

Parse individual VHDL files using the `xvhdl` command with the following syntax format:

```
xvhdl [options] <VHDL_file>
```

For a complete list of available `xvlog` and `xvhdl` command options, see the *Vivado Design Suite User Guide: Logic Simulation* ([UG900](#)). The `parse_standalone.bat` file in `<Extract_Dir>/scripts` or `<Extract_Dir>/completed` provide examples of running `xvlog` and `xvhdl` directly.

Step 2: Building the Simulation Snapshot

In this step, you use the `xelab` command on the project script you previously edited (`simulate_xsim.prj`) to elaborate, compile, and link all the sources for the design. The `xelab` utility creates a simulation snapshot that lets you to simulate the design in the Vivado simulator.

The typical `xelab` command syntax is:

```
xelab -prj <project_file> -s <simulation snapshot> <library>.<top_unit>
```

Where:

- `-prj <project_file>`: Specifies a Vivado simulation project script to use for input.
- `-s <simulation_snapshot>`: Specifies the name of the output simulation snapshot.
- `<library>.<top_unit>`: Specifies the library and top-level module of the design.

Running `xelab`

In this step, you use the `xelab` command with the project file completed in Step 1 to elaborate, compile, and link all the design sources to create the simulation snapshot. To run the `xelab` command, must open and configure a command window.

1. On Windows, open a Command Prompt window. On Linux, simply skip to the next step.
2. Change directory to the Xilinx installation area, and run `settings64.bat` as needed to setup the Xilinx tool paths for your computer:

```
cd install_path\Vivado\2019.x\settings64.bat
```

Note: The `settings64.bat` file configures the path on your computer to run the Vivado Design Suite.



TIP: When running the `xelab`, `xsc`, `xsim`, `xvhdl`, or `xvlog` commands in batch files or scripts, it may also be necessary to define the `XILINX_VIVADO` environment variable to point to the installation hierarchy of the Vivado Design Suite. To set the `XILINX_VIVADO` variable, you can add one of the following to your script or batch file:

On Windows -

```
set XILINX_VIVADO=<Vivado_install_area>/Vivado/2019.x
```

On Linux -

```
setenv XILINX_VIVADO <Vivado_install_area>/Vivado/2019.x
```

3. Change directory to the `<Extract_Dir>/scripts` folder.

The provided `xelab` batch file, `xelab_batch.bat`, is incomplete, and you must modify it using the `xelab` syntax as previously described to produce the correct simulation snapshot.

4. Edit the `xelab_batch.bat` file to add the following options:

- Specify the project file: `-prj simulate_xsim.prj`
- Specify the output simulation snapshot: `-s run_sineGen`
- Specify the library and top-level design unit: `xil_defaultlib.testbench`

For a complete list of available `xelab` command options, see the *Vivado Design Suite User Guide: Logic Simulation* ([UG900](#)).

5. Save and close the batch file.
6. In the command window, run the `xelab_batch.bat` file to compile and create the simulation snapshot.

```
xelab_batch.bat
```

7. Examine the `xelab` output as it is transcribed to the Command Prompt window.

Note: The `xelab` command also writes `xelab.log` file in the directory from which it was run. The log file contains all of the messages and results of the `xelab` command for you to review.



TIP: You can also use the `xelab` command after the `xvlog` and `xvhdl` commands have parsed the HDL design sources to read the specified simulation libraries. The `xelab` command would be the same as described here, except that it would not require the `-prj` option since there would be no simulation project file.

Step 3: Manually Simulating the Design

In this step, you launch the Vivado simulator GUI by running the `xsim` command with the simulation snapshot that you generated using the `xelab` command in Step 2. After you complete this step, you can use the Vivado simulator GUI to explore the design in more detail.

In the same command window that you used for Step #2, type the following command:

```
xsim run_sineGen -gui -wdb simulate_xsim.wdb -view xsim_waveConfig
```

Where:

- `run_sineGen -gui`: Specifies the simulation snapshot that you generated using `xelab`, and launches Vivado simulator in GUI mode.
- `-wdb`: Specifies the file name of the simulation waveform database file to output, or write, upon completion of the simulation run.
- `-view`: Opens the specified waveform configuration file within the Vivado simulator GUI.

Note: You can use the waveform configuration file specified above, or use the `tutorial_1.wcfg` file that you created in Lab #2 of this tutorial.

The Vivado Simulator GUI opens and loads the design (Figure 33). The simulator time remains at 0 ns until you specify a run time. Run the simulation and explore the design.

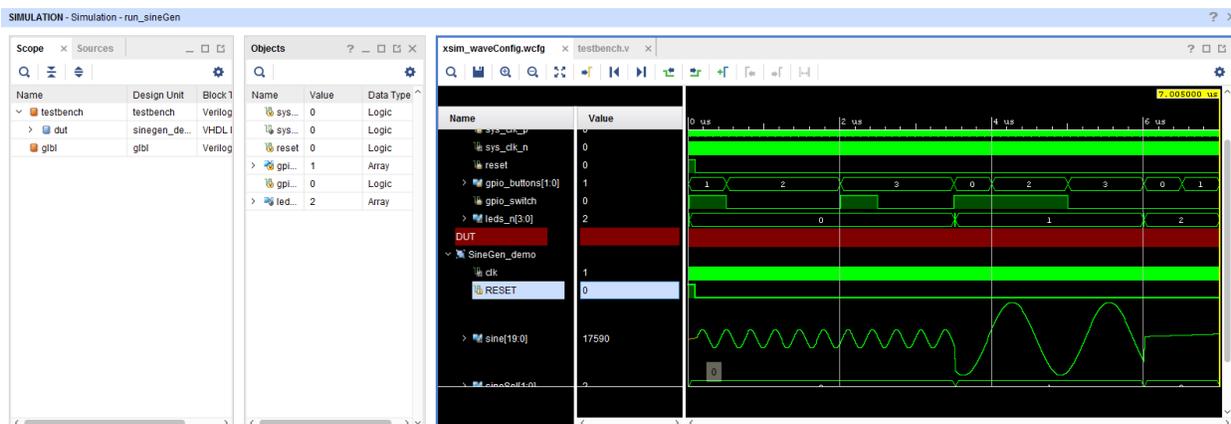


Figure 33: Run Xsim GUI

Conclusion

In this tutorial, you:

- Created a Vivado IDE project
- Downloaded source files and ran Vivado simulation
- Examined the simulation customization features
- Debugged and fixed a known issue within the source files
- Ran a Vivado simulation in batch mode using the Vivado simulation executable and switch options

Introduction

Vivado simulator now supports synthesizable as well as test bench/verification feature of System Verilog IEEE 1800-2012. In this chapter, you will go through a System Verilog example to learn about different debugging capabilities added in the Vivado simulator. You will use an IP example design provided with Vivado.

Creating an Example Design

You will now generate an AXI-VIP example design.

1. Open Vivado.
2. Create a project with the name `mySystemVerilog` by invoking the following command in Vivado Tcl console.

```
create_project mySystemVerilog ./mySystemVerilog
```

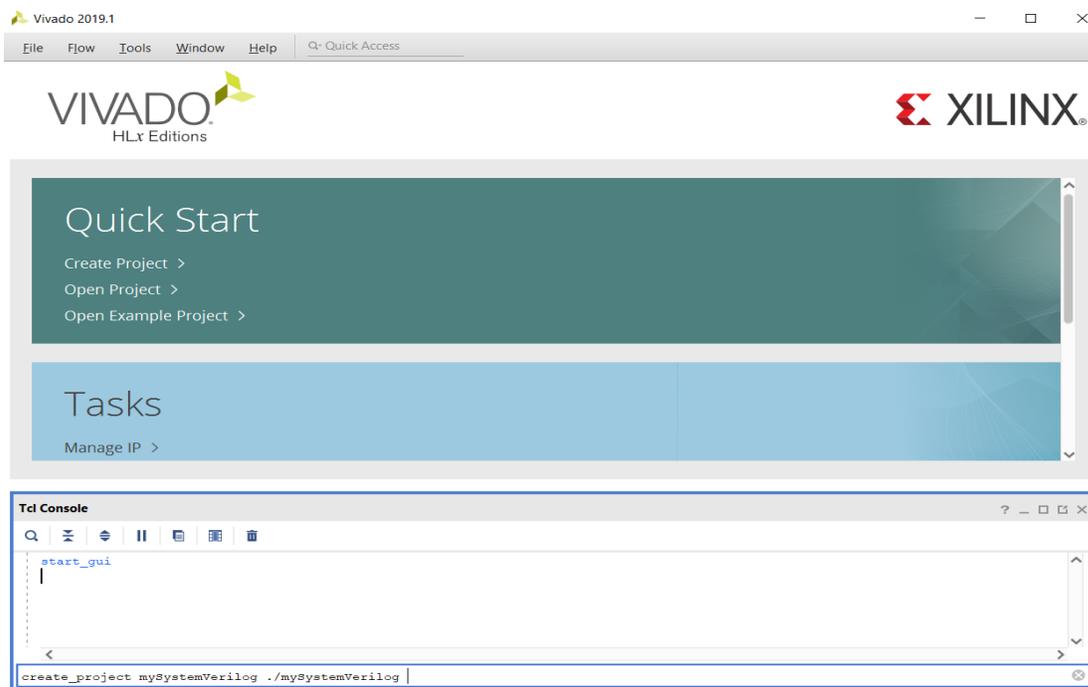


Figure 34: Vivado IDE Tcl Console

3. You will create an AXI-VIP example design that includes the following features:
 - a. Random Constraint

- b. Dynamic Types and Class
 - c. Virtual Interface
 - d. Assertion
 - e. Clocking Block
4. Invoke the following commands in Tcl console:
 - a. `create_ip -name axi_vip -vendor xilinx.com -library ip -version 1.1 -module_name axi_vip_0`
 - b. `open_example_project -force [get_ips axi_vip_0]`

Now you have created an example design for AXI-VIP with the name `axi_vip_0_ex`.

Launching Simulation

You have an example project ready. Next, you will run the behavioral simulation. By default, the simulation runs in a pre-compiled mode where the source code for static IP is not added in the project. Run the simulation in a non-precompiled mode for a better understanding of the feature. Invoke the following commands in Tcl console:

- `set_property sim.use_ip_compiled_libs 0 [current_project]`
- `launch_simulation`

This will run the simulation for 1000 ns.

Debugging Using Vivado Simulator

Vivado simulator supports System Verilog feature. In this exercise, you will explore the System Verilog feature using the following:

- Scope Window
- Object Window
- Tcl Console

Scope Window

System Verilog has a building block called interface. It differs in functionality when you compare it with the module.

1. On the Scope window, click **Expand All** button .
2. You can now view the IF and PC interface instances. The IF and PC interface instance icons are different than the module icons.

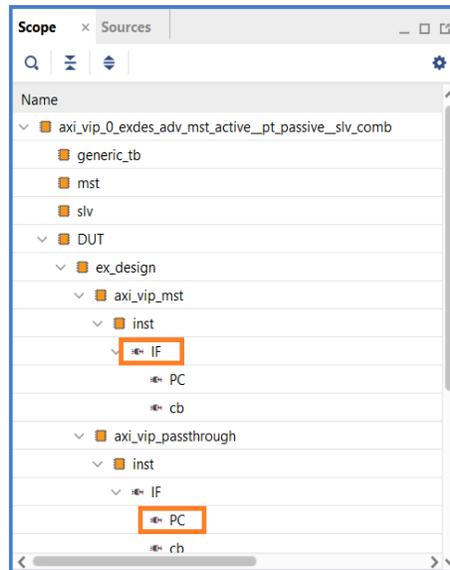


Figure 35: Scope Window

3. Right-click **IF** and select **Go to Source Code**. It will point to an interface definition.

Object Window

In System Verilog, all the net/variables are static type. They exist throughout the simulation. In System Verilog, dynamic type is a new type along with static type. Class, Queue, and Associative Array are some examples of dynamic type.

Unlike static type variables (int a; wire [7:0] b;), dynamic type variables do not have a fixed size throughout the simulation. Variables keep changing during run-time. Through Object window, you can view the value of a dynamic type variable during the simulation.

1. Click **Restart** button .
2. From the Scope window, select scope **generic_tb** present under the top module.



Figure 36: Scope Window

3. Maximize the **Objects** window. As the simulation is yet to start, observe the Queue and Class dynamic type in the Data Type column. The Value for Queue appears empty while for class it appears null.

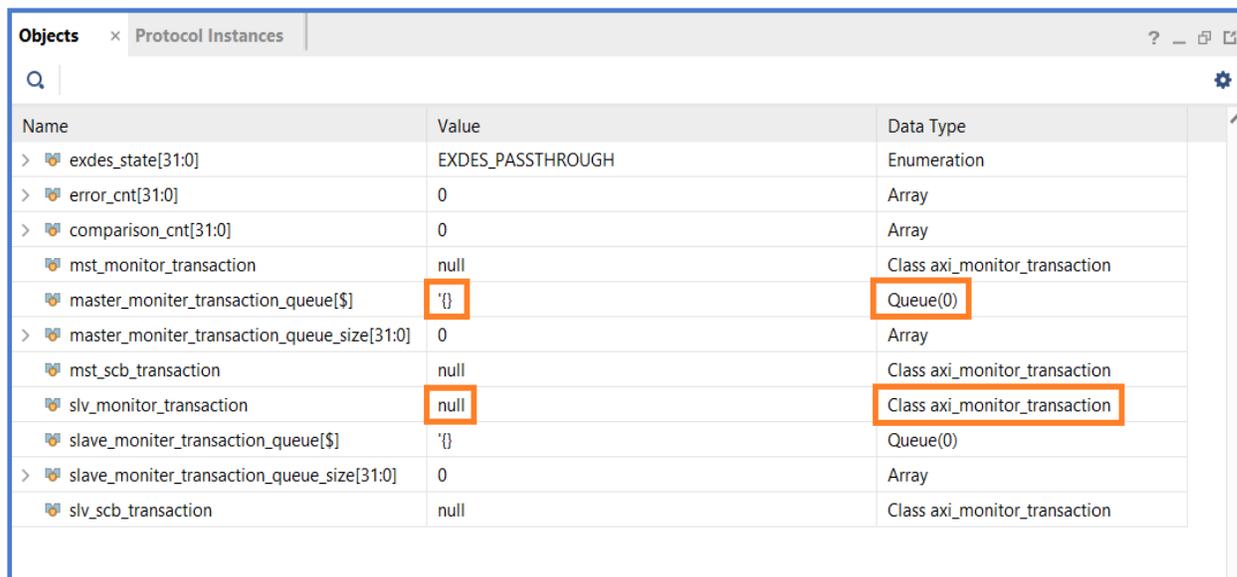


Figure 37: Objects Window

4. On Scope window, double-click **generic_tb** to see the text file.

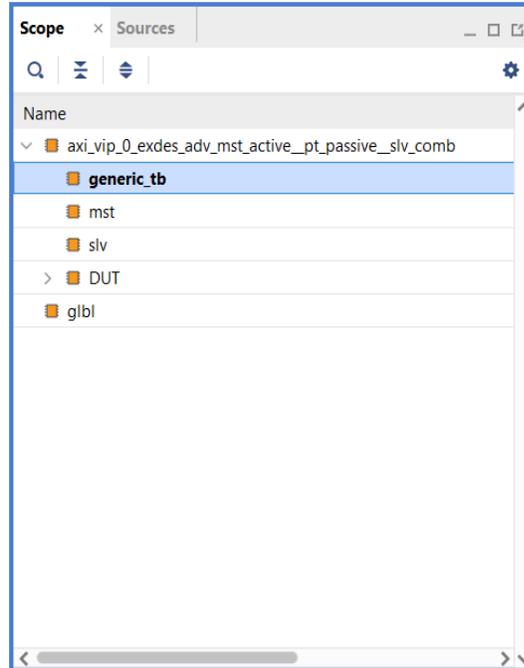


Figure 38: Scope Window

- On the text editor window, click the **Circle** on line number 64 to add a break point.

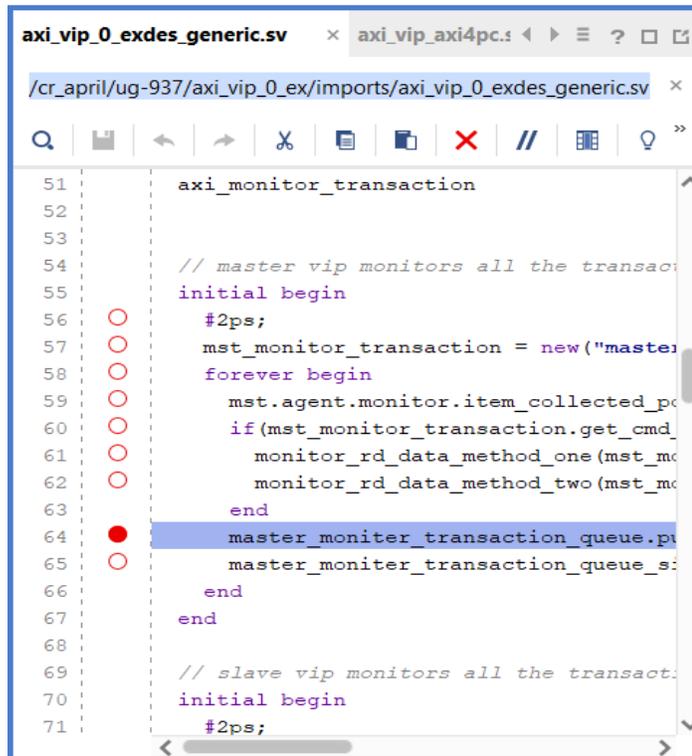


Figure 39: Text Editor

- Click **Run All** button, the simulation will stop at line number 64. On the Object window `master_monitor_transaction_queue` value appears empty.

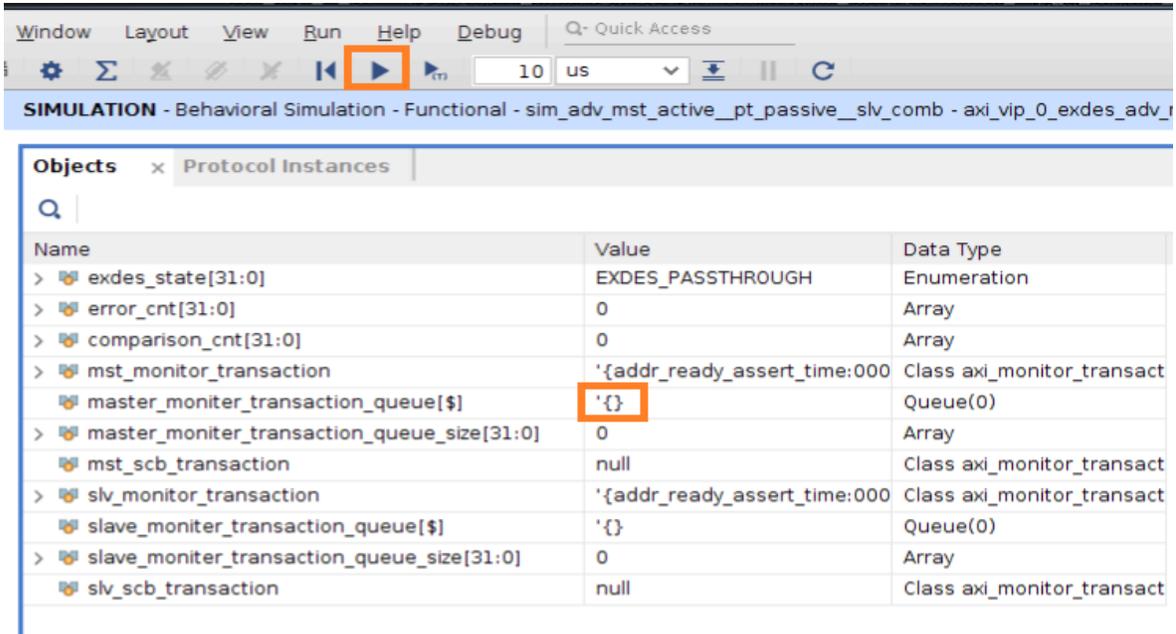


Figure 40: Objects Window

- On the Vivado simulator toolbar menu, click the **Step** button . It executes the current statement that is on line number 64 where the simulation is currently waiting. At this statement, you are pushing an element after the execution. Your queue will be populated with a single element.
- In the Object window, the `master_monitor_transaction_queue` value is populated. This way you can view the value of any dynamic type on the Object window.

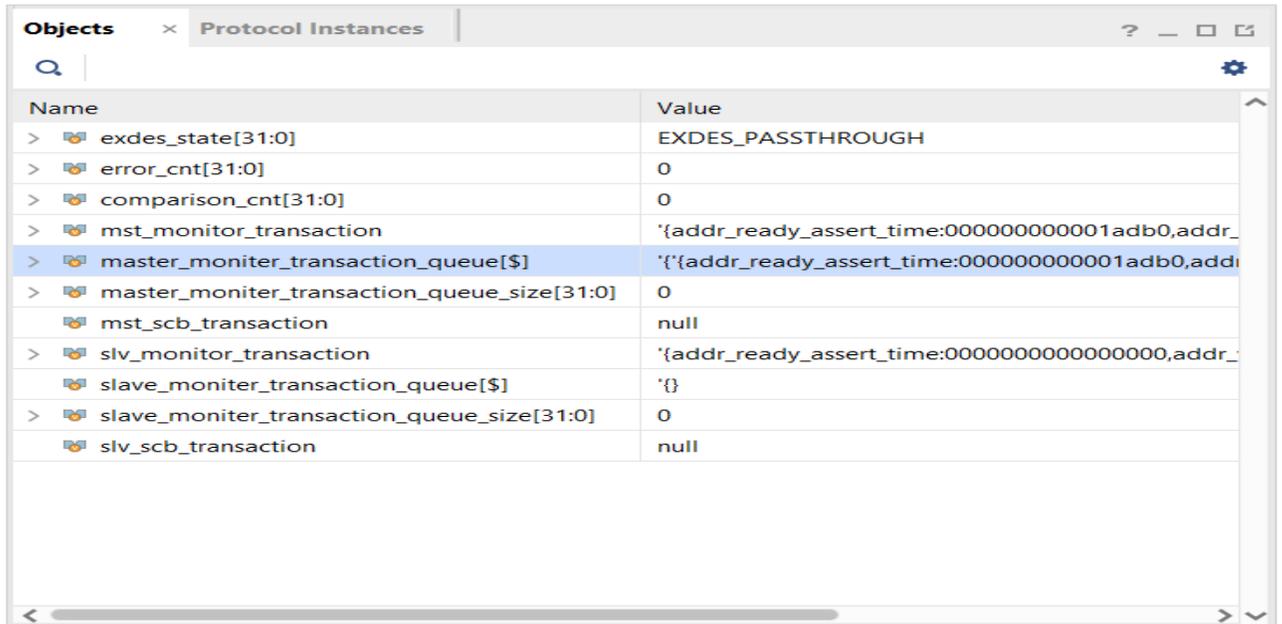


Figure 41: Objects Window

Tcl Console

Like Objects window, you can view the value of any dynamic type variable from Tcl Console as well.

1. Click **Restart** button  .
2. From the Scope window, select scope **generic_tb** present under the top module.

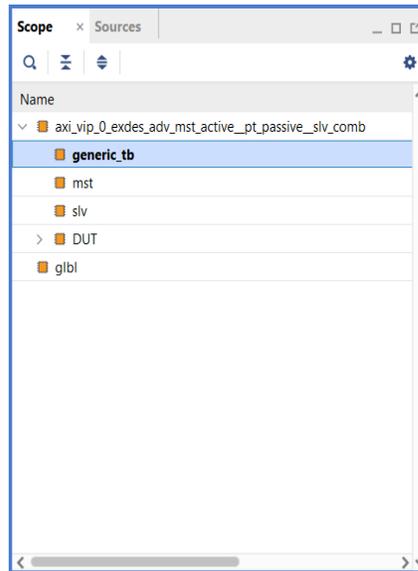


Figure 42: Scope Window

3. In Tcl console, invoke the `report_objects` command to display all objects present in the selected scope. Also, the Queue and Class appear as object type.

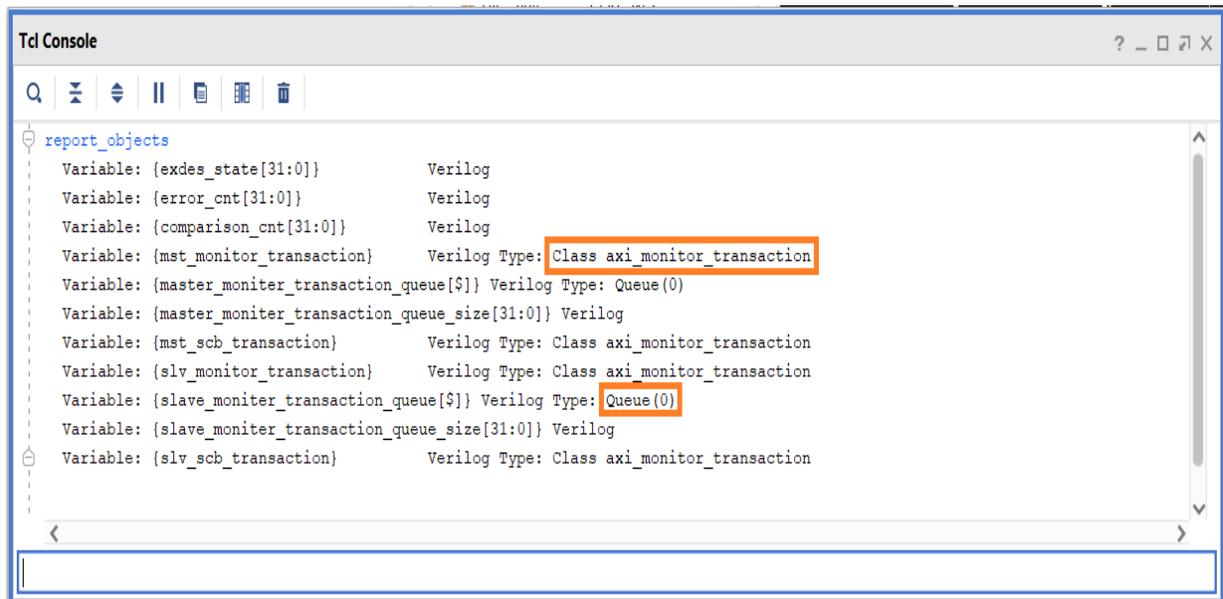


Figure 43: Tcl Console

Cover Group Declaration

```
covergroup my_cover_group @(posedge aclk);
  m_axi_awlen_cp : coverpoint m_axi_awlen;
  m_axi_awcache_cp : coverpoint m_axi_awcache {
    option.comment = "cp with transition bins";
    bins a1 = (1=>2);
    bins b1 = (1,3=>4,5);
    bins b2[] = (1,3=>4,5);
    bins b3 = (1=>2), ([4:6] => 11,12);
  }
  m_axi_bresp_cp : coverpoint m_axi_bresp;
endgroup
my_cover_group obj1 = new();
```

In this example, you are declaring a covergroup named as `my_cover_group` and the sampling event as `posedge aclk`. This covergroup contains three coverpoints. You will add the cover group declaration code in the example design.

1. In Tcl console invoke the following command:

```
current_scope
/axi_vip_0_exdes_adv_mst_active_pt_passive_slv_comb/DUT/ex_design/axi_vip_m
st
```

2. Double-click **axi_vip_mst** scope to see the source code.

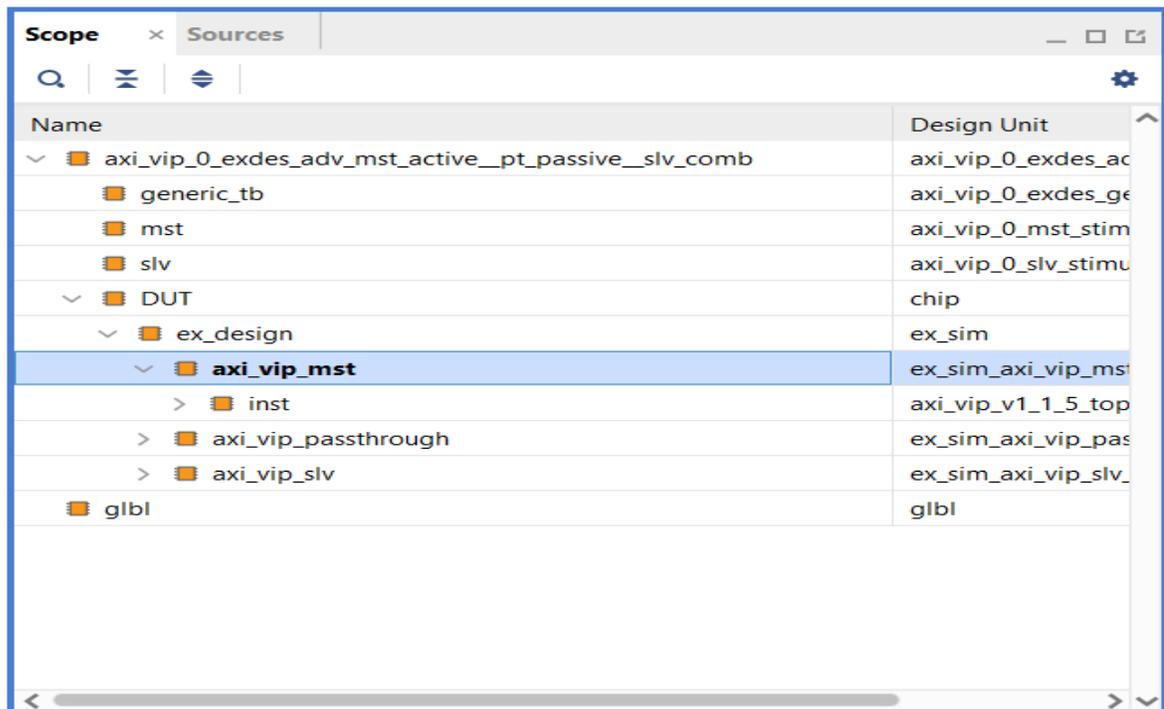
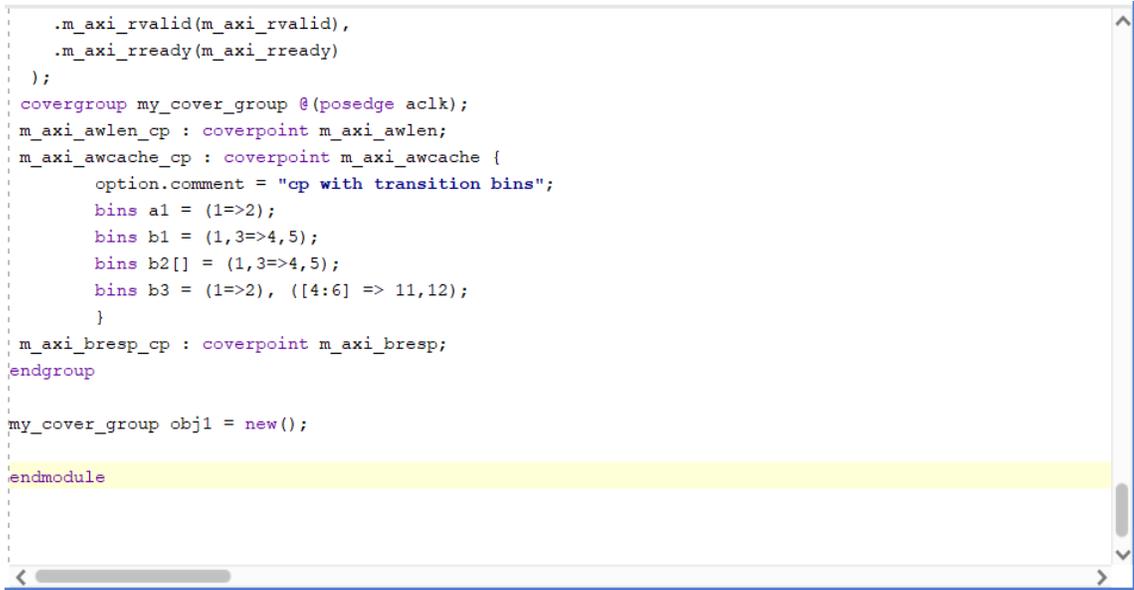


Figure 45: Scope Window

- The following file path will appear in the text editor, then add the cover group declaration before `endmodule` and save it.

```
axi_vip_0_ex/axi_vip_0_ex.ip_user_files/bd/ex_sim/ip/ex_sim_axi_vip_mst_0/sim
/ex_sim_axi_vip_mst_0.sv
```



```
.m_axi_rvalid(m_axi_rvalid),
.m_axi_rready(m_axi_rready)
);
covergroup my_cover_group @(posedge aclk);
m_axi_awlen_cp : coverpoint m_axi_awlen;
m_axi_awcache_cp : coverpoint m_axi_awcache {
    option.comment = "cp with transition bins";
    bins a1 = (1=>2);
    bins b1 = (1,3=>4,5);
    bins b2[] = (1,3=>4,5);
    bins b3 = (1=>2), ([4:6] => 11,12);
}
m_axi_bresp_cp : coverpoint m_axi_bresp;
endgroup

my_cover_group obj1 = new();

endmodule
```

Figure 46: Text Editor

- In Tcl console, invoke `close_sim` command to close the simulation running previously.
- In Tcl console, invoke `reset_simulation` command to clean the simulation directory.
- In Tcl console, invoke `launch_simulation` command to run the simulation.
- Click **Run All** button ▶.

The simulation will stop after reaching `$finish` statement. The Vivado simulator has generated the coverage database at the following location with the name (default name) `xsim.covdb`:

```
./axi_vip_0_ex.sim/sim_adv_mst_active_pt_passive_slv_comb/behav/xsim/
```

Invoke the following command to generate a report:

```
xcrp -report_format html -dir
./axi_vip_0_ex.sim/sim_adv_mst_active_pt_passive_slv_comb/behav/xsim/xsim.covdb/
```

This will generate a directory with the name `xcrq_report`, it contains a `.html` report. The following is the description of an example report:

1. Open `dashboard.html` file. The file contains details such as command, version, date and coverage summary that shows only 8.85% of total bins are covered.

XILINX		Dashboard	Groups
Date	04/18/19 15:39:59 India Standard Time		
User	dprasad		
Version	Vivado Simulator Coverage Report 2019.1		
Command Line	Y:/2019.1_daily_latest/installs/nt64/Vivado/HEAD/bin/unwrapped/win64.o/xcrq.exe - report_format html -dir ./axi_vip_0_ex.sim/sim_adv_mst_active_pt_passive_slv_comb/behav/xsim/xsim.covdb/		
Number of Tests	1		

Total Groups Coverage Summary	
Score	Inst Score
8.85417	8.85417

Figure 47: XCRG Dashboard

2. Click **Groups** button .
3. Click the **Link** under group report.

Total groups in report: 1		
Name 	Score 	Nur Inst
axi_vip_0_exdes_adv_mst_active_pt_passive_slv_comb.DUT.ex_design.axi_vip_mst.inst:my_cover_group	8.85417	1

Figure 48: XCRG Group Report

4. You will see a detailed report as shown in the following figure:

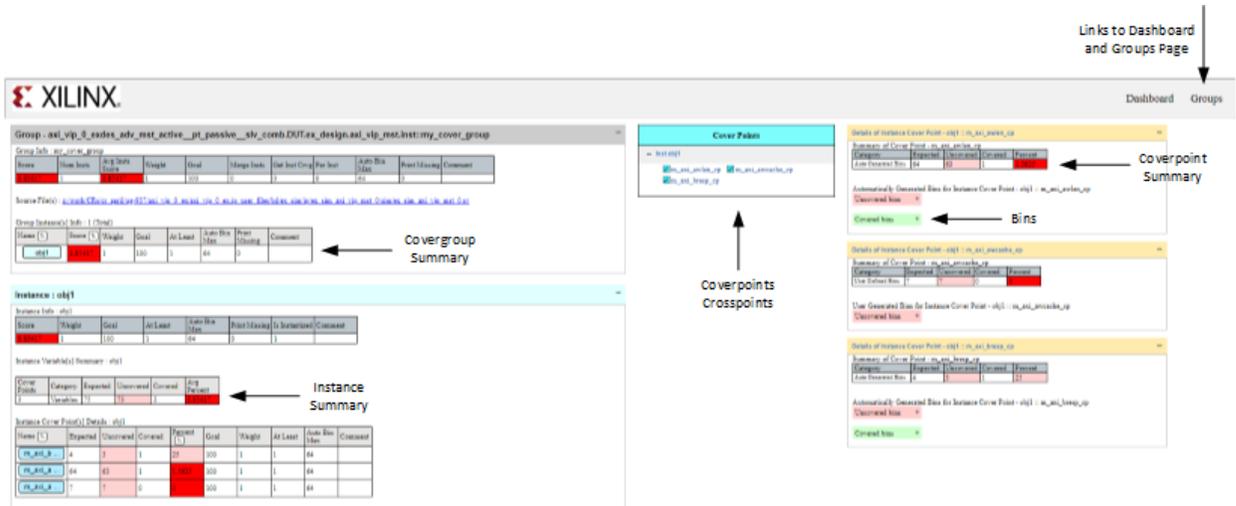


Figure 49: XCRG Detailed Report

This way you can view the coverage and change your test bench/seed value to improve the coverage.

Assertion

In System Verilog, you have the following two types of assertion:

- Immediate assertion
- Concurrent Assertion

Immediate Assertion

Evaluated like an expression in 'if' statement.

```
always@(posedge clk)
    assert(data == 4'b1010);
```

Concurrent Assertion

This assertion is based on clock semantic and use sampled value of their expression. These assertions can expand over multiple cycle.

```
always@ (posedge clk)
    a1: assert property (a ##2 b);

    a2: assert property (@(posedge clk) a ##2b);
```

In Vivado simulator, the concurrent assertion of 2nd form that is used outside the procedural block is supported.

1. In Tcl console, invoke `close_sim` command to close the simulation running previously.
2. In Tcl console, invoke `reset_simulation` command to clean the simulation directory.
3. In Tcl console, invoke `launch_simulation` command to run the simulation.
4. In Tcl console, invoke the following command:

```
current_scope
/axi_vip_0_exdes_adv_mst_active__pt_passive__slv_comb/DUT/ex_design/axi_vip_p
assthrough/inst/IF/PC
```

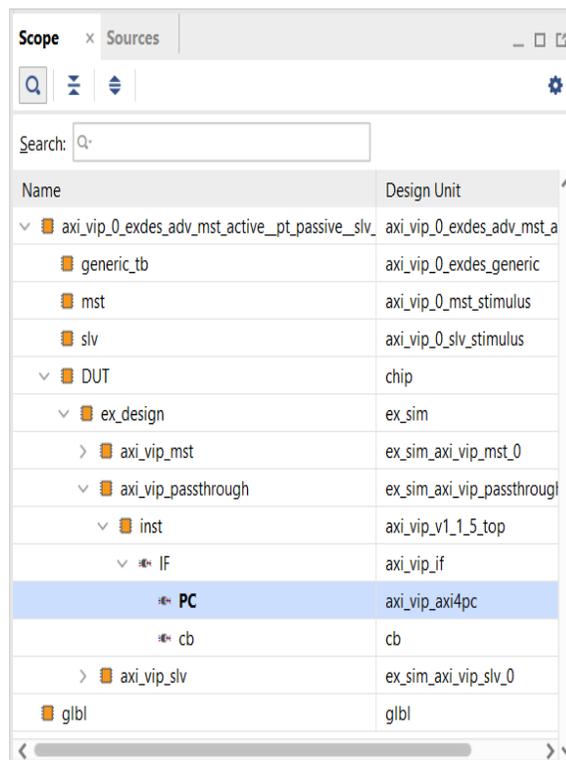


Figure 50: Scope Window

5. Double-click the scope **PC** to open the source code.
6. Look at line no 1669 onwards. All the property declarations and assertions have been used.

Random Constraint

System Verilog has random constraint, which is used to generate a random value. Using this feature, you can even set the constraint on a random variable.

For each simulation, the simulator is supposed to generate fixed set of values. In this example, randomize call is happening 10 times so each time the simulator is expected to assign a different value

on variable 'b1'. If we close the simulation and run it again, the simulator is expected to give same 10 set of values like the previous run. This is called as random stability.

```
module top();
class c1;
rand bit [3:0] b1;
endclass
c1 obj1 = new();
initial
begin
    for(int i = 0; i < 10; i++)
    begin
        #5 obj1.randomize();
        $display("At time %t the value is %p", $time, obj1);
    end
end
endmodule
```

If you want different set of values, you should change the random seed value. In Vivado simulator, it is done by passing -seed option to xsim. In Tcl Console, you need to invoke the following command:

```
set_property -name {xsim.simulate.xsim.more_options} -value {-seed 2000} -objects
[get_filesets sim_adv_mst_active__pt_passive__slv_comb]
```

With seed, you have to provide any integer value. So just by changing a 'seed' value, you can get a different value. You don't need to do compilation and elaboration again.

1. Add the following code in a file and name it as random.sv.

```
module top();
class c1;
rand bit [3:0] b1;
endclass
c1 obj1 = new();
initial
begin
    for(int i = 0; i < 10; i++)
    begin
        #5 obj1.randomize();
        $display("At time %t the value is %p", $time, obj1);
    end
end
endmodule
```

2. Perform the following in Tcl console:

Logic Simulation

UG937 (v2019.1) June 4, 2019

- a. Invoke, `xvlog -sv random.sv` command to compile the code.
- b. Invoke, `xelab top -s top` command to elaborate the code.
- c. Invoke, `xsim top -R` command to simulate the code.

Notice o/p

run-all

```
At time          5000 the value is '{b1:3}
At time          10000 the value is '{b1:7}
At time          15000 the value is '{b1:7}
At time          20000 the value is '{b1:0}
At time          25000 the value is '{b1:0}
At time          30000 the value is '{b1:5}
At time          35000 the value is '{b1:9}
At time          40000 the value is '{b1:3}
At time          45000 the value is '{b1:12}
At time          50000 the value is '{b1:0}
exit
```

3. Re-run step 2b and notice the value is similar to the previous one.
4. Simulate the code with different SV seed `xsim top -R -sv_seed 5000` and observe that the value is different. Thus, you can generate different value without going through compile and elaboration step.

Please Read: Important Legal Notices

The information disclosed to you hereunder (the "Materials") is provided solely for the selection and use of Xilinx products. To the maximum extent permitted by applicable law: (1) Materials are made available "AS IS" and with all faults, Xilinx hereby DISCLAIMS ALL WARRANTIES AND CONDITIONS, EXPRESS, IMPLIED, OR STATUTORY, INCLUDING BUT NOT LIMITED TO WARRANTIES OF MERCHANTABILITY, NON-INFRINGEMENT, OR FITNESS FOR ANY PARTICULAR PURPOSE; and (2) Xilinx shall not be liable (whether in contract or tort, including negligence, or under any other theory of liability) for any loss or damage of any kind or nature related to, arising under, or in connection with, the Materials (including your use of the Materials), including for any direct, indirect, special, incidental, or consequential loss or damage (including loss of data, profits, goodwill, or any type of loss or damage suffered as a result of any action brought by a third party) even if such damage or loss was reasonably foreseeable or Xilinx had been advised of the possibility of the same. Xilinx assumes no obligation to correct any errors contained in the Materials or to notify you of updates to the Materials or to product specifications. You may not reproduce, modify, distribute, or publicly display the Materials without prior written consent. Certain products are subject to the terms and conditions of Xilinx's limited warranty, please refer to Xilinx's Terms of Sale which can be viewed at <http://www.xilinx.com/legal.htm#tos>; IP cores may be subject to warranty and support terms contained in a license issued to you by Xilinx. Xilinx products are not designed or intended to be fail-safe or for use in any application requiring fail-safe performance; you assume sole risk and liability for use of Xilinx products in such critical applications, please refer to Xilinx's Terms of Sale which can be viewed at <http://www.xilinx.com/legal.htm#tos>.

AUTOMOTIVE APPLICATIONS DISCLAIMER

AUTOMOTIVE PRODUCTS (IDENTIFIED AS "XA" IN THE PART NUMBER) ARE NOT WARRANTED FOR USE IN THE DEPLOYMENT OF AIRBAGS OR FOR USE IN APPLICATIONS THAT AFFECT CONTROL OF A VEHICLE ("SAFETY APPLICATION") UNLESS THERE IS A SAFETY CONCEPT OR REDUNDANCY FEATURE CONSISTENT WITH THE ISO 26262 AUTOMOTIVE SAFETY STANDARD ("SAFETY DESIGN"). CUSTOMER SHALL, PRIOR TO USING OR DISTRIBUTING ANY SYSTEMS THAT INCORPORATE PRODUCTS, THOROUGHLY TEST SUCH SYSTEMS FOR SAFETY PURPOSES. USE OF PRODUCTS IN A SAFETY APPLICATION WITHOUT A SAFETY DESIGN IS FULLY AT THE RISK OF CUSTOMER, SUBJECT ONLY TO APPLICABLE LAWS AND REGULATIONS GOVERNING LIMITATIONS ON PRODUCT LIABILITY.

© Copyright 2012-2019 Xilinx, Inc. Xilinx, the Xilinx logo, Artix, ISE, Kintex, Spartan, Virtex, 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.