

Vivado Design Suite Tutorial

Logic Simulation

UG937 (v2018.1) April 4, 2018





Table of Contents

Revision History	2
Vivado Simulator Overview.....	4
Introduction	4
Tutorial Description.....	5
Locating Tutorial Design Files	6
Software and Hardware Requirements.....	7
Lab 1: Running the Simulator in Vivado IDE	8
Introduction	8
Step 1: Creating a New Project.....	8
Step 2: Adding IP from the IP Catalog.....	14
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.....	33
Step 6: Saving the Waveform Configuration.....	34
Step 7: Re-Simulating the Design	36
Step 8: Using Cursors, Markers, and Measuring Time	38
Step 9: Debugging with Breakpoints	41
Step 10: Relaunch Simulation	47
Conclusion	49

Revision History

Section	Revision Summary
04/04/2018 Version 2018.1	
General updates	Updated menu commands
	Updated to 2018.1 version
	Updated editorial issues

Extra Material

Title	Location	Summary	Source
Understanding the Flow Navigator	page 14	Reviews the overall layout and function of the Flow Navigator	Vivado Design Flows Overview Tutorial (UG892)
Running the Vivado Simulator	page 25	Explains the features of the XSim GUI, and their purposes	Vivado Logic Simulation Tutorial (UG900)

Note Symbol in PDF

Wherever you see this symbol in this tutorial, be sure to read the note I have added. Failure to do so might make things much more difficult for you! Mouse over the symbol to read the text.

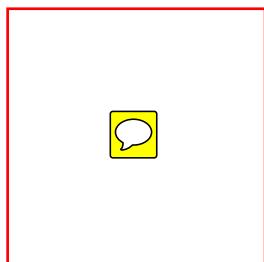
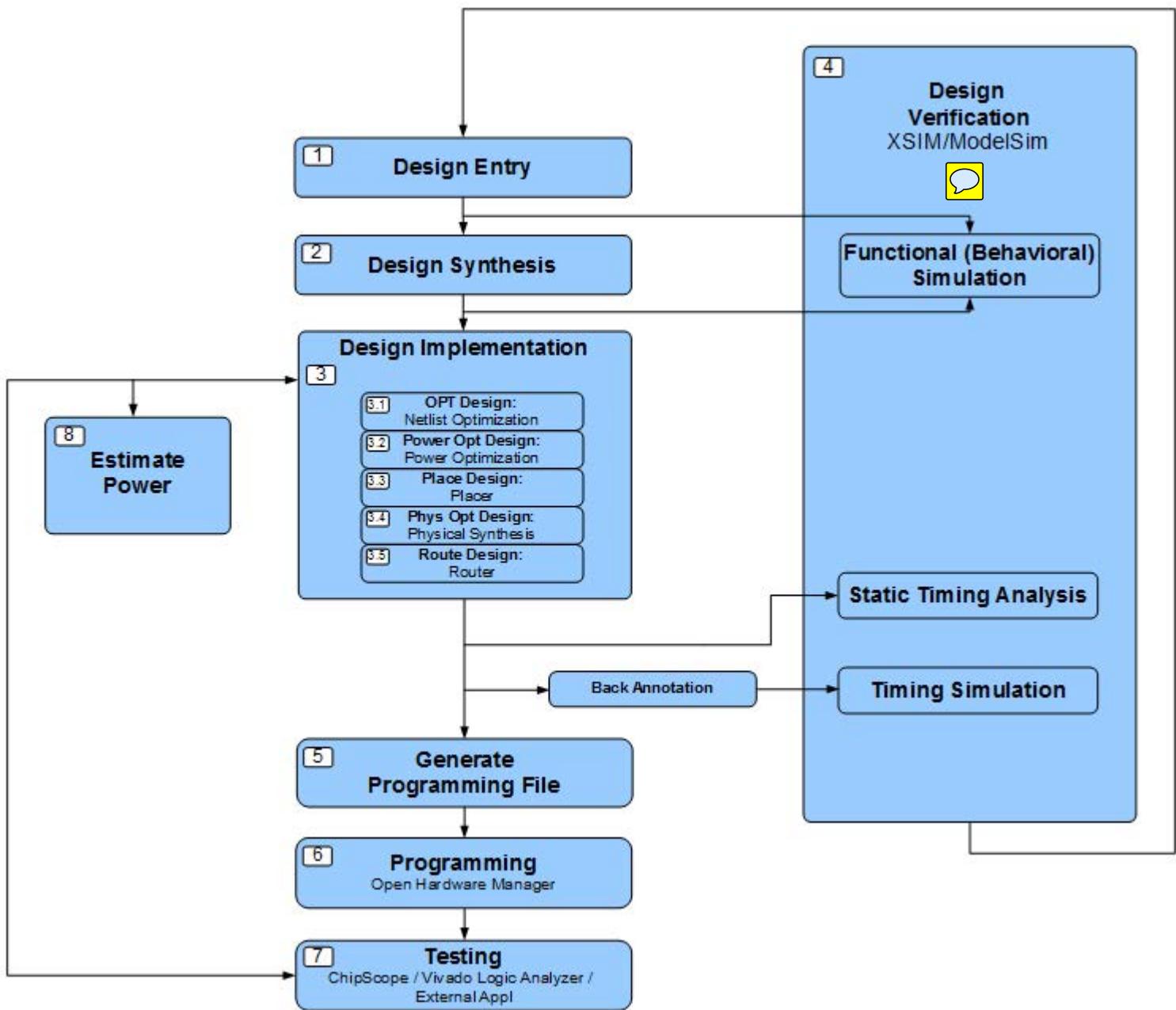


Figure 0: Note Symbol

Vivado FPGA Design Flow





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

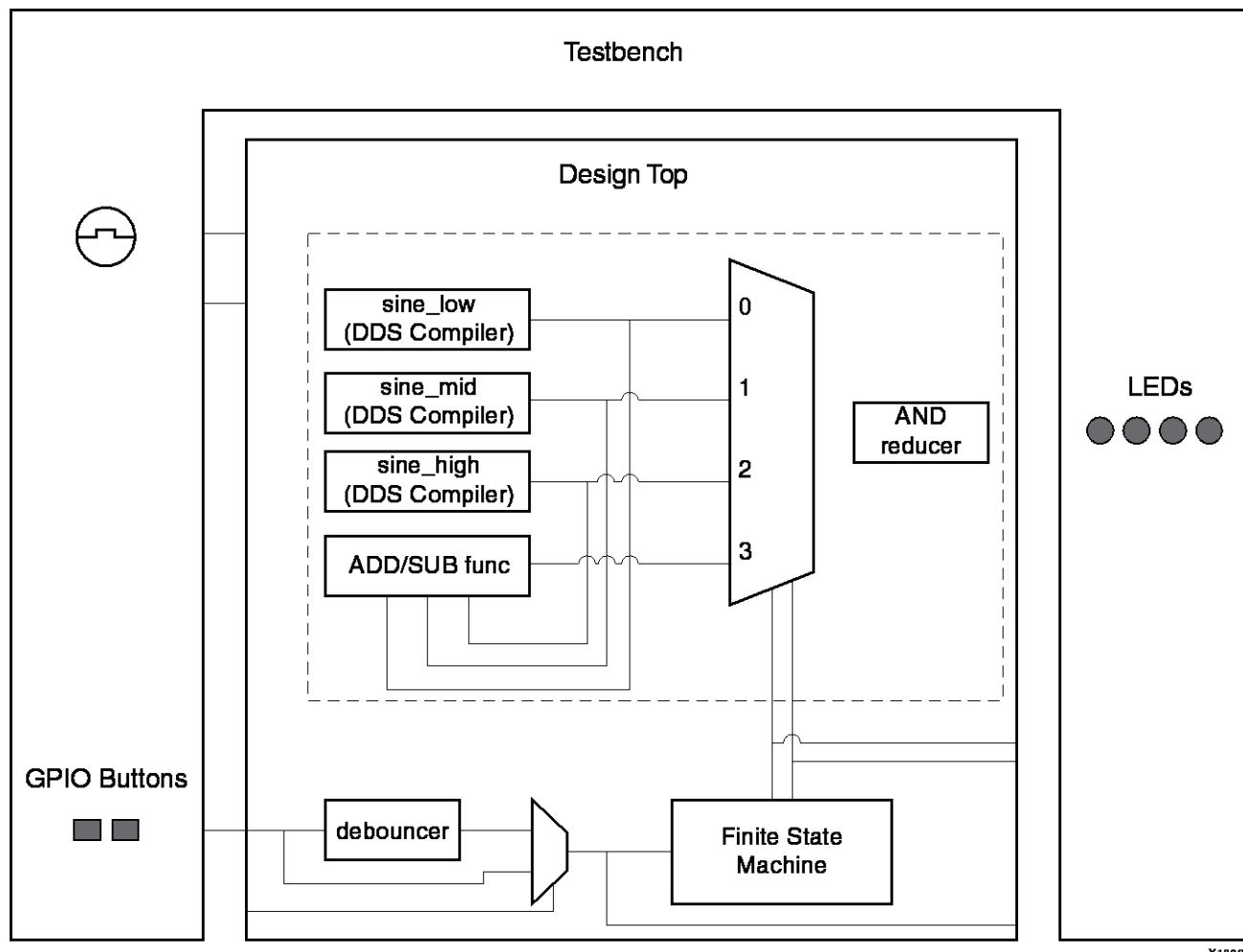
This tutorial demonstrates a design flow in which you can use the Vivado simulator for performing behavioral, functional, or timing simulation from the Vivado Integrated Design Environment (IDE).



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.



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 2018.x project of the tutorial design for reference. (x denotes the latest version of Vivado 2018 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 2018.1 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.

Lab 1: Running the Simulator in Vivado IDE

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.

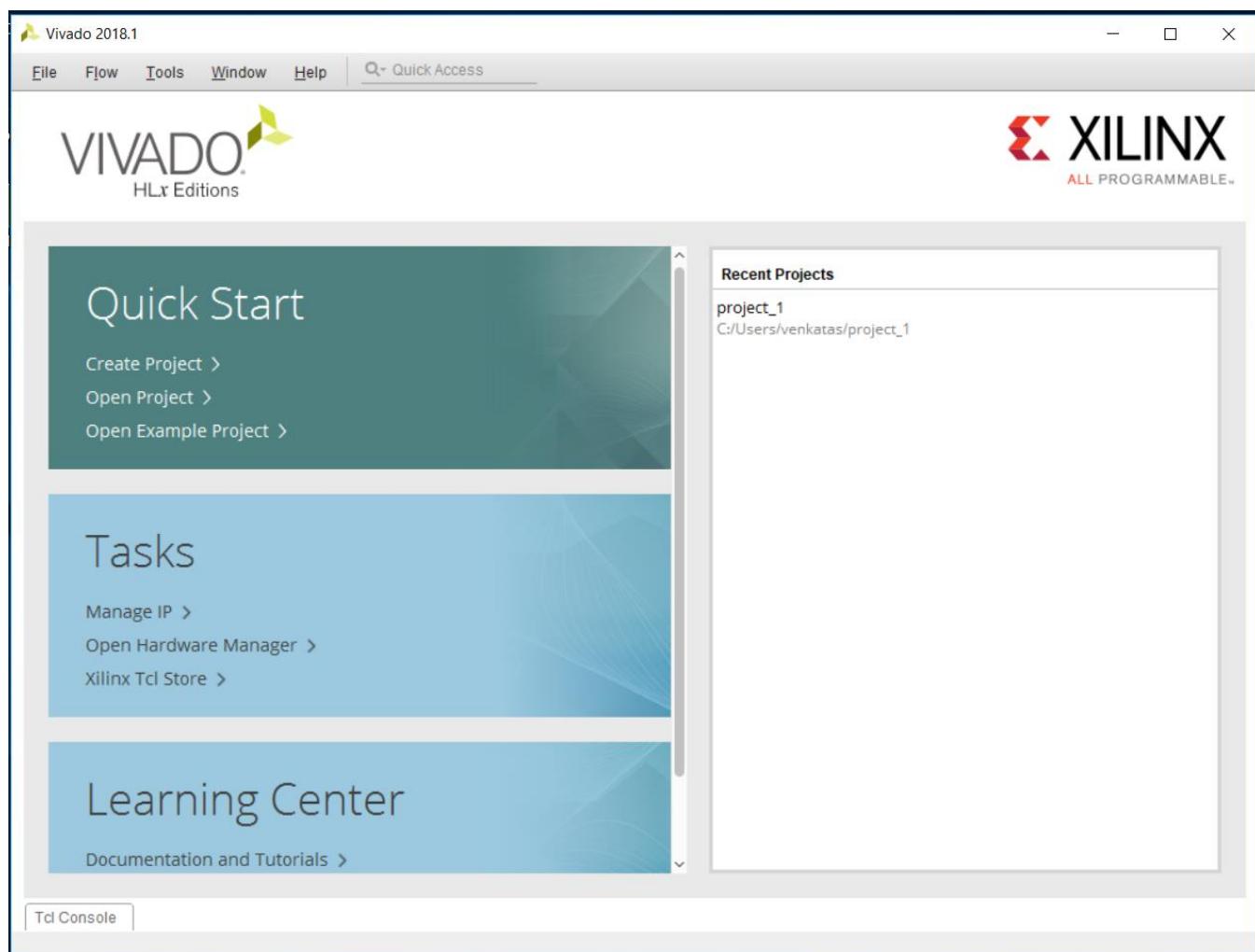


Figure 2: Vivado IDE - Getting Started Page

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 2018.x > Vivado 2018.x

(x denotes the latest version of Vivado 2018 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_xsimsim*.
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).

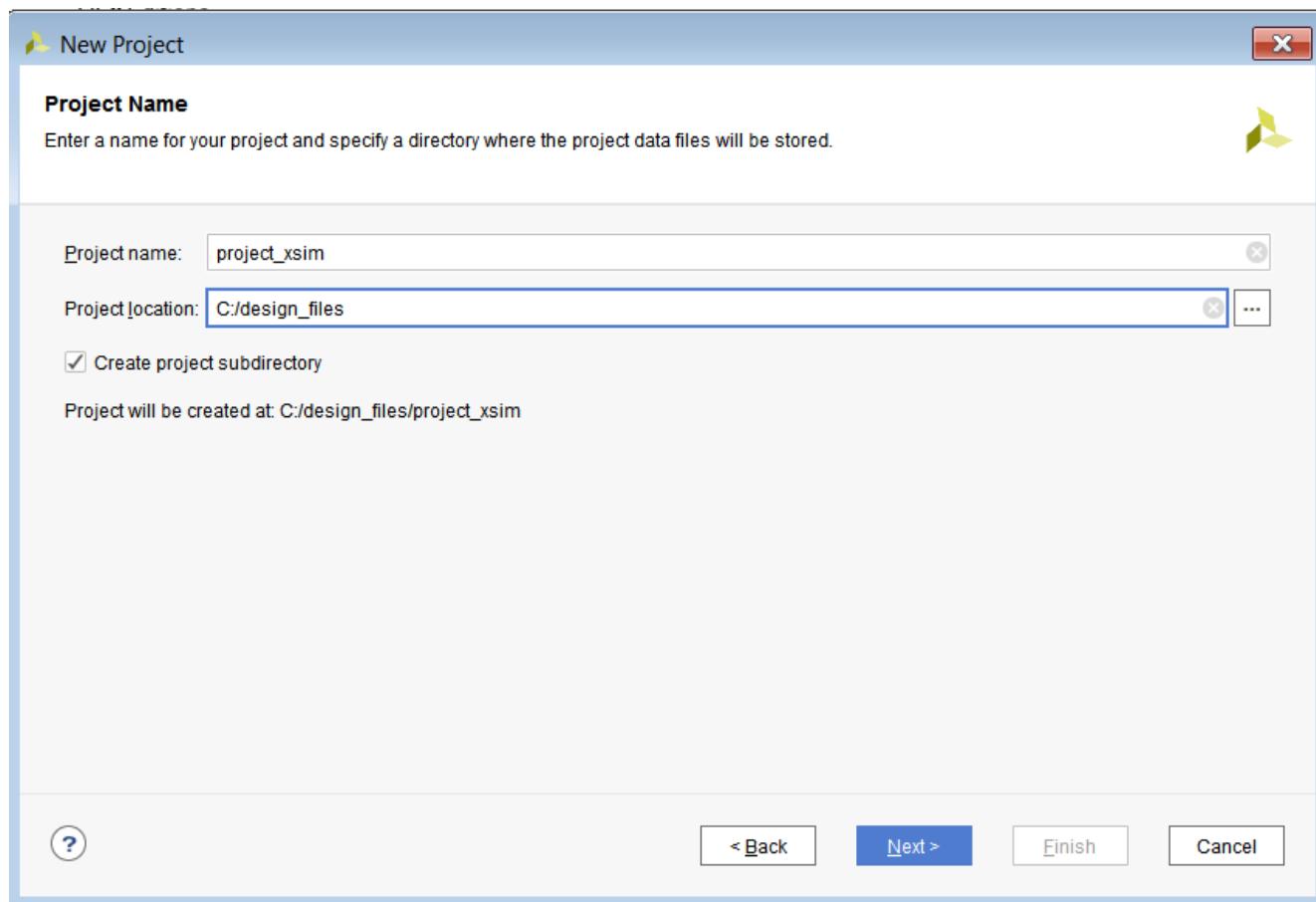


Figure 3: Create Project

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

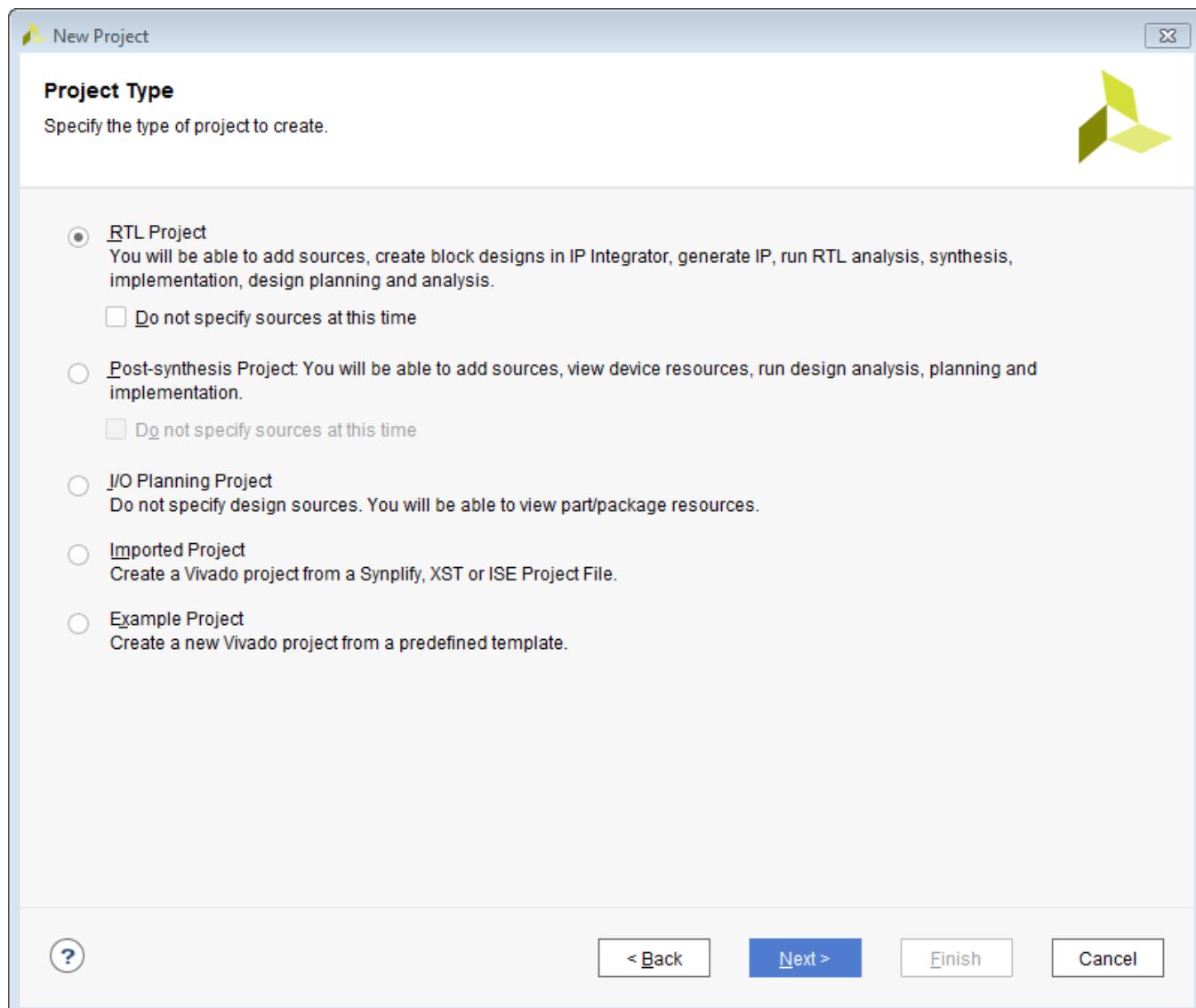


Figure 4: Selecting RTL Project

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 Logic Simulation

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.

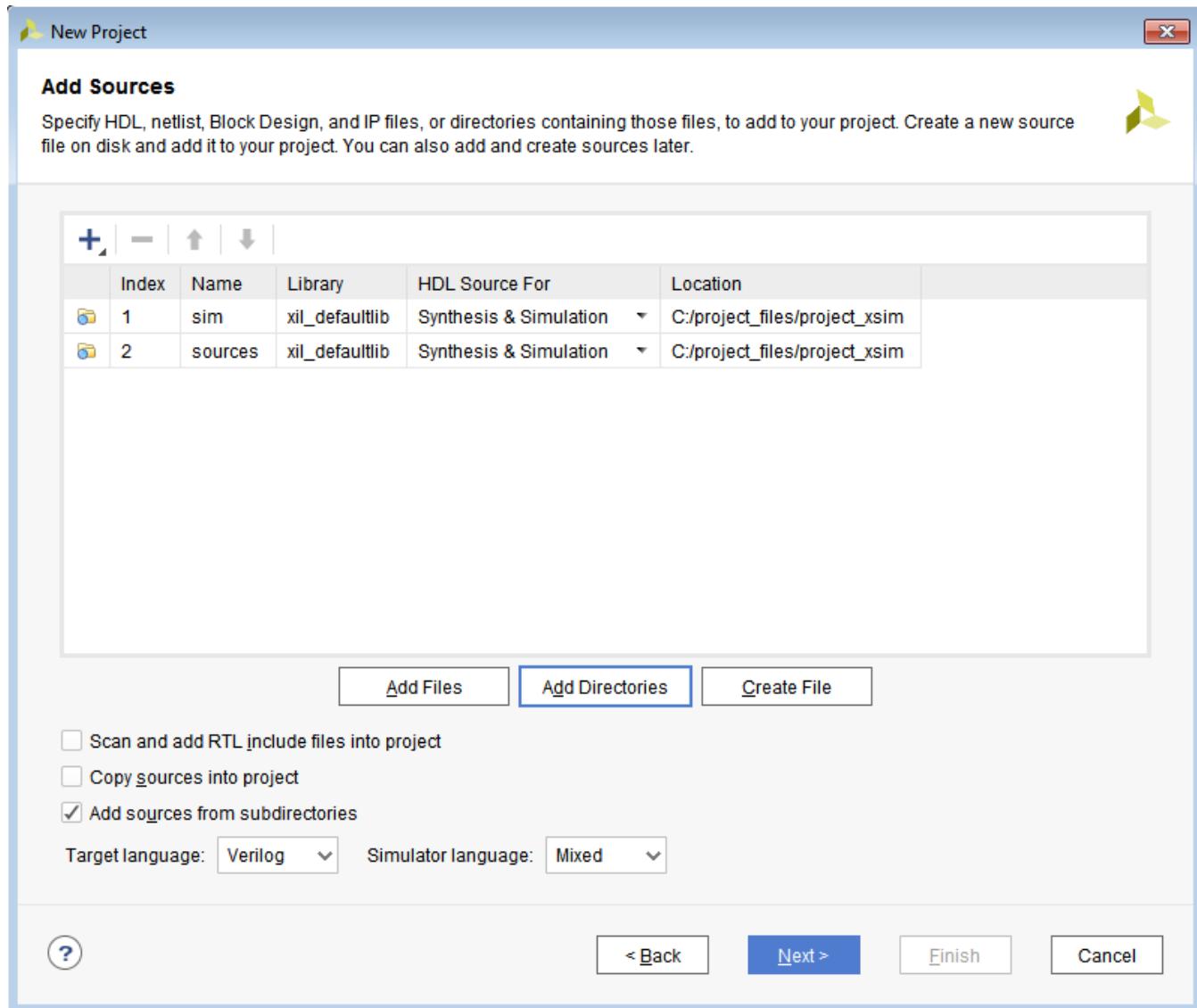


Figure 5: Add Sources

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**. 

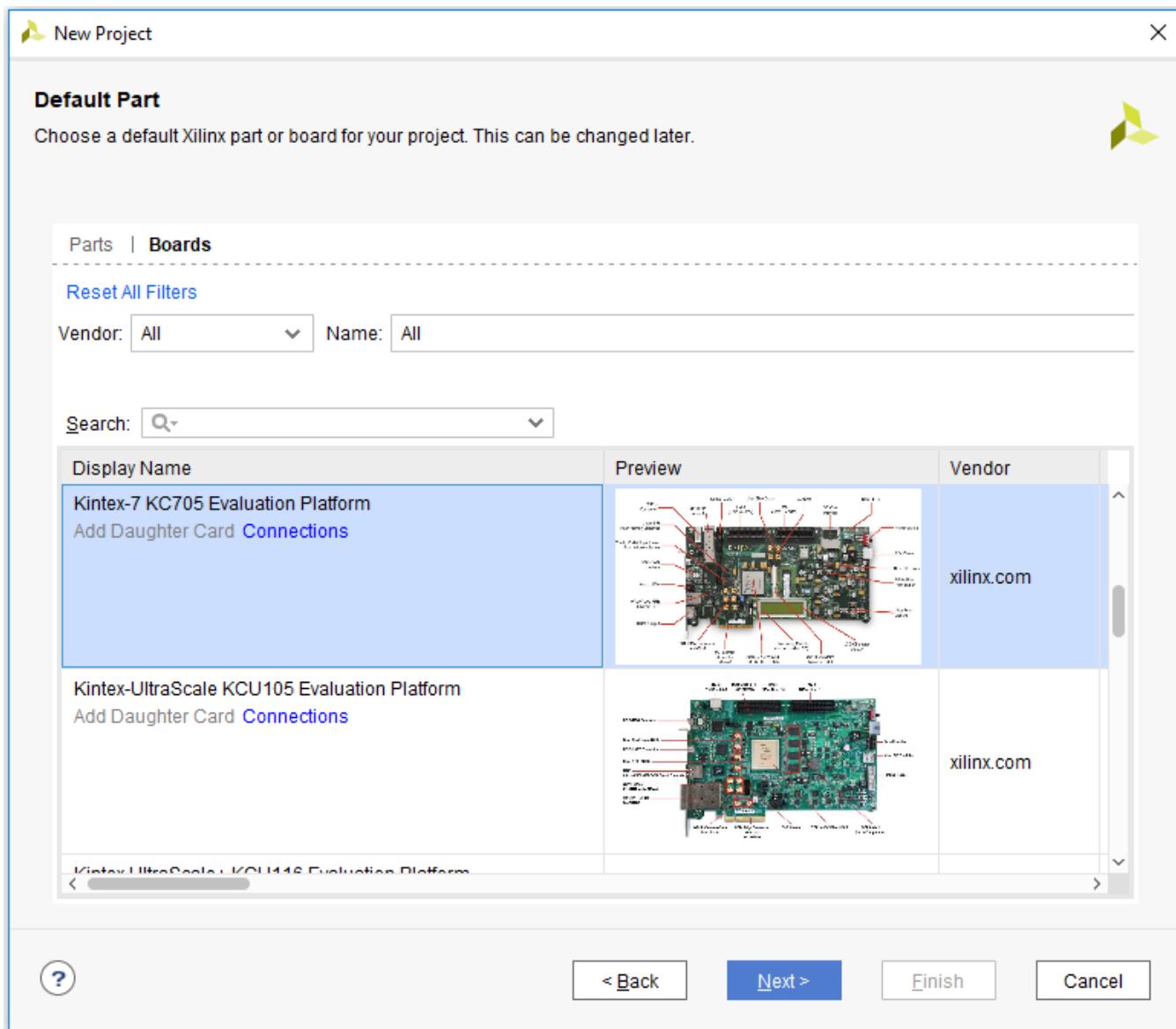
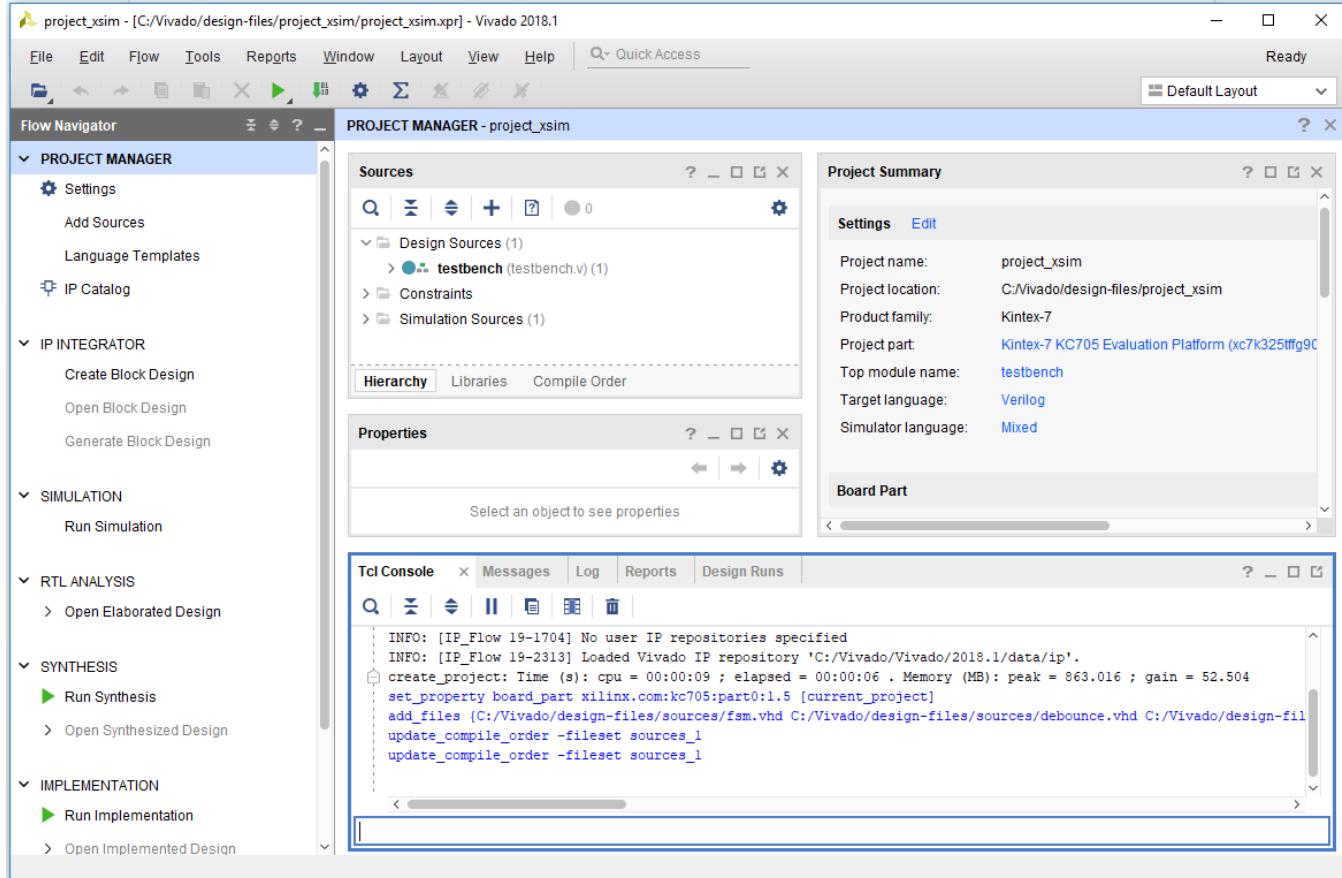


Figure 6: Specify Default Part or Board

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).



Understanding the Flow Navigator



The Flow Navigator (Figure 3-2) provides control over the major design process tasks, such as project configuration, synthesis, implementation, and bitstream generation. The commands and options available in the Flow Navigator depend on the status of the design. Unavailable steps are grayed out until required design tasks are completed.

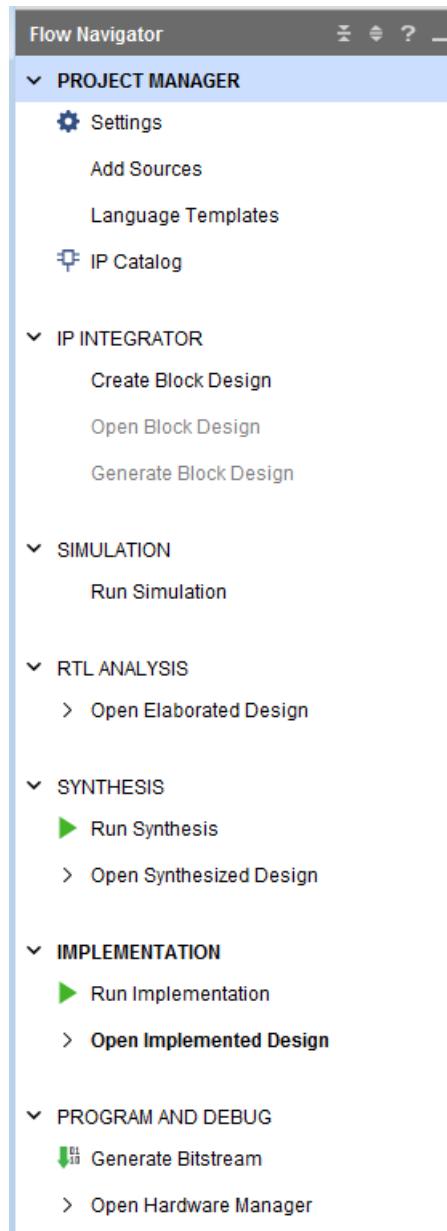


Figure 3-2: Flow Navigator

The Flow Navigator (Figure 3-3) differs when working with projects created with third-party netlists. For example, system-level design entry, IP, and synthesis options are not available.

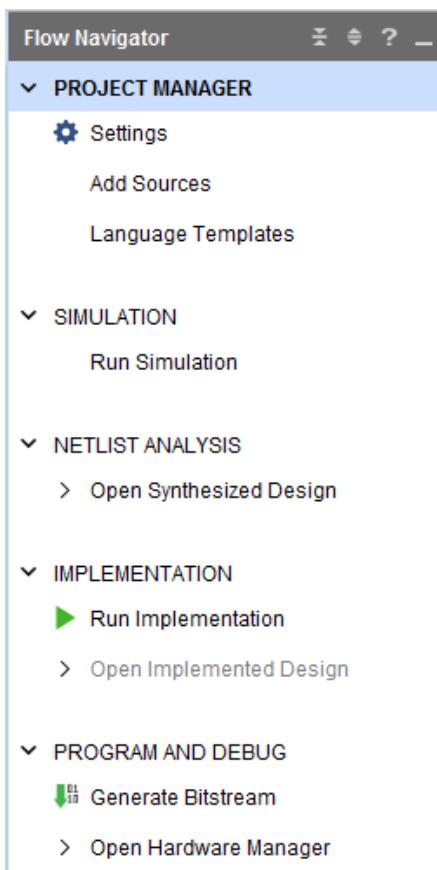


Figure 3-3: Flow Navigator for Third-Party Netlist Project

As the design tasks complete, you can open the resulting designs to analyze results and apply constraints. In the Flow Navigator, click **Open Elaborated Design**, **Open Synthesized Design**, or **Open Implemented Design**. For more information, see [Opening Designs to Perform Design Analysis and Constraints Definition](#).

When you open a design, the Flow Navigator shows a set of commonly used commands for the applicable phase of the design flow. Selecting any of these commands in the Flow Navigator opens the design, if it is not already opened, and performs the operation. For example, [Figure 3-4](#) shows the commands related to synthesis.

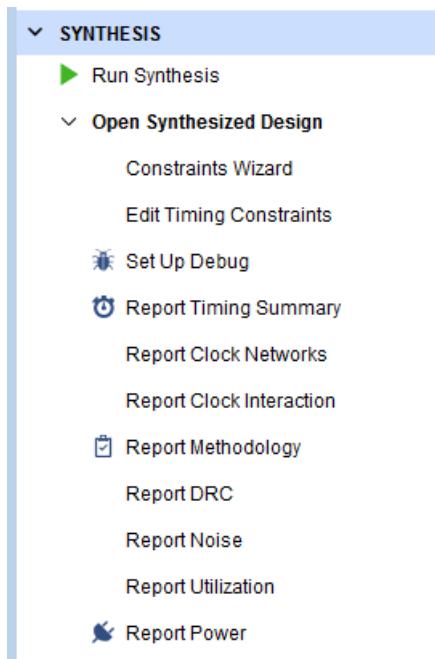


Figure 3-4: Synthesis Section in the Flow Navigator



Taken from:

Design Flows Overview

UG892 (v2018.1) April 4, 2018

www.xilinx.com

[Send Feedback](#)

iii

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.

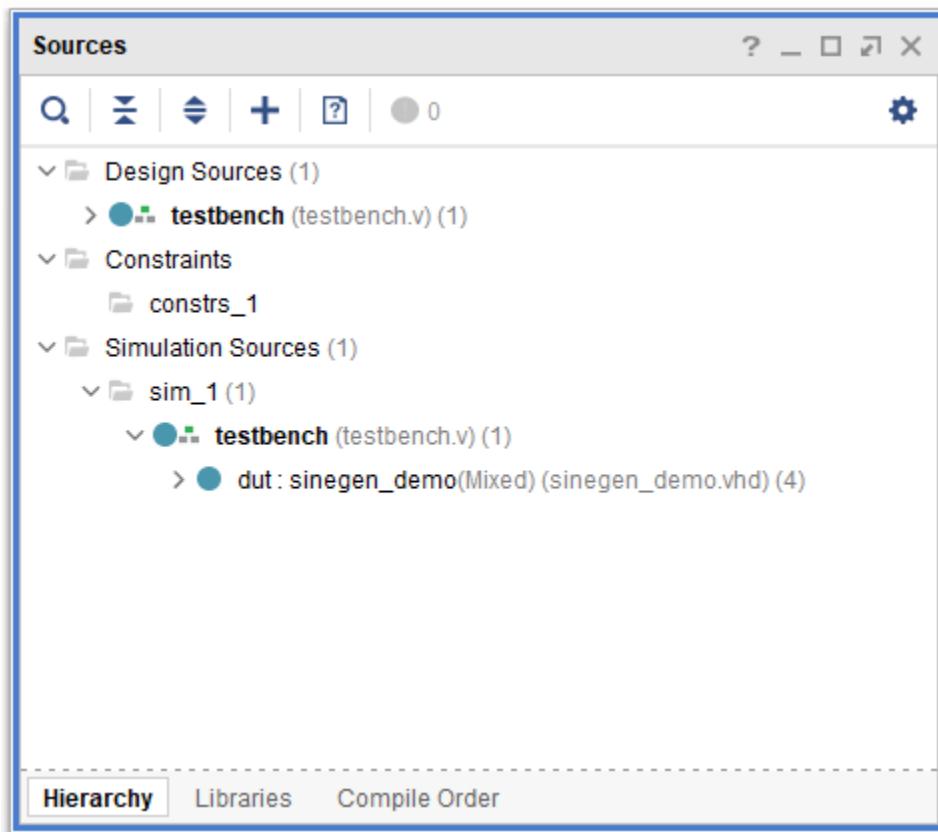


Figure 8: Sources window

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](#)).

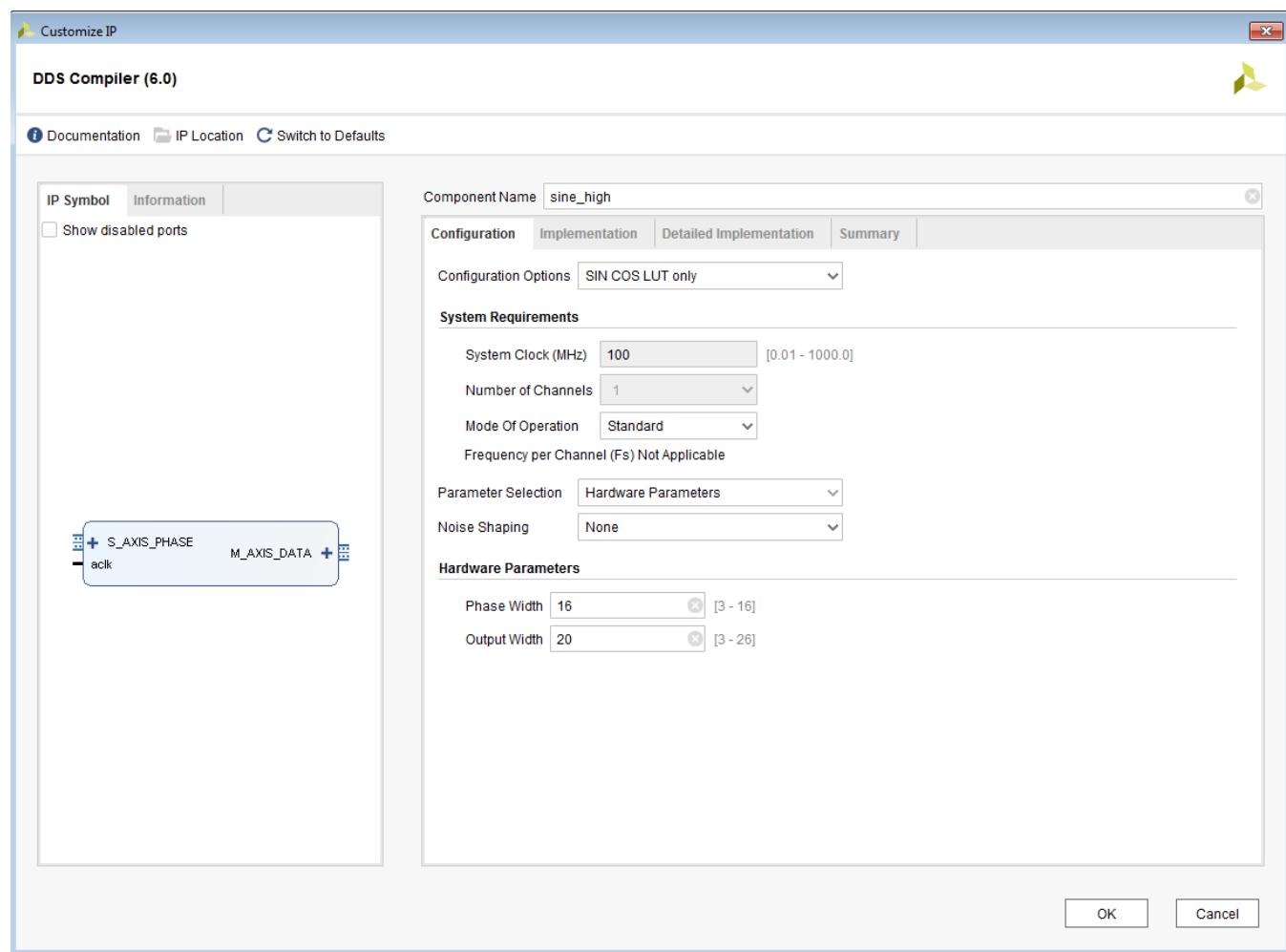
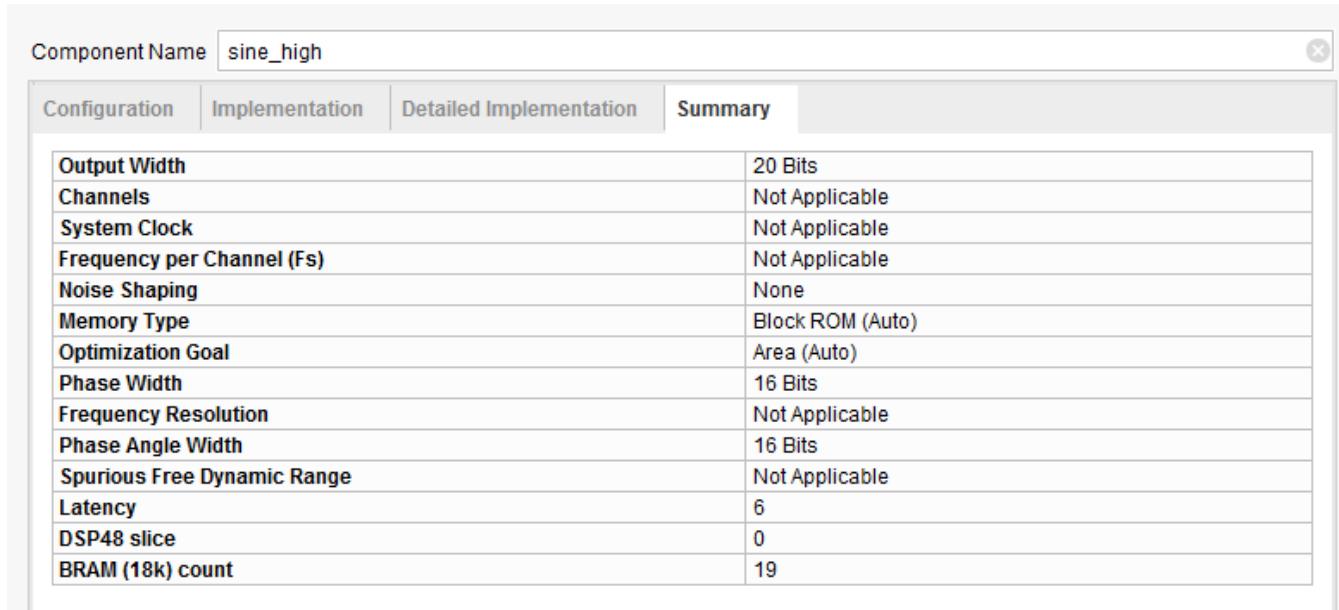


Figure 9: Customize IP - DDS Compiler

4. In the IP Symbol on the left, ensure that **Show disabled ports** is unchecked.
5. Specify the following on the **Configuration** tab:

- **Component Name:** type *sine_high*
- **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**.



The screenshot shows the 'sine_high' IP core in the Vivado IP Catalog. The 'Summary' tab is selected. The component name is 'sine_high'. The table lists various parameters and their values:

Parameter	Value
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.

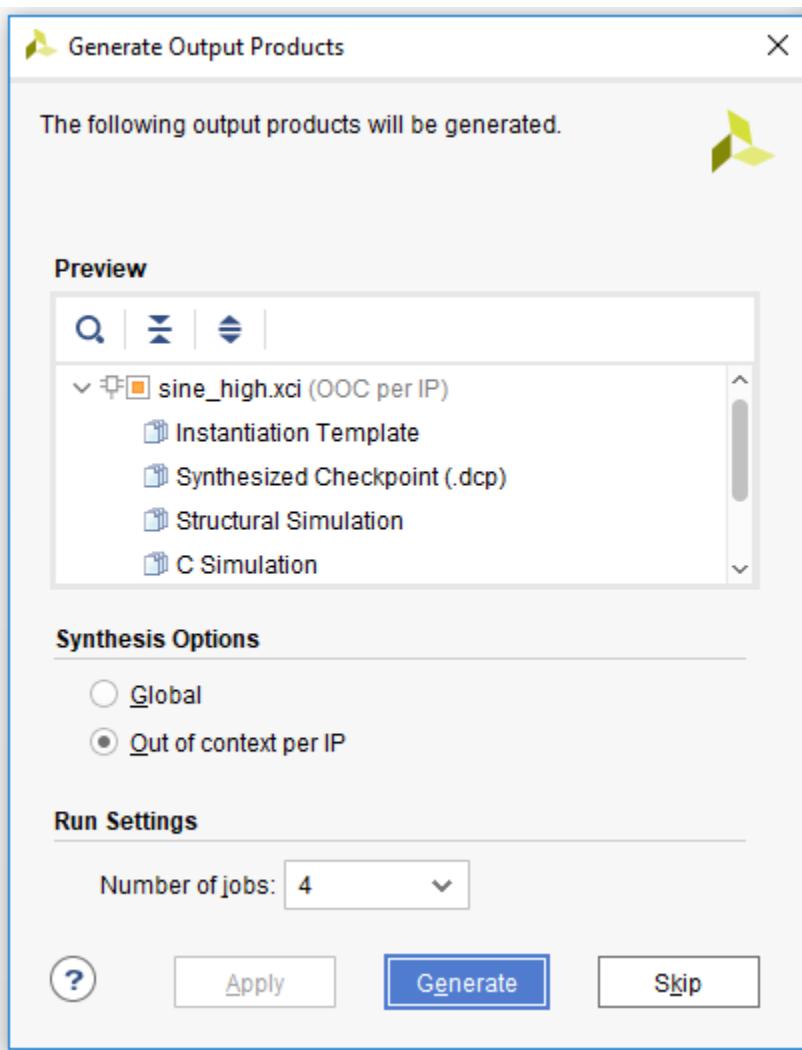


Figure 11: Generate Output Products

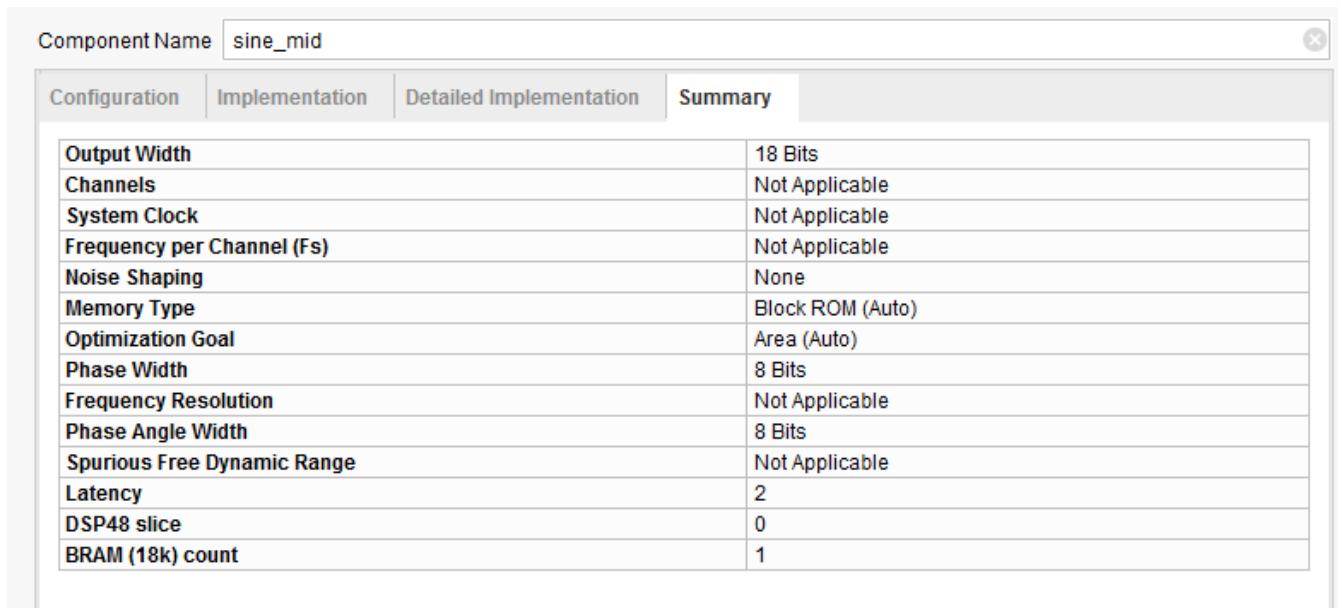
The output products allow the IP to be synthesized, simulated, and implemented as part of the design. For more information on working with IP cores and the Xilinx IP Catalog, refer to the *Vivado Design Suite User Guide: Design with IP* ([UG896](#)). You can also work through the *Vivado Design Suite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

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

Adding Sine Mid

- In the **IP catalog**, double-click the **DDS Compiler** IP a second time.
- 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](#)).



The screenshot shows the 'sine_mid' IP core summary in the Xilinx Vivado IDE. The 'Summary' tab is selected. The table lists the following parameters and their values:

Parameter	Value
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

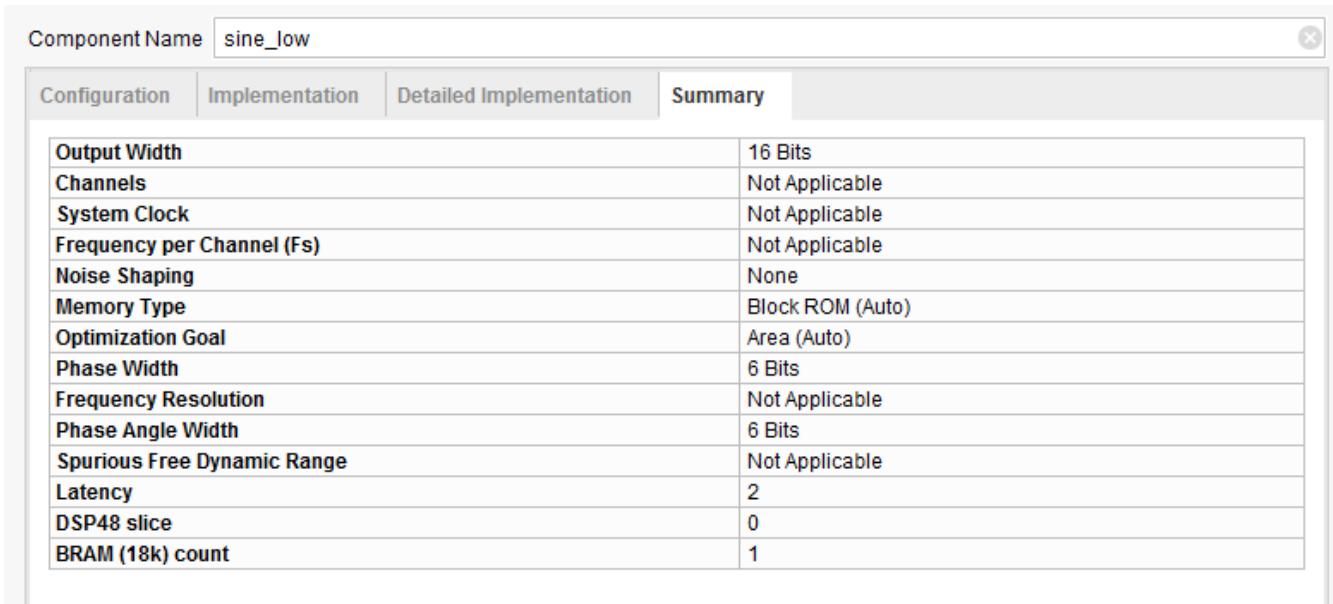
Figure 12: Sine Mid Summary

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**.



The screenshot shows the Xilinx IP Catalog interface with the component name "sine_low" entered in the search bar. The "Summary" tab is selected, displaying the following configuration parameters:

Parameter	Value
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:

1. 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**
2. In the **Elaboration** tab (Figure 14), ensure that the debug level is set to **typical**, which is the default value.

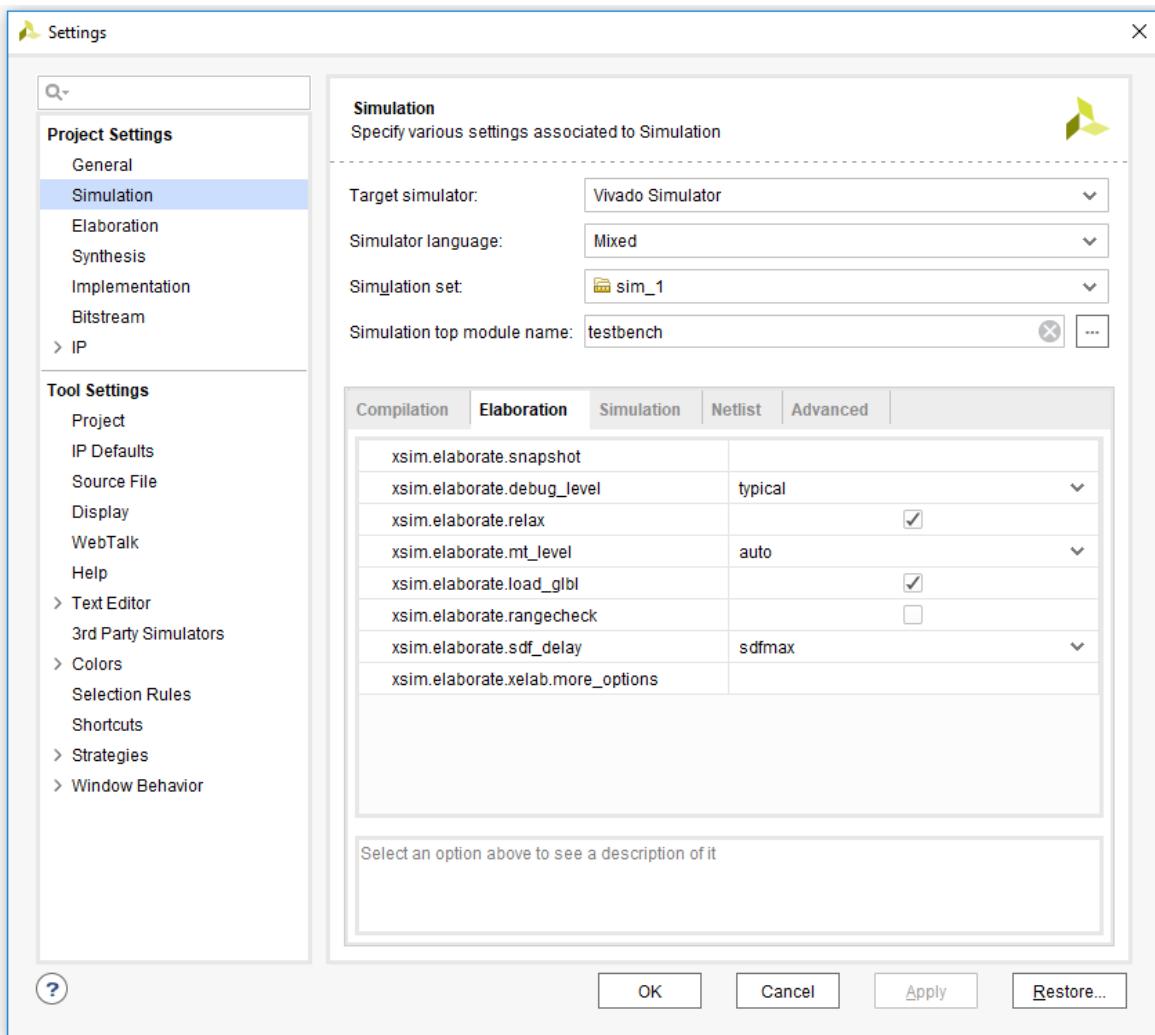


Figure 14: Simulation Settings: Elaboration

3. 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.

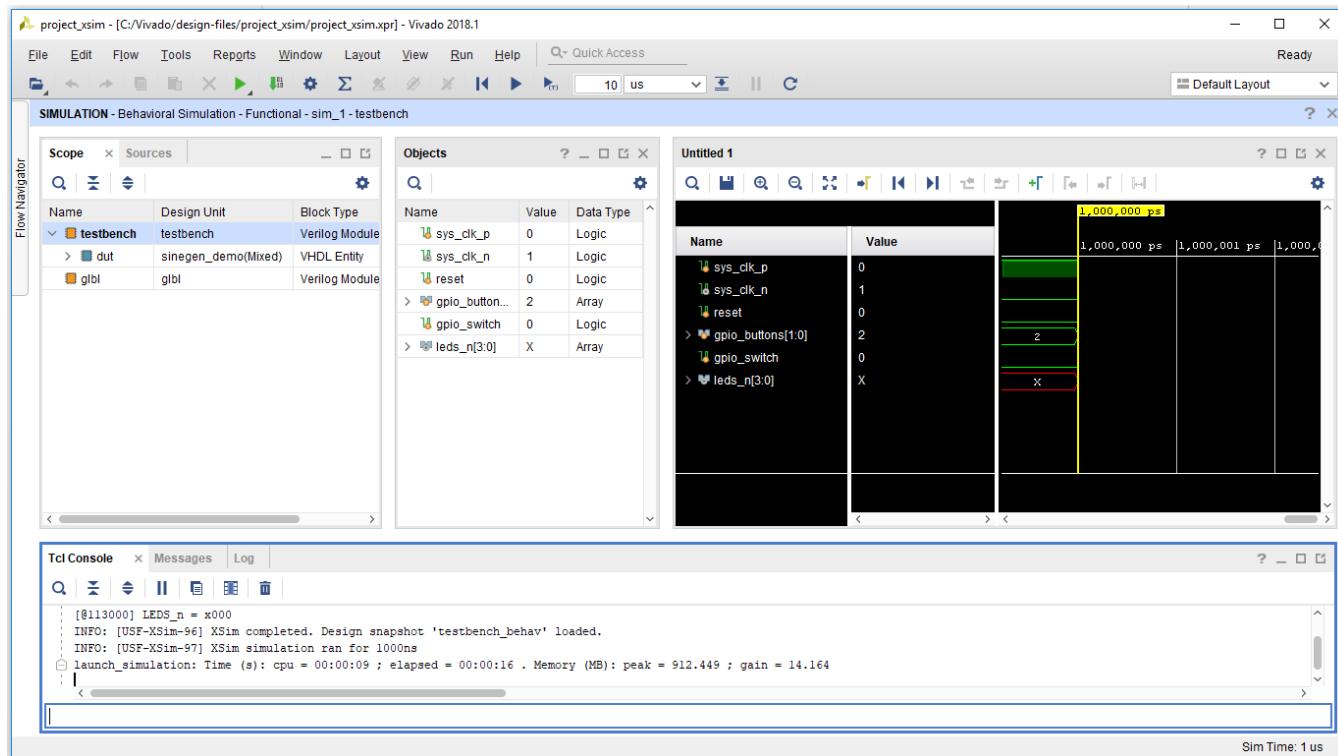


Figure 15: Vivado Simulation GUI

Running the Vivado Simulator

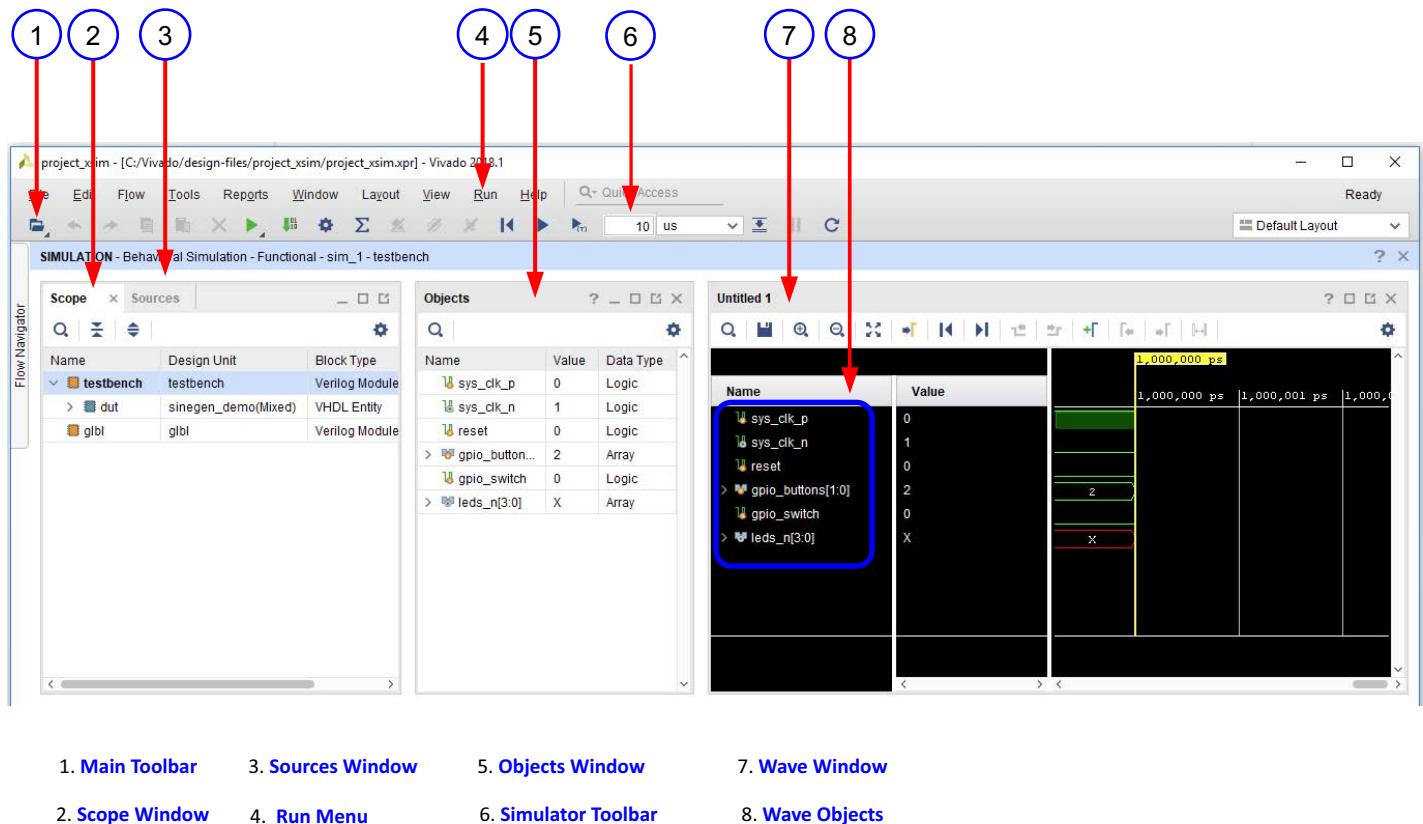


Figure 4-1: Vivado Simulator Workspace

Main Toolbar

The main toolbar provides one-click access to the most commonly used commands in the Vivado IDE. When you hover over an option, a tool tip appears that provides more information.

Run Menu

The menus provide the same options as the Vivado IDE with the addition of a Run menu after you have run a simulation.

The Run menu for simulation is shown in [Figure 4-2](#).

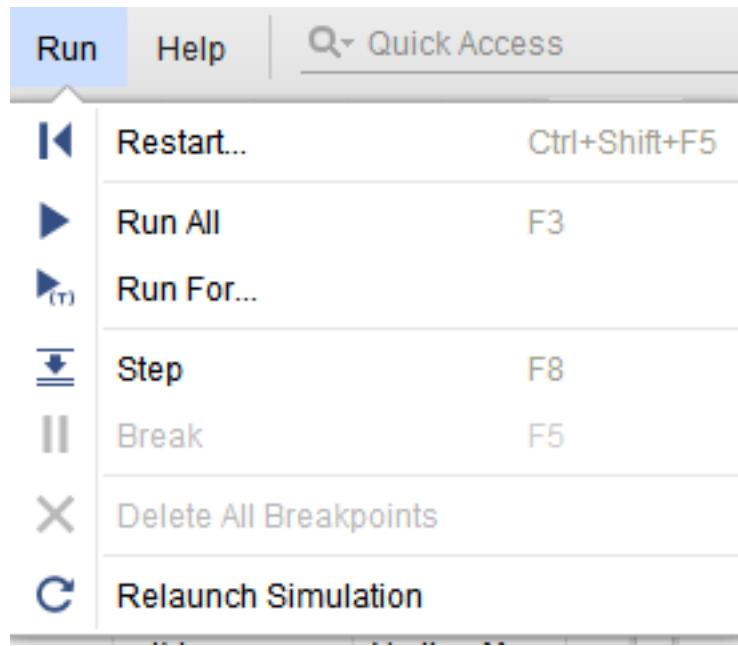


Figure 4-2: Simulation Run Menu Options

The Vivado simulator Run menu options:

- **Restart:** Lets you restart an existing simulation from time 0.
Tcl Command: `restart`
- **Run All:** Lets you run an open simulation to completion.
Tcl Command: `run -all`
- **Run For:** Lets you specify a time for the simulation to run.
Tcl Command: `run <time>`



TIP: While you can always specify time units in the run command such as `run 100 ns`, you can also omit the time unit. If you omit the time unit, the Vivado Simulator will assume the time unit of the `TIME_UNIT` Tcl property. To view the `TIME_UNIT` property use the Tcl command `get_property time_unit [current_sim]`. To change the `TIME_UNIT` property use the Tcl command `set_property time_unit <unit> [current_sim]`, where `<unit>` is one of the following: `fs`, `ps`, `ns`, `us`, `ms`, and `s`.

- **Step:** Runs the simulation up to the next HDL source line.
- **Break:** Lets you pause a running simulation.
- **Delete All Breakpoints:** Deletes all breakpoints.
- **Relaunch Simulation:** Recompiles the simulation files and restarts the simulation. See [Re-running the Simulation After Design Changes \(relaunch\)](#) for more information.

Simulation Toolbar

When you run the Vivado simulator, the simulation-specific toolbar (shown in the figure below) opens to the right of the main toolbar.



Figure 4-3: **Simulation Toolbar**

These are the same buttons labeled in [Figure 4-2, page 50](#), above (without the Delete All Breakpoints option), and they are provided for ease of use.

Simulation Toolbar Button Descriptions

Hover over the toolbar buttons for tool-tip descriptions.

- **Restart:** resets the simulation time to zero.
- **Run all:** runs the simulation until it completes all events or until an HDL statement indicates that the simulation should stop.
- **Run For:** runs for a specified period of time.
- **Step:** runs the simulation until the next HDL statement.
- **Break:** Pauses the current simulation.
- **Relaunch:** Recompiles the simulation sources and restarts the simulation (after making code changes, for example). See [Re-running the Simulation After Design Changes \(relaunch\)](#) for more information.

Sources Window

The Sources window displays the simulation sources in a hierarchical tree, with views that show Hierarchy, IP Sources, Libraries, and Compile Order, as shown in [Figure 4-4](#).

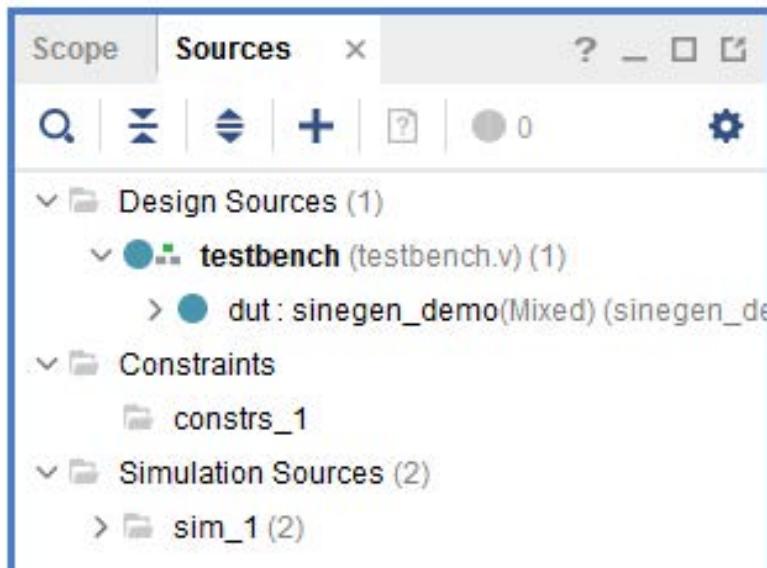


Figure 4-4: Sources Window

The Sources buttons are described by tool tips when you hover the mouse over them. The buttons let you examine, expand, collapse, add to, open, filter and scroll through files.

You can also open or add a source file by right-clicking on the source object and selecting the **Open File** or **Add Sources** options.

Scopes Window

A scope is a hierarchical partition of an HDL design. Whenever you instantiate a design unit or define a process, block, package, or subprogram, you create a scope.

In the scopes window (shown in the figure below), you can see the design hierarchy. When you select a scope in the Scopes hierarchy, all HDL objects visible from that scope appear in the Objects window. You can select HDL objects in the Objects window and add them to the waveform viewer.

Name	Design U...	Block Type
testbench	testbench	Verilog Mo...
dut	sinegen...	VHDL Entity
U_DEBOU...	debounc...	VHDL Entity
U_DEBOU...	debounc...	VHDL Entity
U_SINEGEN	sinegen(...)	VHDL Entity
U_SH	sine_hig...	VHDL Entity
U_SM	sine_mi...	VHDL Entity
U_SL	sine_low...	VHDL Entity
U_FSM	fsm(Mixed)	VHDL Entity
glbl	lbl	Verilog Mo...

Figure 4-5: Scopes Window

Filtering Scopes

- Click **Settings** option on the scopes sub-menu to toggle between showing or hiding (check or uncheck) the corresponding scope type.



TIP: When you hide a scope using Setting option, all scopes inside that scope are also hidden regardless of type. For example, in the figure above, clicking the Verilog Module button to hide all Verilog module scopes would hide not only the `bft_tb` scope but also `uut` (even though `uut` is a VHDL entity scope).

- To limit the display to scopes containing a specified string, click the **Search** button  and type the string in the text box.

The objects displayed in the Objects window change (or are filtered) based on the current scope. Select the current scope to change the objects in the Objects window.

Objects Window

The Objects window displays the HDL simulation objects associated with the scope selected in the Scopes window, as shown in [Figure 4-9](#).

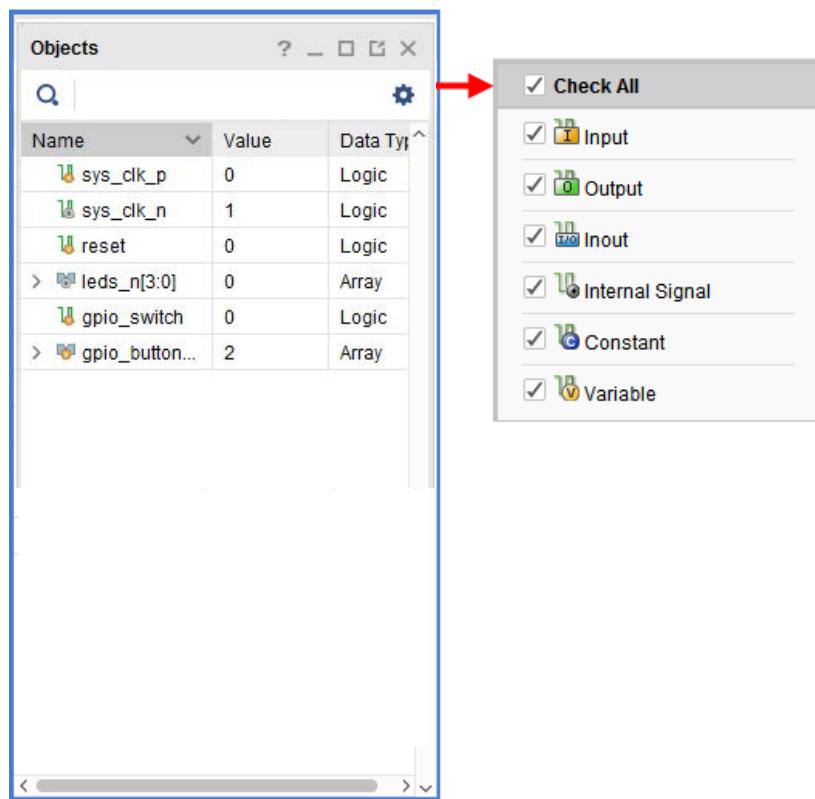


Figure 4-9: Objects Window

Icons beside the HDL objects show the type or port mode of each object. This view lists the Name, Value, and Data Type of the simulation objects.

You can obtain the current value of an object by typing the following in the Tcl Console.

```
get_value <hdl_object>
```



TIP: To limit the number of digits to display for vectors, use the `set_property array_display_limit <bits> [current_sim]` command, where `<bits>` is the number of bits to display



TIP: If you notice that some HDL objects do not appear in the Waveform Viewer, it is because Vivado simulator does not support waveform tracing of some HDL objects, such as named events in Verilog and local variables.

Wave Window

When you invoke the simulator it opens a Wave window by default. The Wave window displays a new wave configuration consisting of the traceable HDL objects from the top module of the simulation, as shown in [Figure 4-11](#).



TIP: On closing and reopening a project, you must rerun simulation to view the Wave window. If, however, you unintentionally close the default Wave window while a simulation is active, you can restore it by selecting **Window > Waveform** from the main menu.

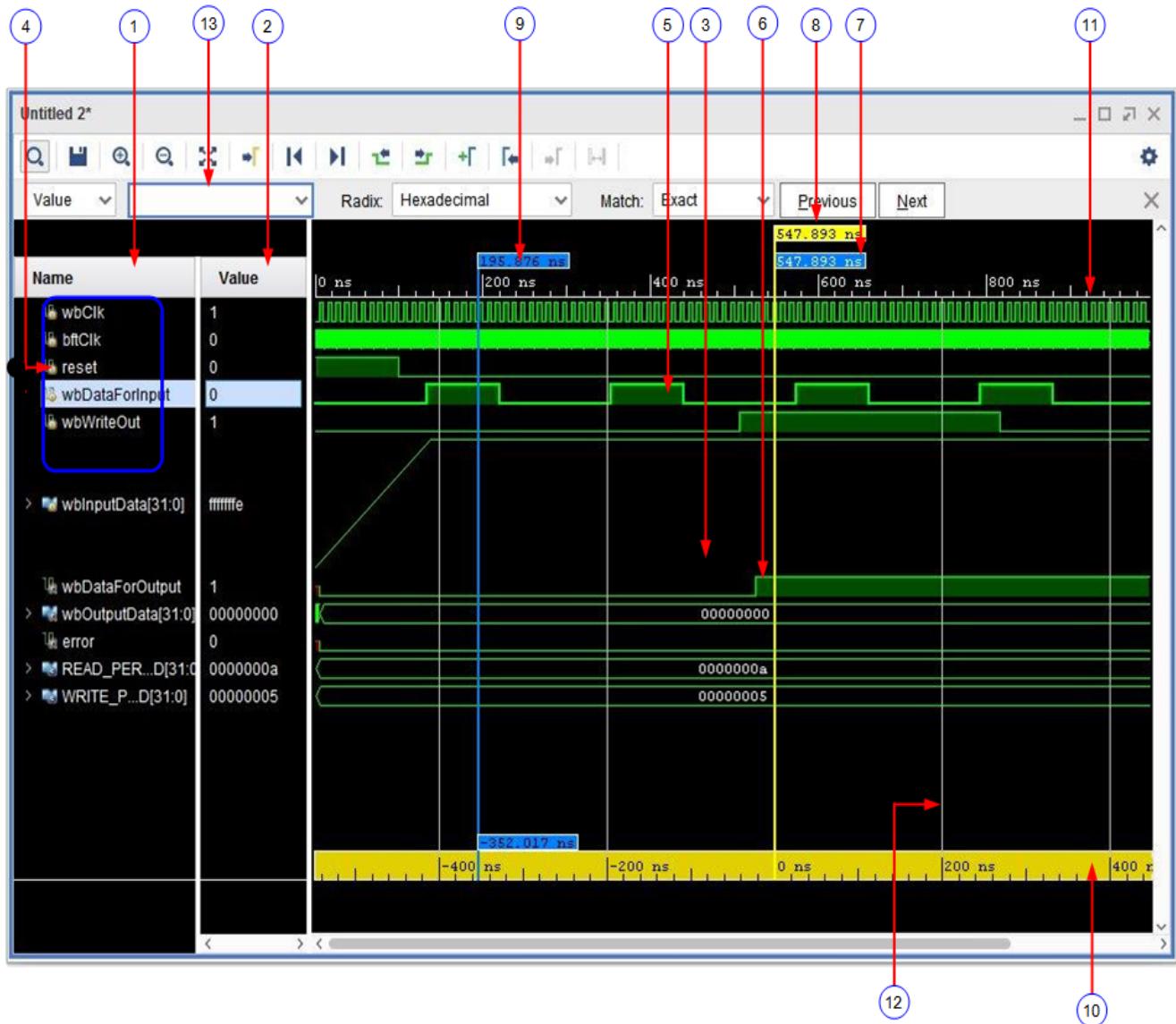


Figure 4-11: Wave Window

- | | | |
|--------------------|-----------------|----------------------|
| 1. Name Column | 2. Value Column | 3. Waveform Area |
| 4. Wave Object | 5. Waveform | 6. Transition |
| 7. Selected Marker | 8. Main Cursor | 9. Unselected Marker |
| 10. Floating Ruler | 11. Time Scale | 12. Grid Line |
| 13. Find Toolbar | | |

To add an individual HDL object or set of objects to the Wave window: in the Objects window, right-click an object or objects and select the **Add to Wave Window** option from the context menu (shown in [Figure 4-9, page 58](#)).

To add an object using the Tcl command type: `add_wave <HDL_objects>`.

Using the `add_wave` command, you can specify full or relative paths to HDL objects.

For example, if the current scope is `/bft_tb/uut`, the full path to the reset register under `uut` is `/bft_tb/uut/reset`: the relative path is `reset`.

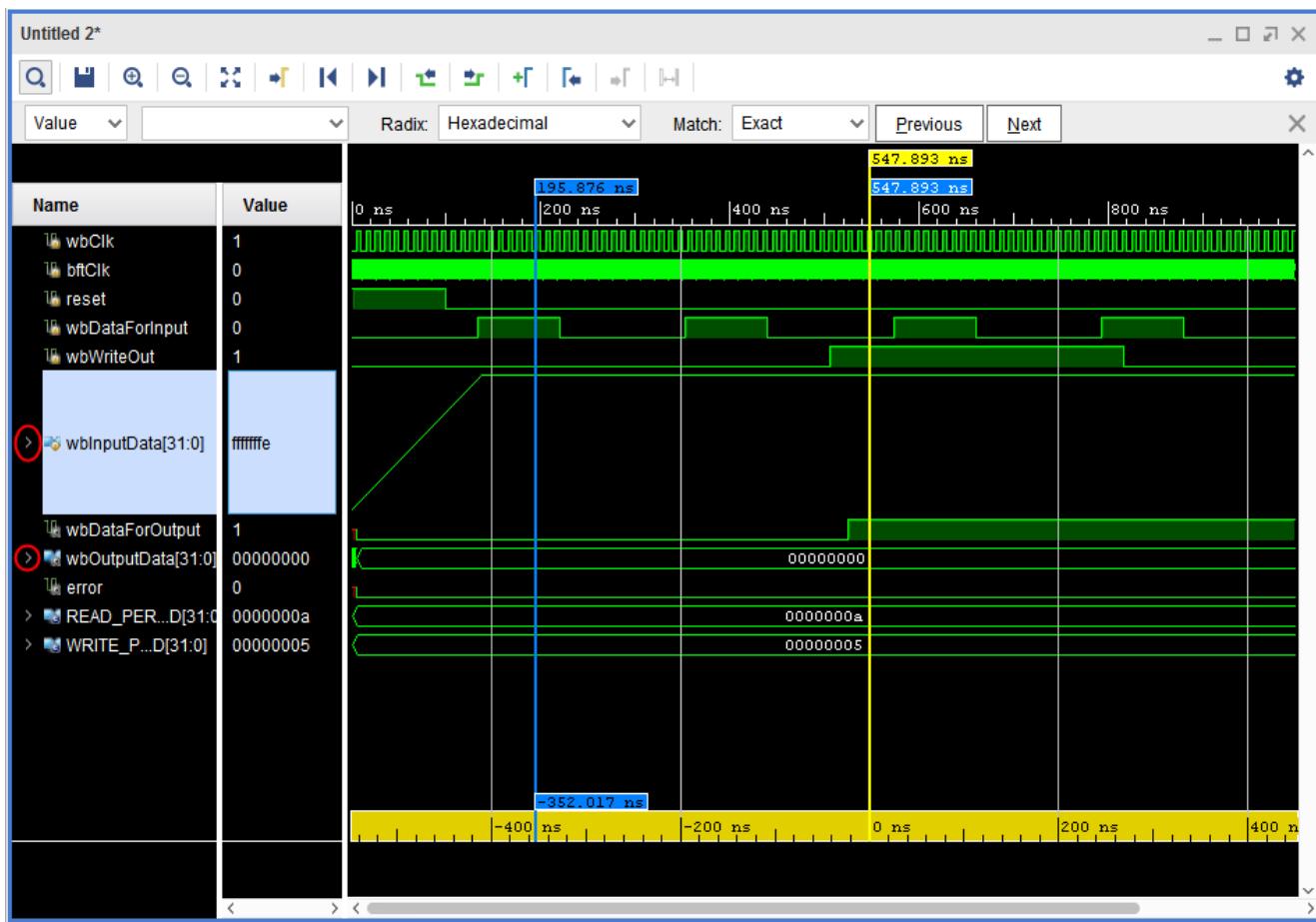
TIP:

*The `add_wave` command accepts HDL scopes as well as HDL objects. Using `add_wave` with a scope is equivalent to the **Add To Wave Window** command in the Scopes window.*

HDL objects of large bit width can slow down the display of the waveform viewer. You can filter out such objects by setting a “display limit” on the wave configuration before issuing the Add to Wave Window command. To set a display limit, use the Tcl command `set_property DISPLAY_LIMIT <maximum bit width> [current_wave_config]`.

Wave Objects

The Vivado IDE Wave window is common across a number of Vivado Design Suite tools. An example of the wave objects in a waveform configuration is shown in [Figure 4-12](#).



The Wave window displays HDL objects, their values, and their waveforms, together with items for organizing the HDL objects, such as: groups, dividers, and virtual buses.

Collectively, the HDL objects and organizational items are called a *wave configuration*. The waveform portion of the Wave window displays additional items for time measurement, that include: cursors, markers, and timescale rulers.

The Vivado IDE traces the value changes of the HDL object in the Wave window during simulation, and you use the wave configuration to examine the simulation results.

The design hierarchy and the simulation waveforms are not part of the wave configuration, and are stored in a separate wave database (WDB) file.



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.

Lab 2: Debugging the Design

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 2018.x > Vivado 2018.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 2018.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_xsims`, 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.

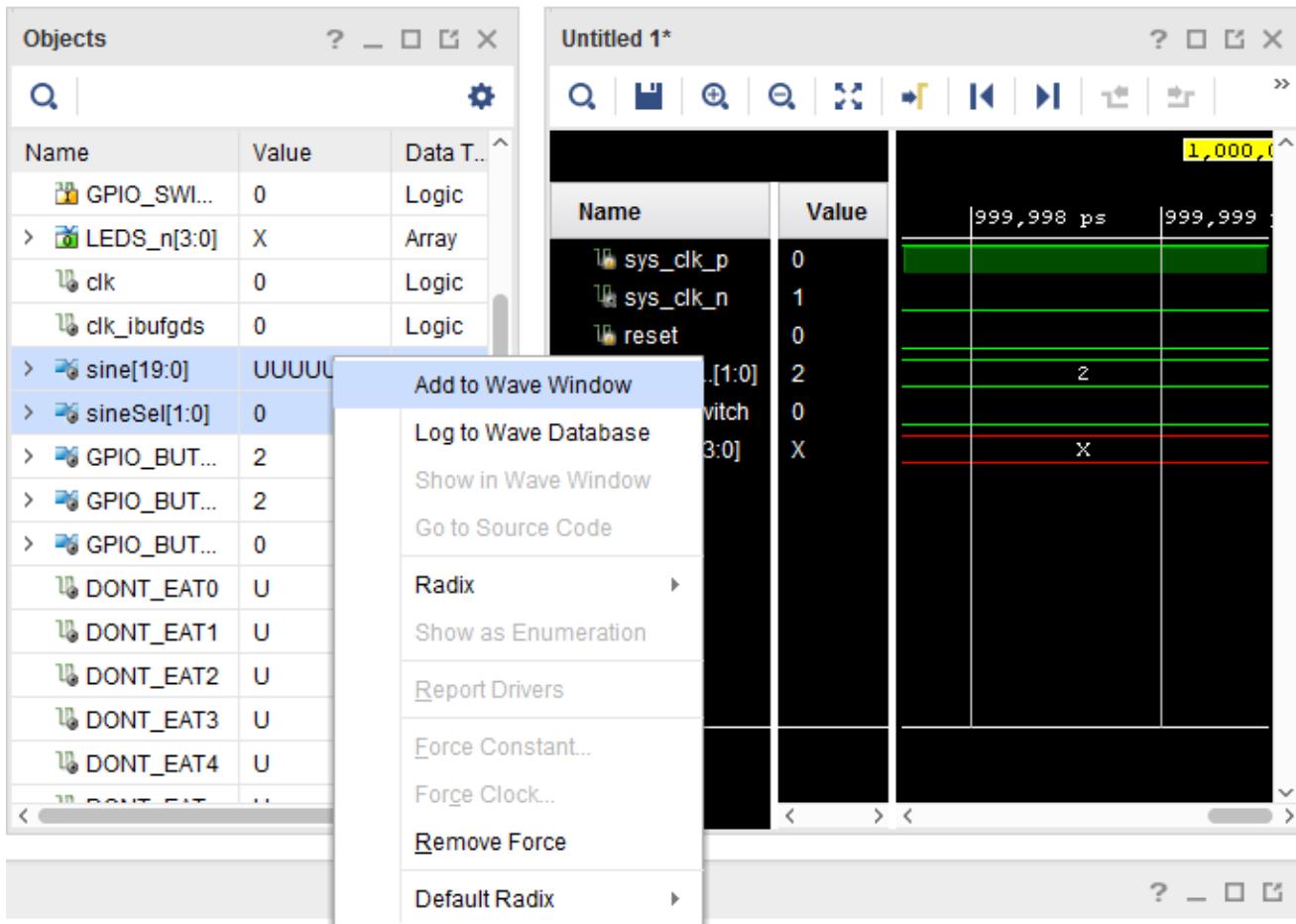


Figure 16: Add signals to Wave Window

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](#).

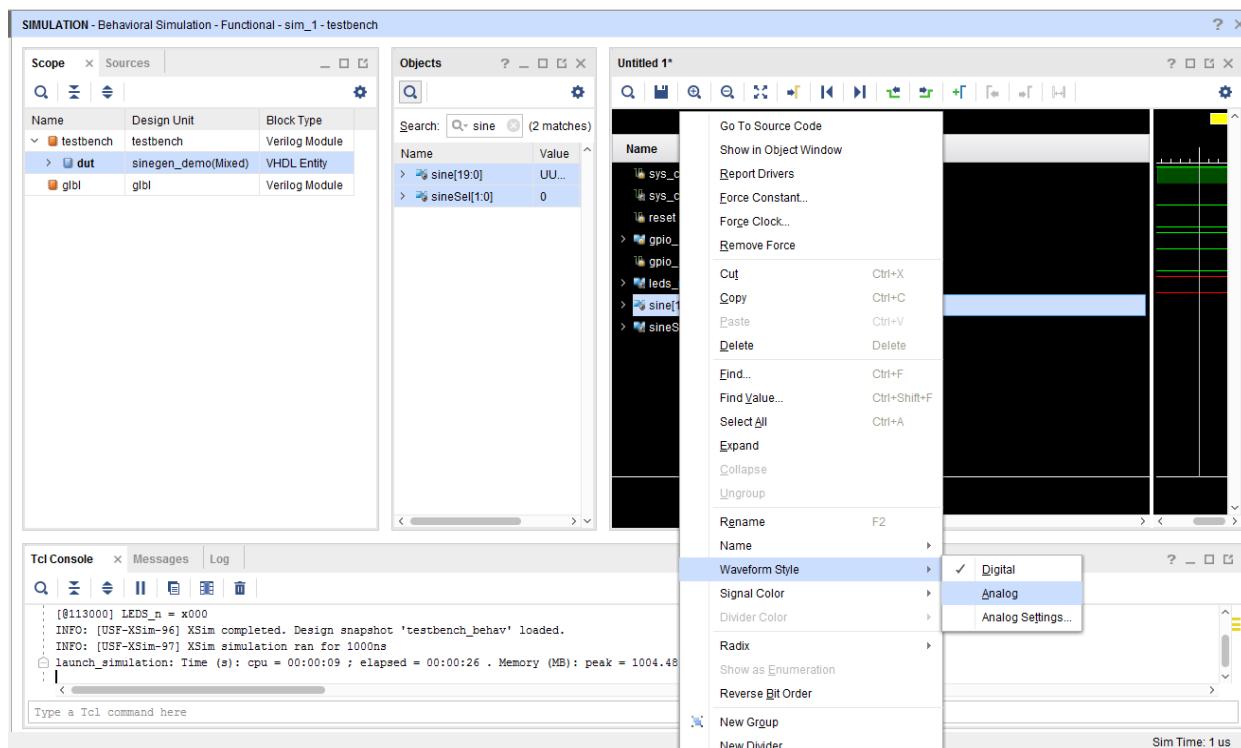


Figure 17: Enable Analog Waveform Style

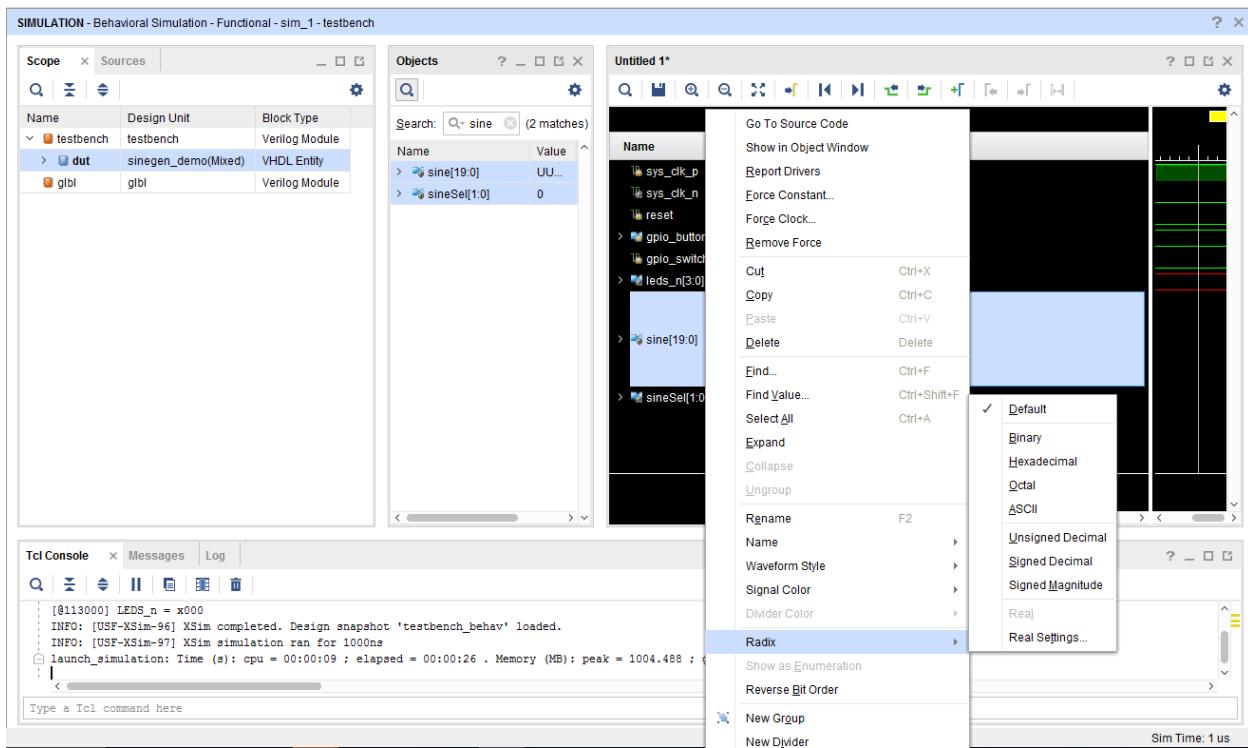


Figure 18: Enable Radix

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/behave
```

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.

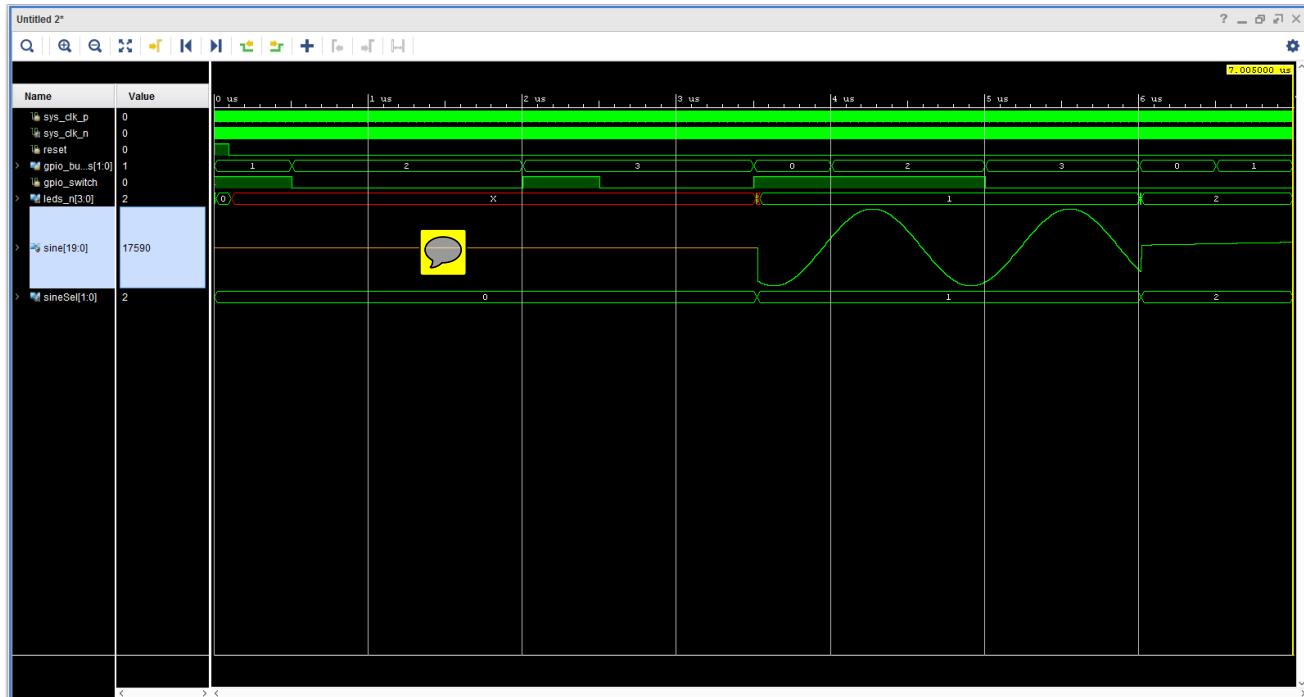


Figure 19: Design Bug - Wave View

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.

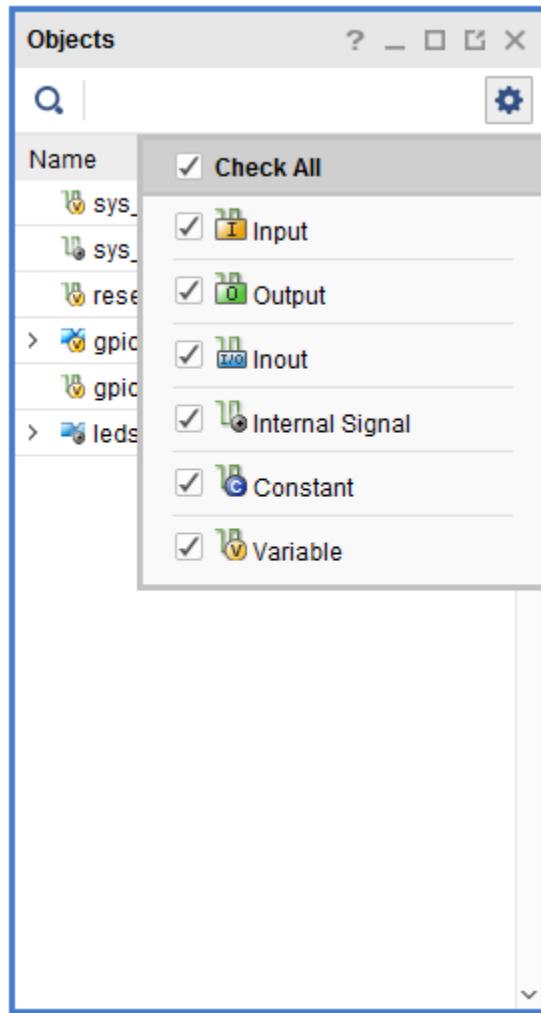


Figure 20: Object Filters

2. 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.

3. In the Scopes window, select the `U_SINEGEN` design unit.
4. 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.

-
5. In the Objects window, select the Input filter to display the Input objects.
 6. 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**.

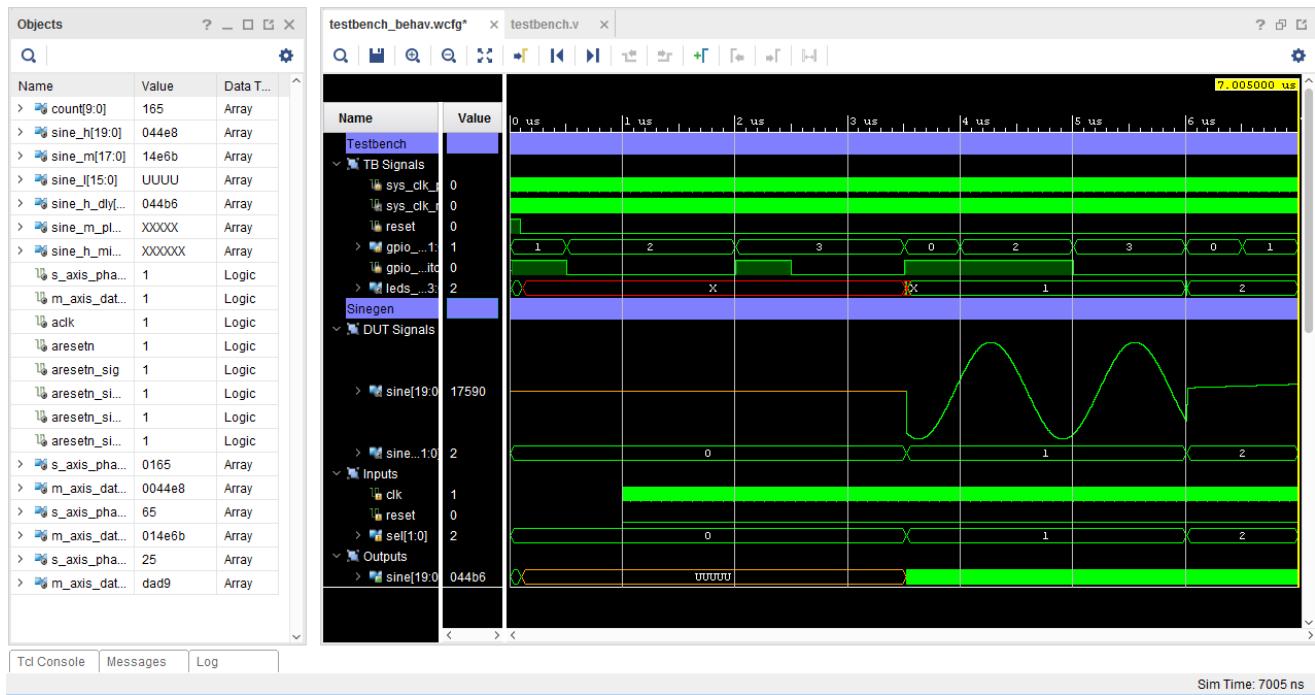


Figure 21: Configuring the Wave Window



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.

- 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.
- Hold **Ctrl**, and click to select the `sine[19:0]` signals listed in the `Outputs` group, under the `SineGen` divider.
- 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](#).

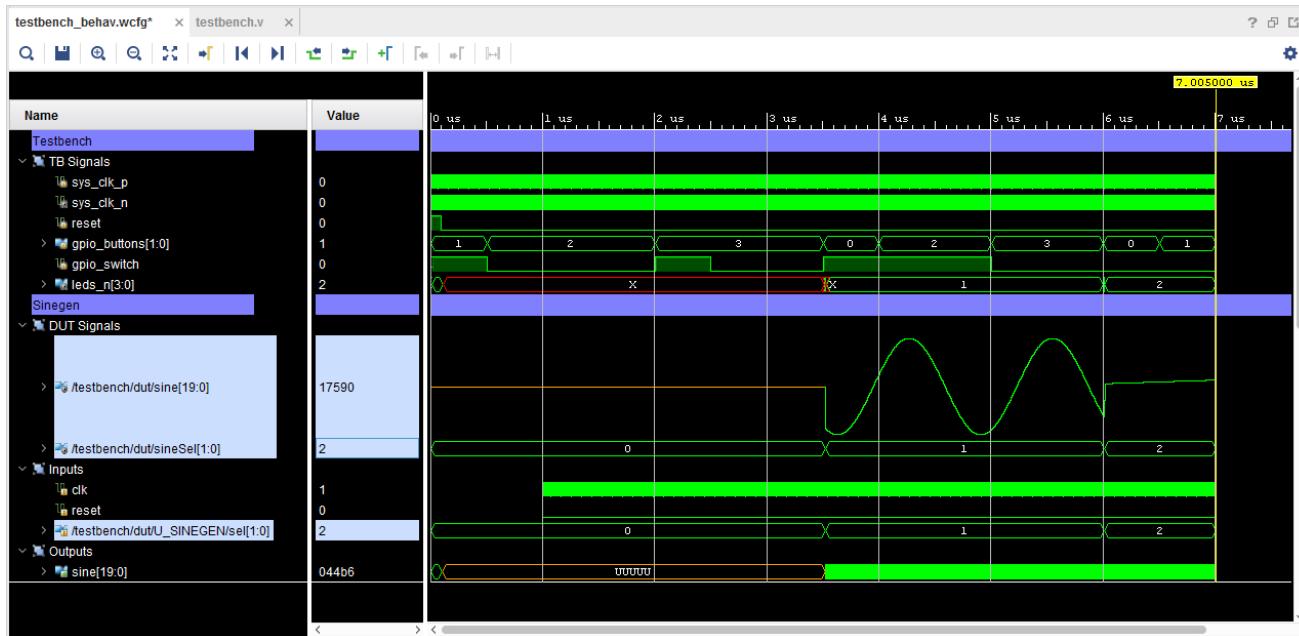


Figure 22: Long Signal Names

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.

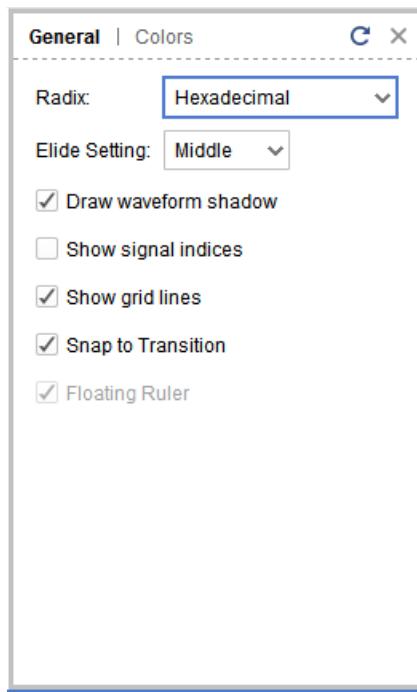


Figure 23: Waveform Options - General View

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 files set, `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.

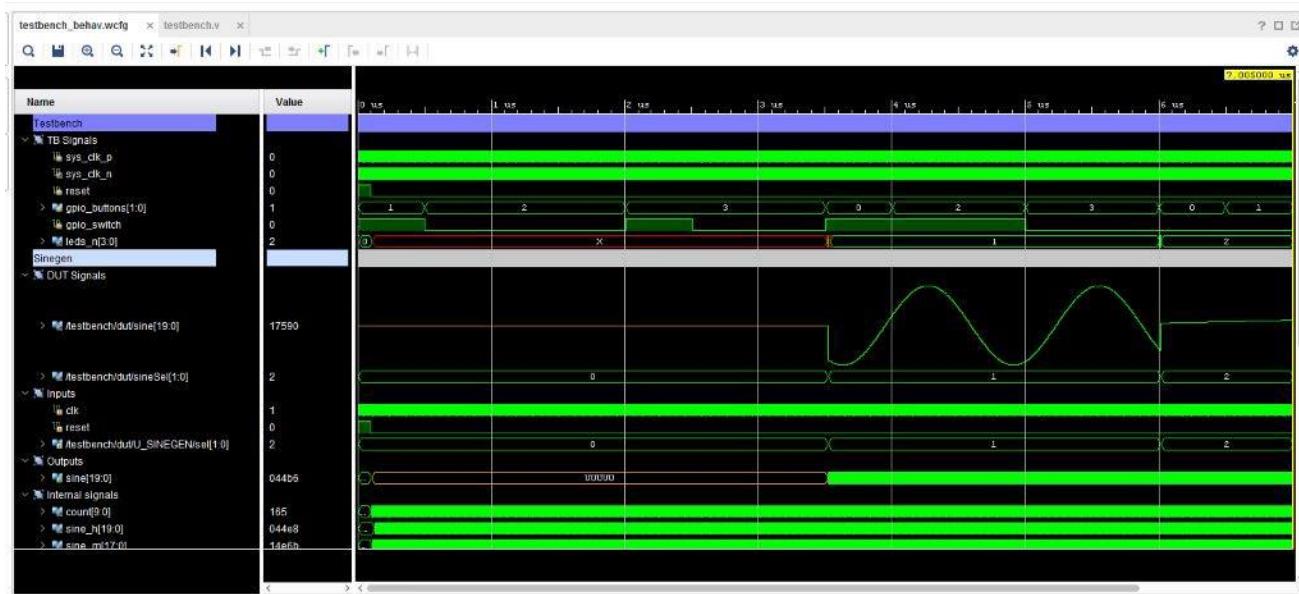


Figure 24: Simulation Waveform at Time 7005 ns

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 `sineSel` 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.

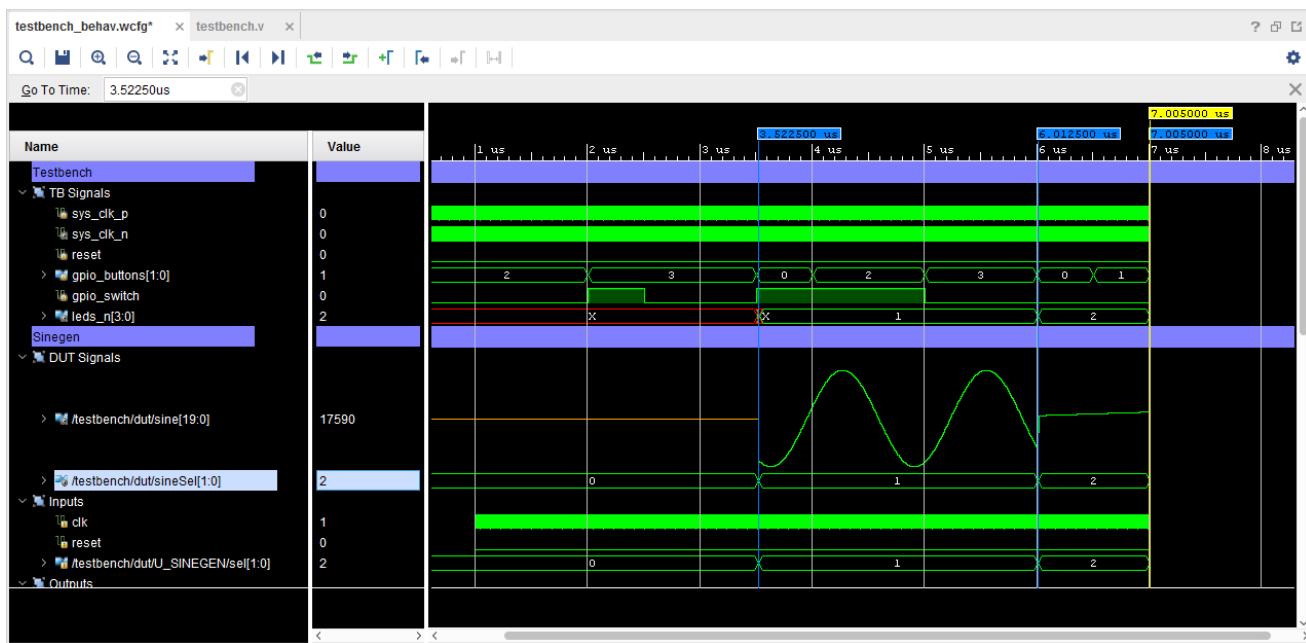


Figure 25: Using Markers



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](#).

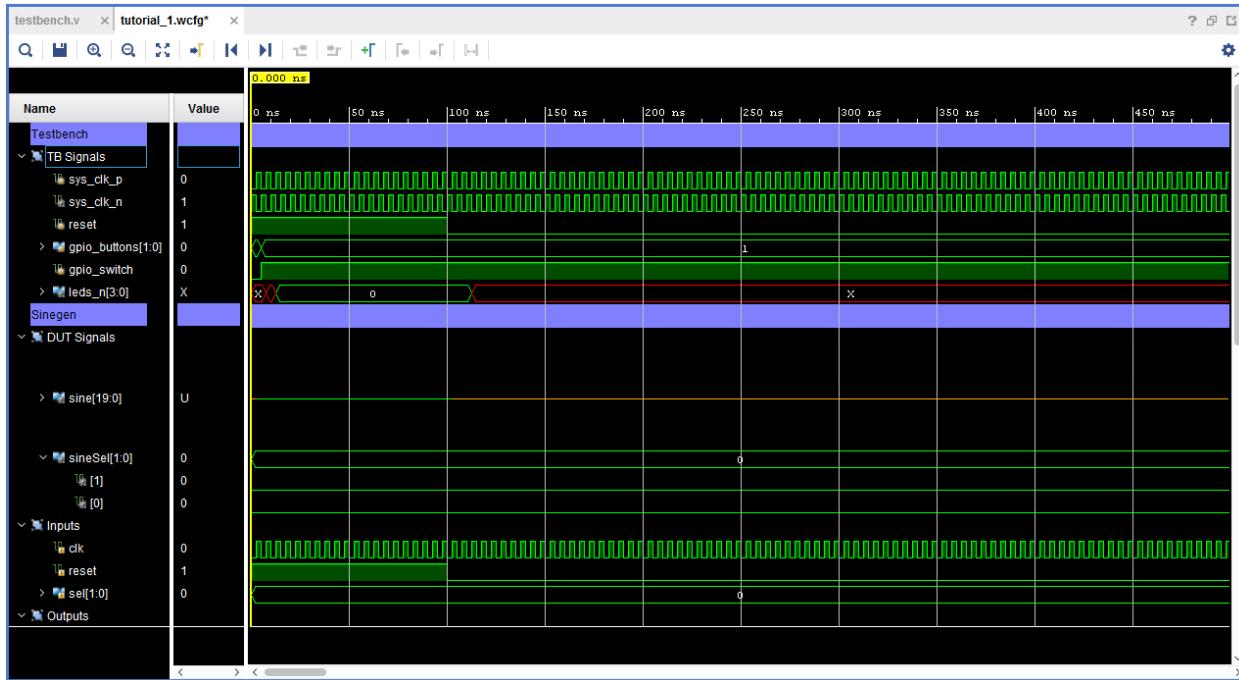


Figure 26: Viewing Reset and Clock Signals

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

9. 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.

10. 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.

11. Place a marker at the time of interest, 102.5 ns, by clicking the **Add Marker** button .

12. 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.*

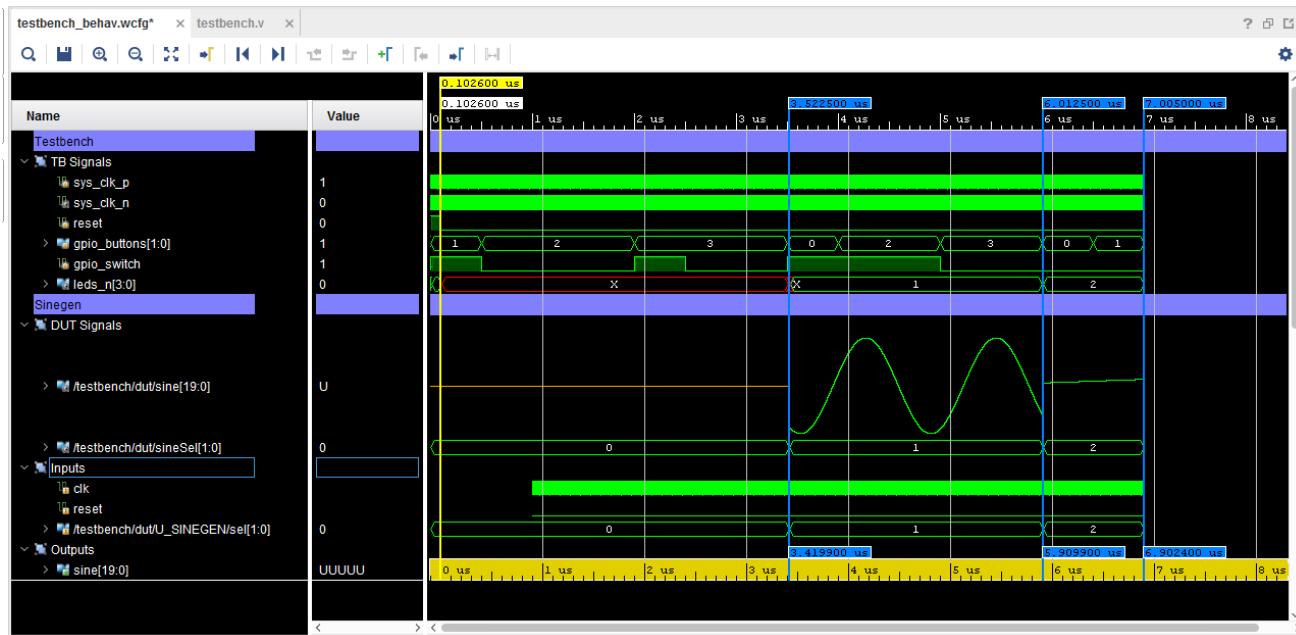


Figure 27: Measuring Time in the Waveform



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.

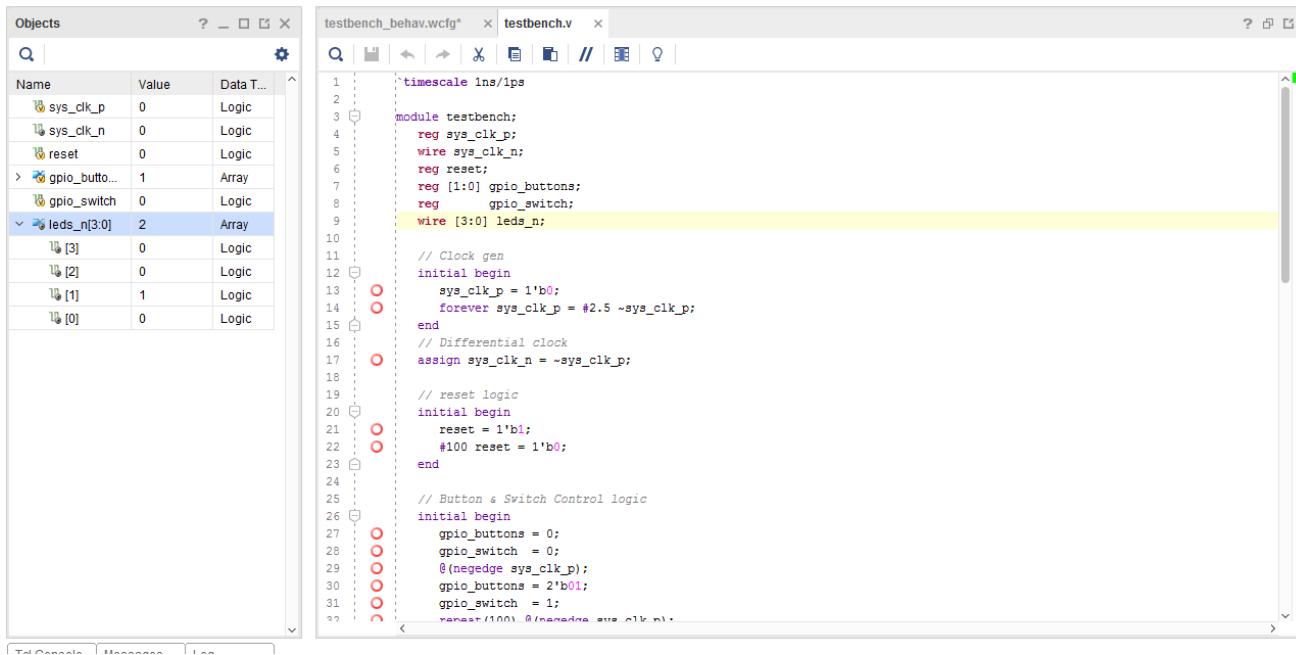
As shown in [Figure 27](#), the ruler measures a time period of 3.420 ns as the period that FSM selected the low frequency output.

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](#).



```

1  `timescale 1ns/1ps
2
3  module testbench;
4      reg sys_clk_p;
5      wire sys_clk_n;
6      reg reset;
7      reg [1:0] gpio_buttons;
8      reg        gpio_switch;
9      wire [3:0] leds_n;
10
11     // Clock begin
12     initial begin
13         sys_clk_p = 1'b0;
14         forever sys_clk_p = #2.5 ~sys_clk_p;
15     end
16     // Differential clock
17     assign sys_clk_n = ~sys_clk_p;
18
19     // reset logic
20     initial begin
21         reset = 1'b1;
22         #100 reset = 1'b0;
23     end
24
25     // Button & Switch Control logic
26     initial begin
27         gpio_buttons = 0;
28         gpio_switch = 0;
29         @(negedge sys_clk_p);
30         gpio_buttons = 2'b01;
31         gpio_switch = 1;
32         #100 @(negedge sys_clk_p);
33     end

```

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.

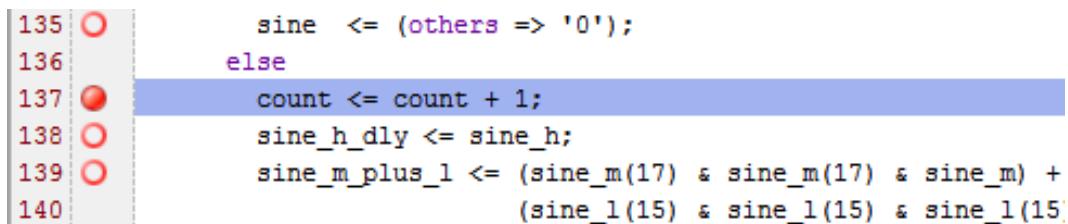
3. Scroll to line 137 in the file.

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
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.

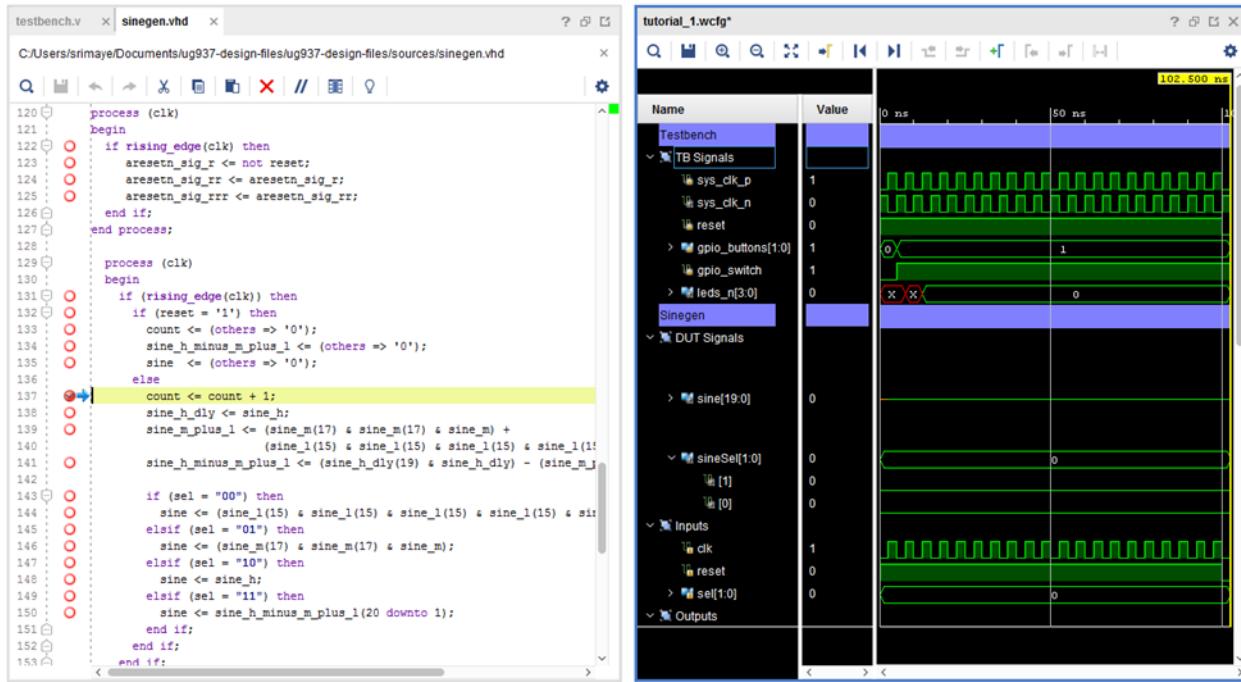


Figure 30: Arrange Windows for Debugging



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_1[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_1`, to the output, `sine`.

10. In the Waveform window, select `sine_1[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.vhd` 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**.

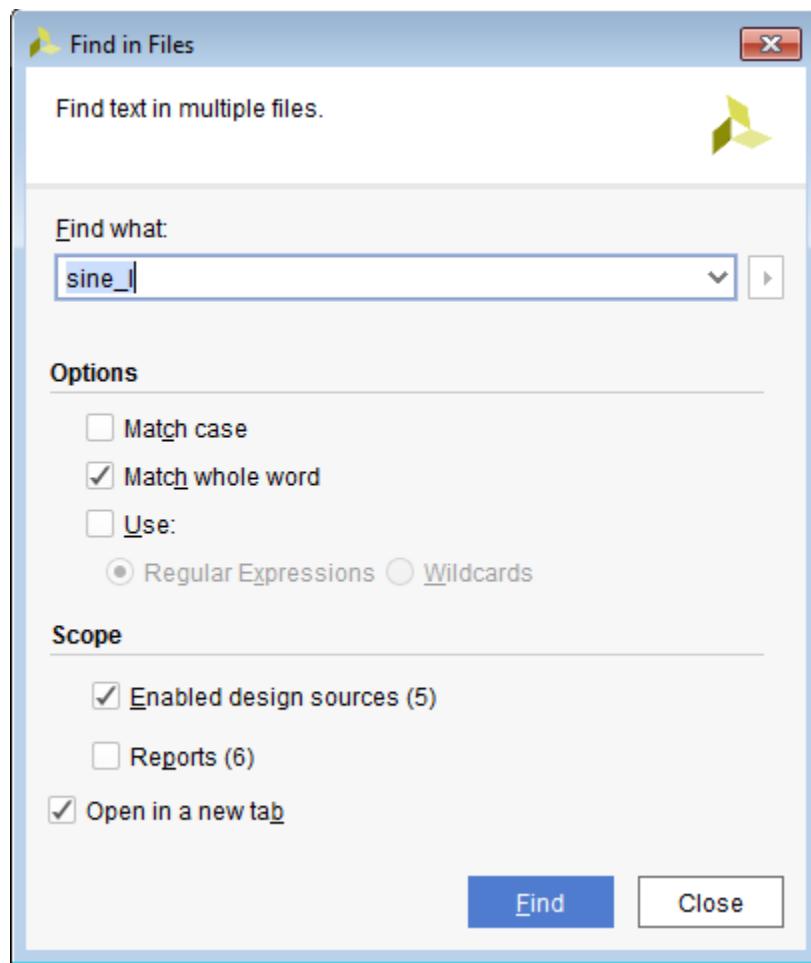


Figure 31: Find in Files

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.

7. 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.

8. Uncomment line 111 in the `sinegen.vhd` file, and click the **Save File** button 
9. 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
```

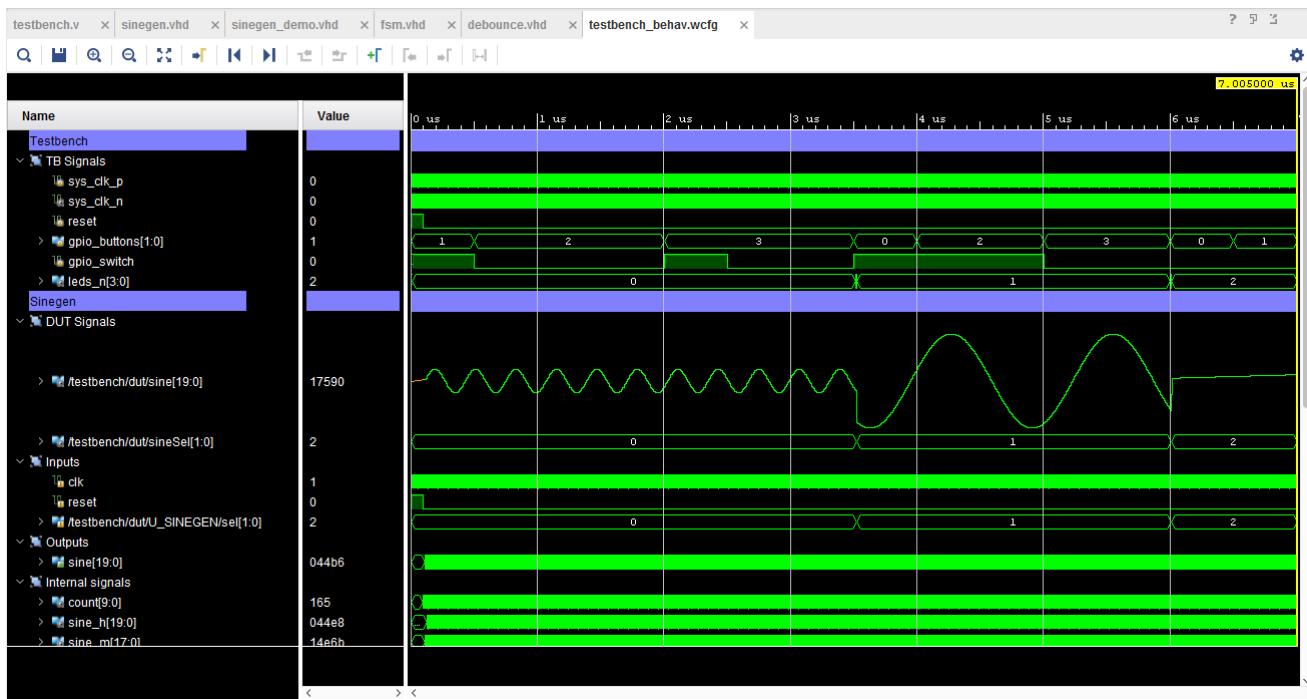


Figure 32: Corrected Low Frequency Output

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