

ISE Tutorial

Using Xilinx ChipScope Pro ILA Core with Project Navigator to Debug FPGA Applications

UG750 (v14.3) October 16, 2012



Xilinx is disclosing this user guide, manual, release note, and/or specification (the “Documentation”) to you solely for use in the development of designs to operate with Xilinx® hardware devices. You may not reproduce, distribute, republish, download, display, post, or transmit the Documentation in any form or by any means including, but not limited to, electronic, mechanical, photocopying, recording, or otherwise, without the prior written consent of Xilinx. Xilinx expressly disclaims any liability arising out of your use of the Documentation. Xilinx reserves the right, at its sole discretion, to change the Documentation without notice at any time. Xilinx assumes no obligation to correct any errors contained in the Documentation, or to advise you of any corrections or updates. Xilinx expressly disclaims any liability in connection with technical support or assistance that may be provided to you in connection with the Information.

THE DOCUMENTATION IS DISCLOSED TO YOU “AS-IS” WITH NO WARRANTY OF ANY KIND. XILINX MAKES NO OTHER WARRANTIES, WHETHER EXPRESS, IMPLIED, OR STATUTORY, REGARDING THE DOCUMENTATION, INCLUDING ANY WARRANTIES OF MERCHANTABILITY, FITNESS FOR A PARTICULAR PURPOSE, OR NONINFRINGEMENT OF THIRD-PARTY RIGHTS. IN NO EVENT WILL XILINX BE LIABLE FOR ANY CONSEQUENTIAL, INDIRECT, EXEMPLARY, SPECIAL, OR INCIDENTAL DAMAGES, INCLUDING ANY LOSS OF DATA OR LOST PROFITS, ARISING FROM YOUR USE OF THE DOCUMENTATION.

IN NO EVENT WILL XILINX BE LIABLE FOR ANY CONSEQUENTIAL, INDIRECT, EXEMPLARY, SPECIAL, OR INCIDENTAL DAMAGES, INCLUDING ANY LOST DATA AND LOST PROFITS, ARISING FROM OR RELATING TO YOUR USE OF THE DESIGN, EVEN IF YOU HAVE BEEN ADVISED OF THE POSSIBILITY OF SUCH DAMAGES. THE TOTAL CUMULATIVE LIABILITY OF XILINX IN CONNECTION WITH YOUR USE OF THE DESIGN, WHETHER IN CONTRACT OR TORT OR OTHERWISE, WILL IN NO EVENT EXCEED THE AMOUNT OF FEES PAID BY YOU TO XILINX HEREUNDER FOR USE OF THE DESIGN. YOU ACKNOWLEDGE THAT THE FEES, IF ANY, REFLECT THE ALLOCATION OF RISK SET FORTH IN THIS AGREEMENT AND THAT XILINX WOULD NOT MAKE AVAILABLE THE DESIGN TO YOU WITHOUT THESE LIMITATIONS OF LIABILITY.

The Design is not designed or intended for use in the development of on-line control equipment in hazardous environments requiring fail-safe controls, such as in the operation of nuclear facilities, aircraft navigation or communications systems, air traffic control, life support, or weapons systems (“High-Risk Applications”) Xilinx specifically disclaims any express or implied warranties of fitness for such High-Risk Applications. You represent that use of the Design in such High-Risk Applications is fully at your risk.

© 2012 Xilinx, Inc. All rights reserved. XILINX, the Xilinx logo, and other designated brands included herein are trademarks of Xilinx, Inc. All other trademarks are the property of their respective owners.

Demo Design License

© 2012 Xilinx, Inc.

This Design is free software; you can redistribute it and/or modify it under the terms of the GNU Lesser General Public License as published by the Free Software Foundation; either version 2.1 of the License, or (at your option) any later version.

This library is distributed in the hope that it will be useful, but WITHOUT ANY WARRANTY; without even the implied warranty of MERCHANTABILITY or FITNESS FOR A PARTICULAR PURPOSE. See the GNU Lesser General Public License for more details.

You should have received a copy of the GNU Library General Public License along with this design file; if not, see:

<http://www.gnu.org/licenses/>

Revision History

The following table shows the revision history for this document.

Date	Revision
01/18/2012	Revalidated for the 13.4 release. Editorial updates only; no technical content updates.
04/24/2012	Revalidated for the 14.1 release. Editorial updates only; no technical content updates.
07/25/2012	Revalidated for the 14.2 release. Editorial updates only; no technical content updates.
10/16/2012	Revalidated for the 14.3 release. Updated Figure 2-3 for CR 657985. No other technical content updates.

Chapter 1: Introduction

Objectives	5
Lab Exercise Setup	6
Design Description	7
Push Button Switches	7
Debounce Circuit	8
Control State Machine	8
LED Displays	8
Files Provided for this Lab Exercise	9
Procedure	9

Chapter 2: Creating and Implementing a Project in Project Navigator

Creating and Implementing Your RTL Design	11
Questions	15

Chapter 3: Adding a ChipScope ILA Core to Your Design

Adding a ChipScope ILA Core	17
Questions	22

Chapter 4: Debugging Your Design using ChipScope Pro Analyzer

Debugging Your Design	23
Questions	32

Chapter 5: Tutorial Conclusion

Conclusion	33
Question Answers	33

Introduction

In this lab exercise, you will explore how an Integrated Logic Analyzer (ILA) core can be inserted within the Project Navigator design environment to debug your FPGA designs. You will take advantage of ChipScope™ Pro Analyzer functions to debug and discover some potential root causes of your design, thereby allowing you to address issues quickly as will be shown by this tutorial.

Example RTL designs will be used to illustrate overall integration flows between ChipScope and Xilinx® ISE® Project Navigator. In order to be successful using this tutorial, you should have some basic knowledge of ISE tool flows.

Objectives

After completing this lab exercise, you will be able to:

- Validate and debug your design using Project Navigator and ChipScope with ILA core and Analyzer
- Understand how to create an ISE project, probe an ILA core, and implement the design in Project Navigator
- Debug the design using ChipScope Analyzer and iterate the design using Project Navigator design environment and an SP601Platform

Lab Exercise Setup

The following software and hardware are required for this lab:

- Xilinx ISE Design Suite 12.3 (Logic, DSP, Embedded, or System Edition)
- SP601Platform
- JTAG Cables that come with the SP601 Platform

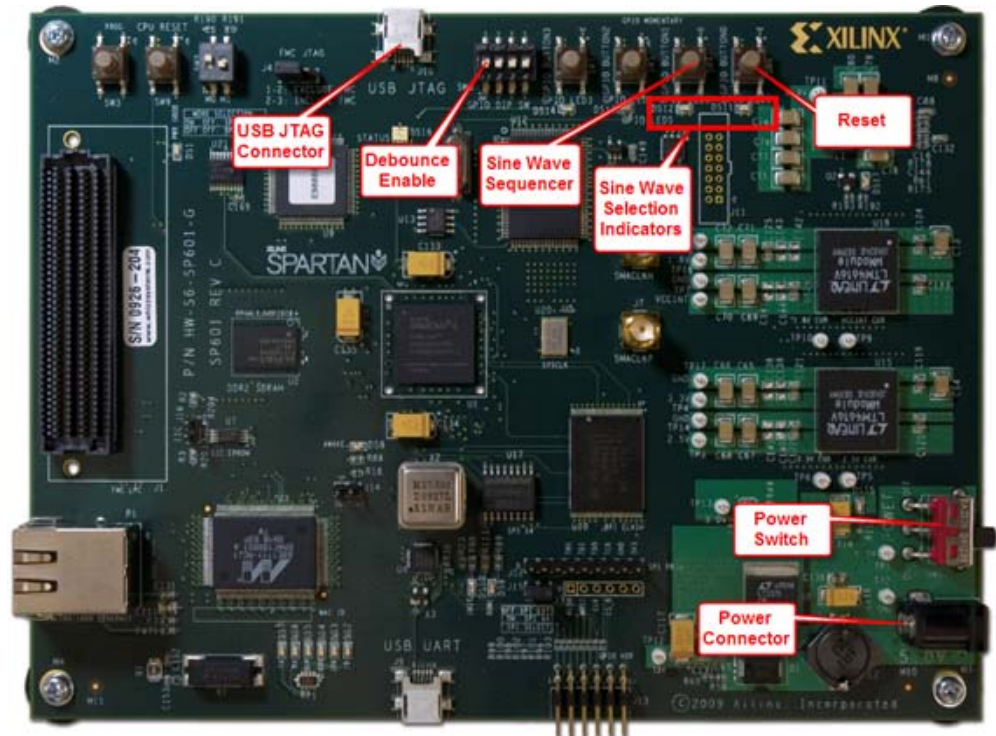


Figure 1-1: SP-601 Platform

Design Description

The top-level block diagram for the RTL example design is shown in the figure below. The design is comprised of a simple control state machine, multiple sine wave generators, common Push Button (GPIO_BUTTON), Dip Switch (GPIO_SWITCH), and LED displays (GPIO_LED).

Push Button Switches

Push Button Switches are used as inputs into the debounce and / or control state machine circuits. A high to low transition pulse is generated when a switch is pushed. Each generated output pulse is then used as an input into the state machine.

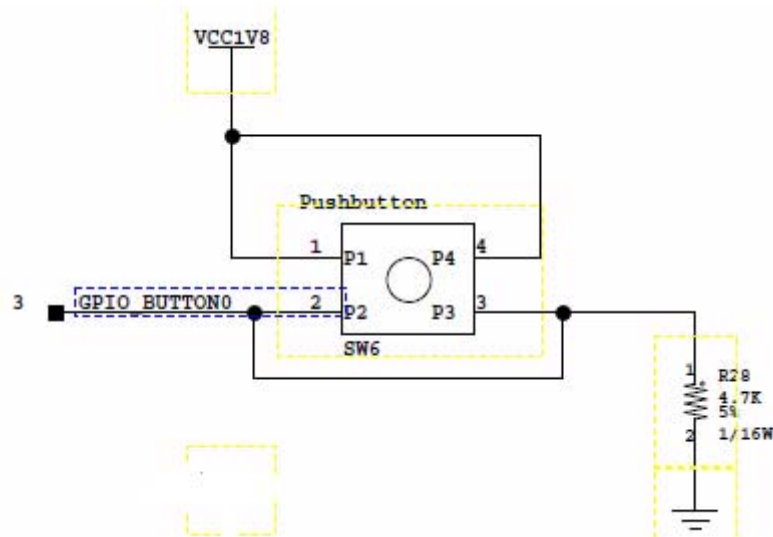


Figure 1-2: Dip Switch: Used to enable or disable a debounce circuit

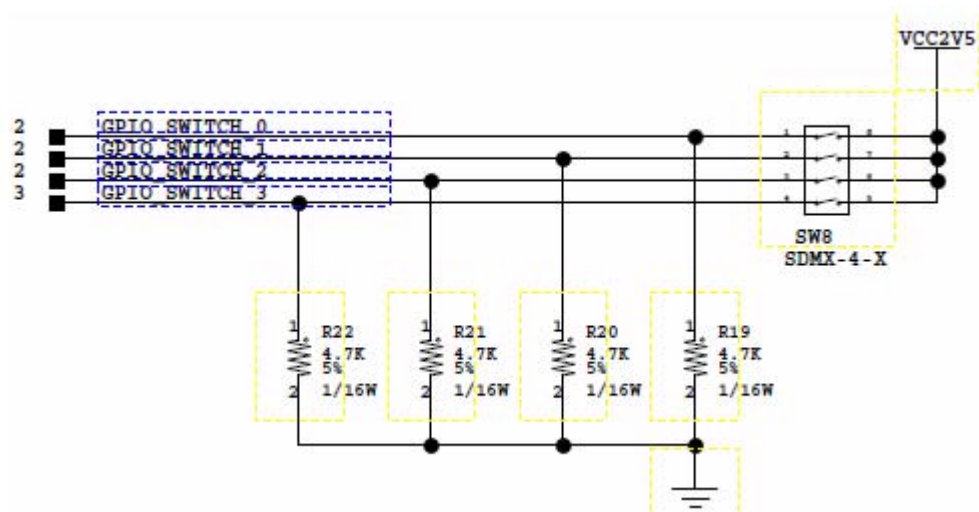


Figure 1-3: Debounce circuit

Debounce Circuit

When enabled, the debounce circuit provides a clean pulse or transition from a high to low on this particular example. It eliminates series of spikes or glitches when a button is pressed and released.

Control State Machine

The control state machine is used to capture and decode input pulses from the two Push Button switches.. It provides sine wave selection and indicator circuits, sequencing between 00, 01, 10, and 11 (zero to three).

LED Displays

GPIO_LED_0 and GPIO_LED_0 are used to display selection status from the state machine outputs, each of which represents a different sine wave frequency - high, medium, and low.

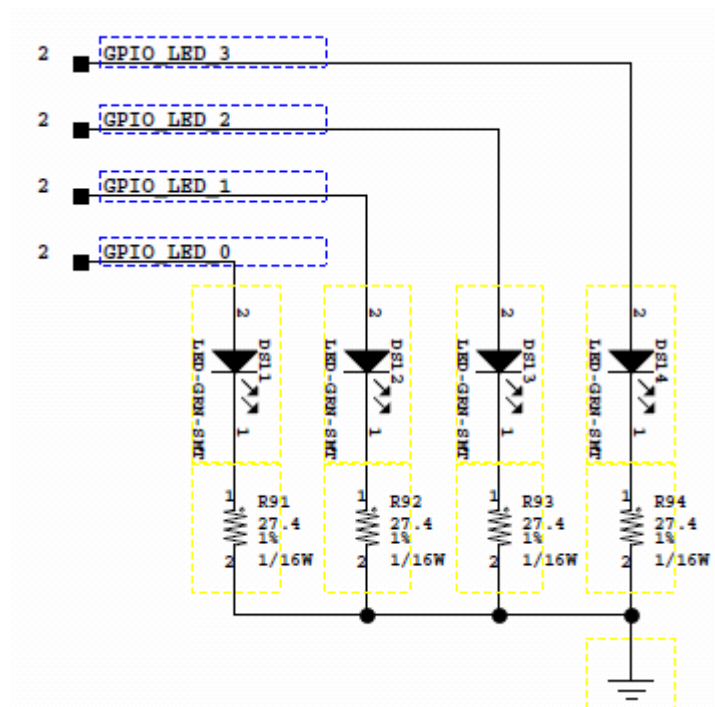


Figure 1-4: LED Displays

Files Provided for this Lab Exercise

The following files and subfolders are provided for this lab exercise:

- debounce.vhdl - Debounce circuit
- fsm.vhdl - Control state machine
- sinegen_demo - Wrapper for Sine wave generators
- sinewave_demo - Top-level wrapper design
- sine_high.xco, sine_mid.xco, sine_low.xco - Xilinx Core Generator files
- sinegen_demo_sp601.ucf - UCF constraint file

Note: This lab also supports other two Xilinx platforms - SP605 and ML605. Use pin-out information provided by the table below to retarget this tutorial to either SP605 or ML605.

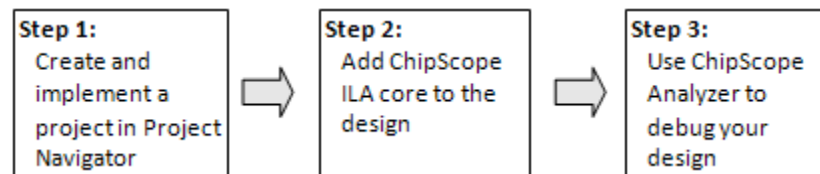
Table 1-1: Pin-Out Information for Xilinx Platforms

	Pin-Out Locations			Function
	SP601	SP605	ML605	
CLK_N	K16	K22	H9	Clock
CLK_P	K15	K21	J9	Clock
GPIO_BUTTONS0	P4	F3	A19	Reset
GOIP_BUTTONS1	F6	G6	G26	Sequencer
GPIO_SWITCH	D14	C18	D22	Debounce circuit selector
GPIO_LED_0	E13	D17	AC22	Selector[0]
GPIO_LED_0	C14	AB4	AC24	Selector[1]

Procedure

In this tutorial, you will complete three tasks:

1. [Creating and Implementing a Project in Project Navigator](#)
2. [Adding a ChipScope ILA Core to Your Design](#)
3. [Debugging Your Design using ChipScope Pro Analyzer](#)



Creating and Implementing a Project in Project Navigator

Creating and Implementing Your RTL Design

In this tutorial, you will explore how Project Navigator can be used to quickly implement your RTL design. You will learn how to create an ISE® project in Project Navigator targeting the SP601 Platform.

1. Unzip the provided source files in C:\ChipScope_ProjNav\
2. Start Project Navigator and select **File > New Project** to create a new project. Provide the name, location, and project type, and then click **Next**

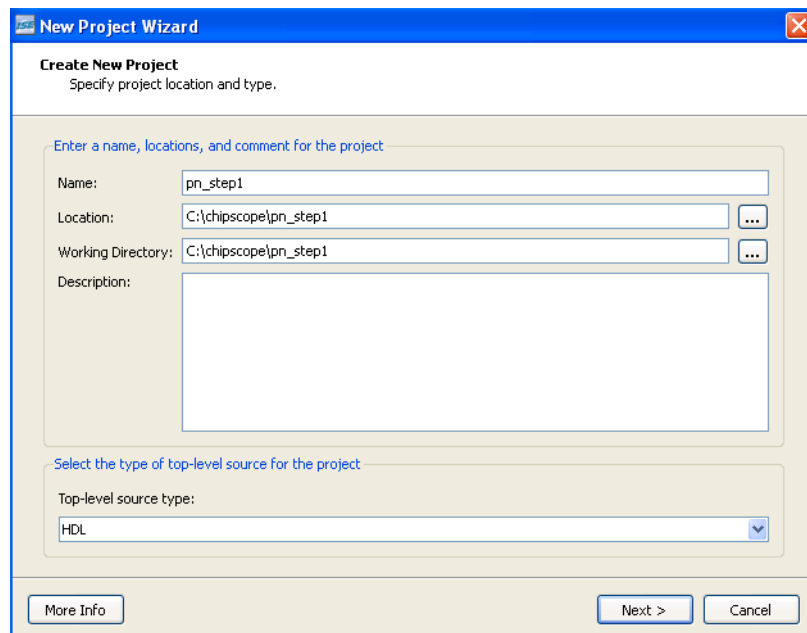
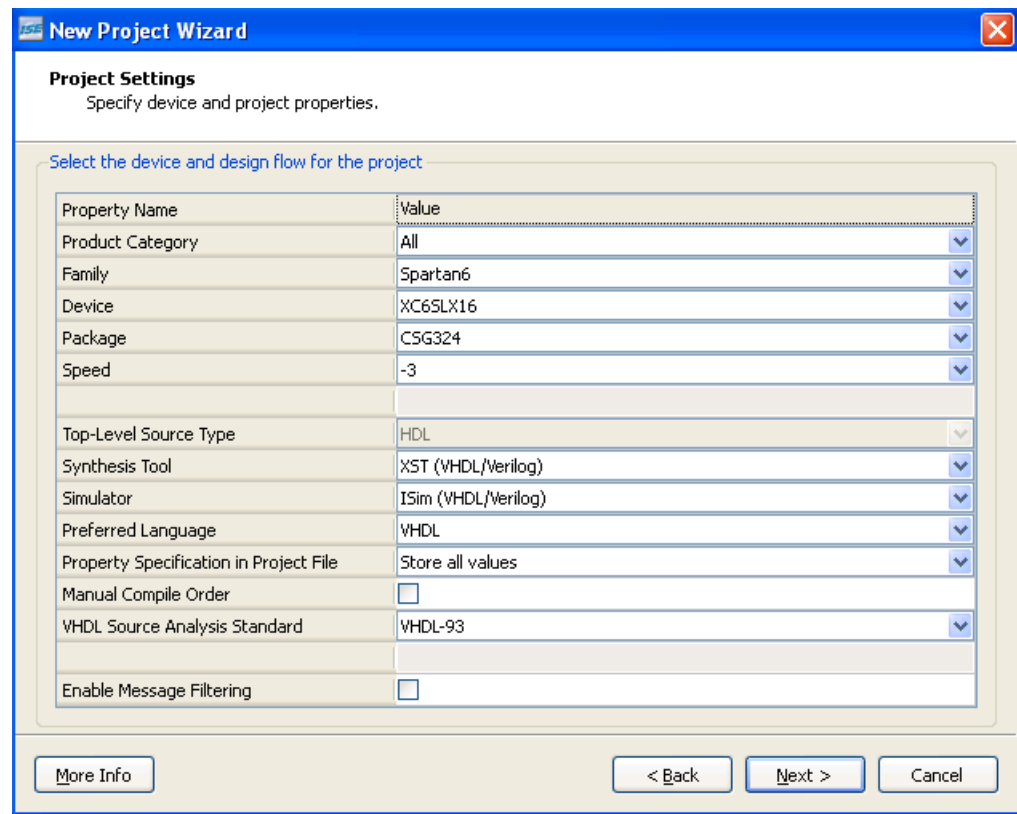


Figure 2-1: Create New Project Window

3. Specify device and project properties as shown here, and click **Next**



The image shows the 'New Project Wizard' dialog box, specifically the 'Project Settings' tab. The title bar says 'New Project Wizard' with a close button. Below the title bar, the text 'Project Settings' is followed by 'Specify device and project properties.' Below this is a section titled 'Select the device and design flow for the project'. This section contains a table with two columns: 'Property Name' and 'Value'. The properties and their values are: Product Category (All), Family (Spartan6), Device (XC6SLX16), Package (CSG324), Speed (-3), Top-Level Source Type (HDL), Synthesis Tool (XST (VHDL/Verilog)), Simulator (ISim (VHDL/Verilog)), Preferred Language (VHDL), Property Specification in Project File (Store all values), Manual Compile Order (unchecked checkbox), VHDL Source Analysis Standard (VHDL-93), and Enable Message Filtering (unchecked checkbox). At the bottom of the dialog are three buttons: 'More Info', '< Back', and 'Next >', and a 'Cancel' button on the far right.

Property Name	Value
Product Category	All
Family	Spartan6
Device	XC6SLX16
Package	CSG324
Speed	-3
Top-Level Source Type	HDL
Synthesis Tool	XST (VHDL/Verilog)
Simulator	ISim (VHDL/Verilog)
Preferred Language	VHDL
Property Specification in Project File	Store all values
Manual Compile Order	<input type="checkbox"/>
VHDL Source Analysis Standard	VHDL-93
Enable Message Filtering	<input type="checkbox"/>

Figure 2-2: Project Settings Window

4. Click **Finish** on the Project Summary page.
5. Right-click the **pn_step1** project and select **Add Source** to add VHDL source files to the new project.

6. Locate the source files in the `src` directory, select all VHDL source files, and open them..

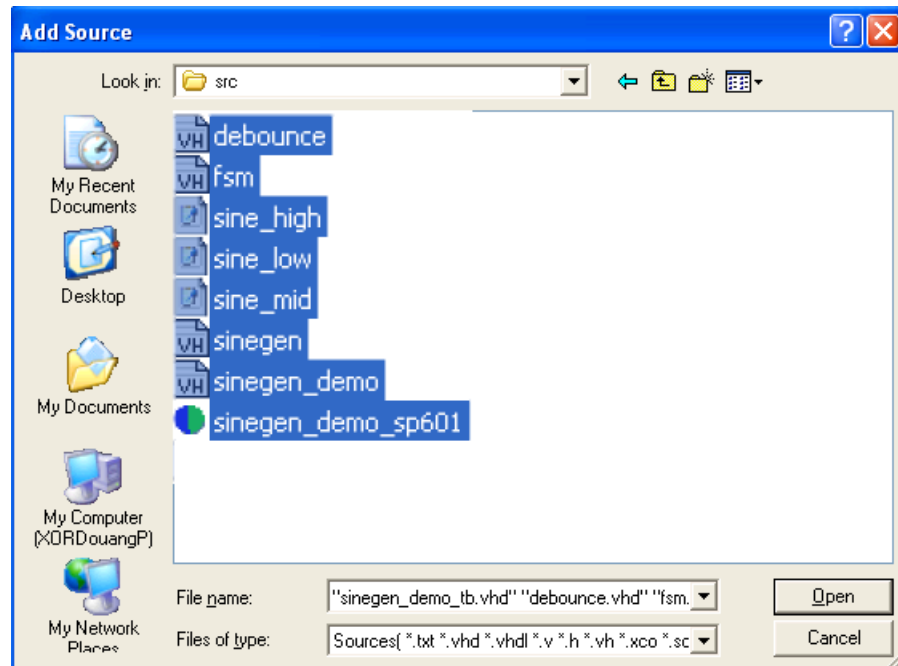


Figure 2-3: Source Files

7. Review the files listed in the Adding Source Files window and click **OK**.

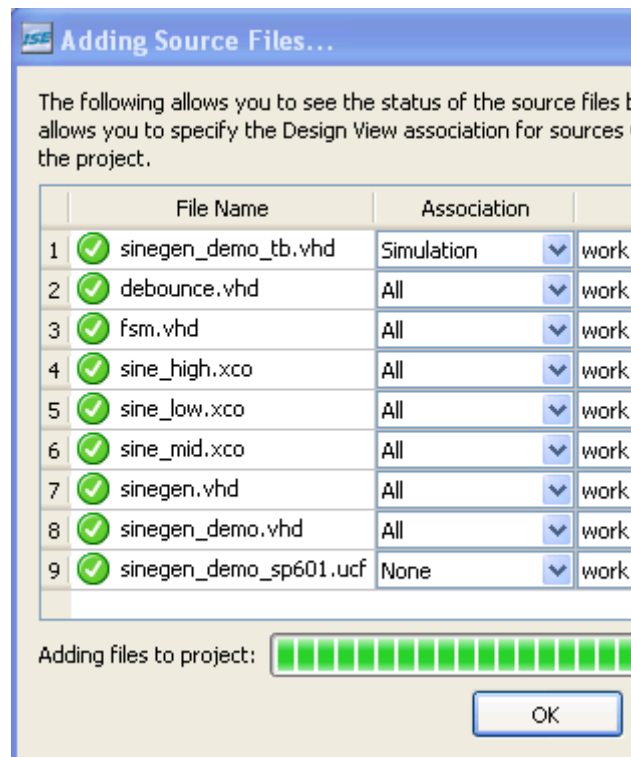


Figure 2-4: Adding Source Files Window

8. Right-click **sinegen_demo** and select **New Source** to add a new definition and connection (.cdc) file to the project.

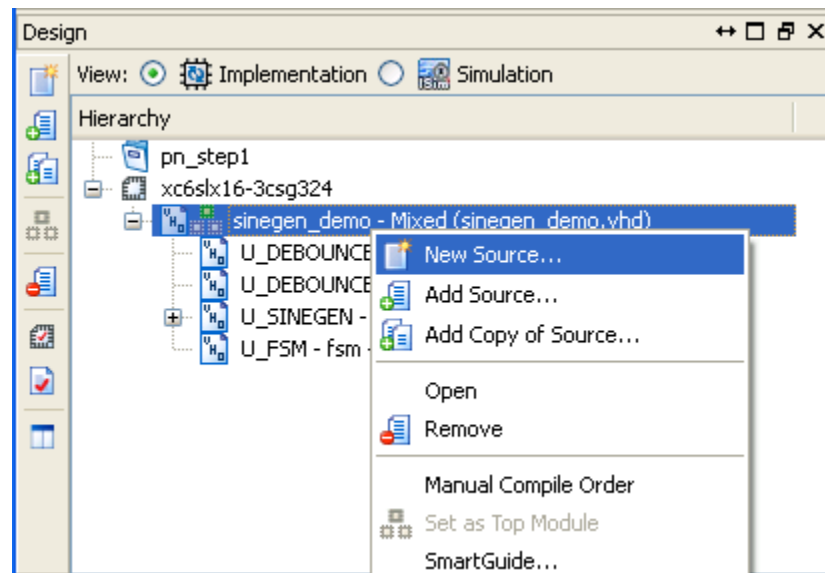


Figure 2-5: New Source Selection

9. In the Select Source Type window, select **ChipScope Defination and Connection File**, and then type **pn_step1.cdc** for the file name and click **Finish**.
 10. Review the information in the Summary dialog box and then click on **Finish**.
- The next step is to generate all three sine wave generator cores.

11. In the Design window, select all three .xco files listed under U_SINEGEN, and then double-click the **Regenerate All Cores** process.

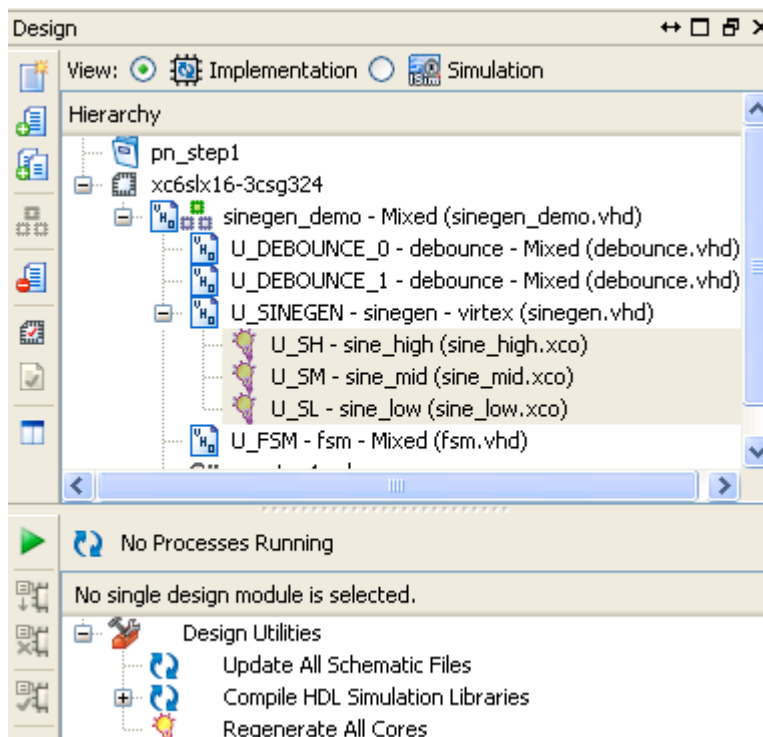


Figure 2-6: Sine Wave Generator Files in the Design Window

12. Prior to synthesizing the design, set the **Keep Hierarchy** option to **Soft** to preserve the design hierarchy and prevent the XST from performing hierarchical optimizations. To do this, right-click the Synthesize process and set the **-Keep_hierarchy** switch to **Soft**.
13. Click **Apply**.
Now you are ready to synthesize the design.
14. Click the **Synthesize** process.
15. Select **File > Copy Project** to copy the project as pn_step2.
16. Type the project name and location, and then click **OK**.

At this point, you have successfully created the ISE project and synthesized the design using Project Navigator.

Questions

1. Describe briefly of what you did in Step1: _____
2. What are some major circuits used in this lab? _____
3. Which source file would you have to modify if you were to target other Xilinx® boards? _____

Adding a ChipScope ILA Core to Your Design

Adding a ChipScope ILA Core

In this section of the tutorial, you will add a ChipScope™ ILA core to your design by taking advantage of integration flows between the Project Navigator and ChipScope Pro Core Inserter tools. Traditionally, you had to manually instantiate the ILA core instances into the RTL designs. However, that method requires you to modify your design source files, which can be cumbersome and can increase the chance of potentially making mistakes.

Instead, you will learn here how to add the core, complete the signal connections, and probe the design without modifying the original RTL source files.

1. In the Hierarchy window, double-click the `pn_step1.cdc` file to open the ChipScope Pro Core Inserter tool.

The first window displays Device Options. Because the target device is already set in Project Navigator, the device options are already set for you. This eliminates some potential conflicts in device selection between ChipScope Pro Core Inserter and Project Navigator.

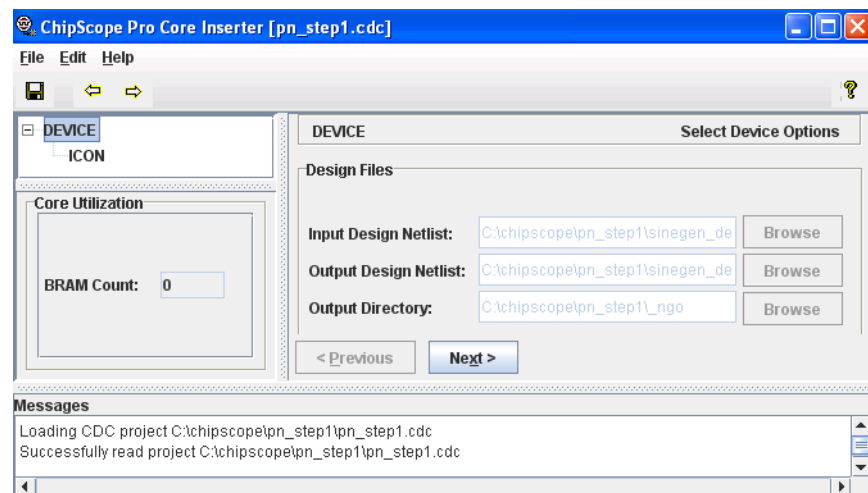


Figure 3-1: ChipScope Pro Core Inserter

- Click **Next**. The window displays the ICON options.

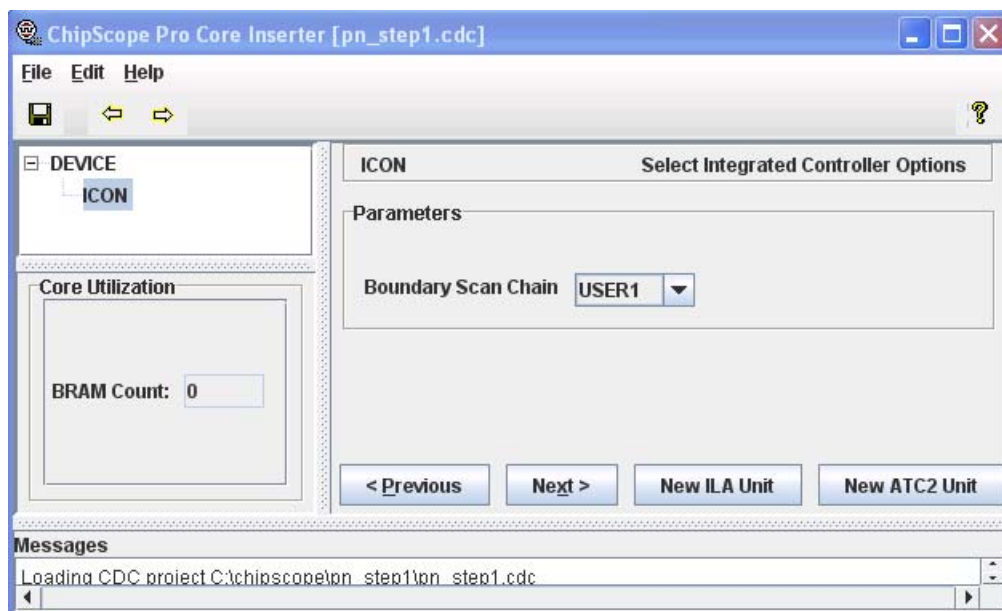


Figure 3-2: ICON Options

- Click **Next** to add an ILA core.
- In the Trigger Parameters tab of the ILA Options window, set the trigger parameters.

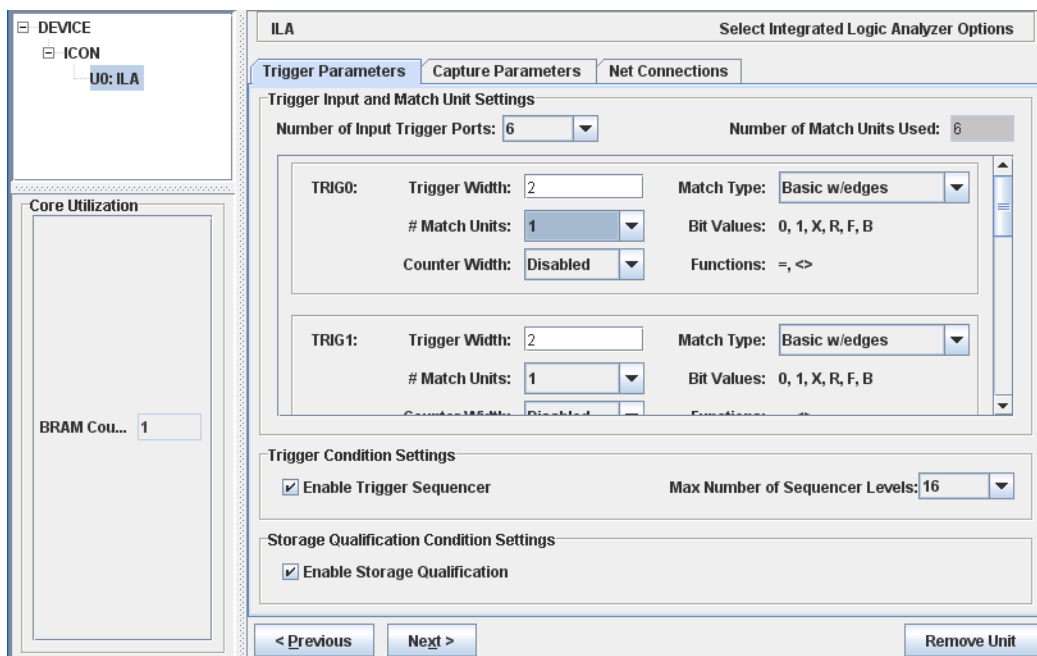


Figure 3-3: Trigger Parameters Tab of the ILA Options Window

For this example, set the trigger parameters as follows.

Table 3-1: Trigger Parameters

Trigger Name	Trigger Width	Match Type
TRIG0	2	Basic w/ edge
TRIG1	2	Basic w/ edge
TRIG2	2	Basic w/ edge
TRIG3	20	Basic
TRIG4	2	Basic w/ edge
TRIG5	3	Basic w/ edge

5. Select the **Capture Parameters** tab and examine the trigger parameters you just set

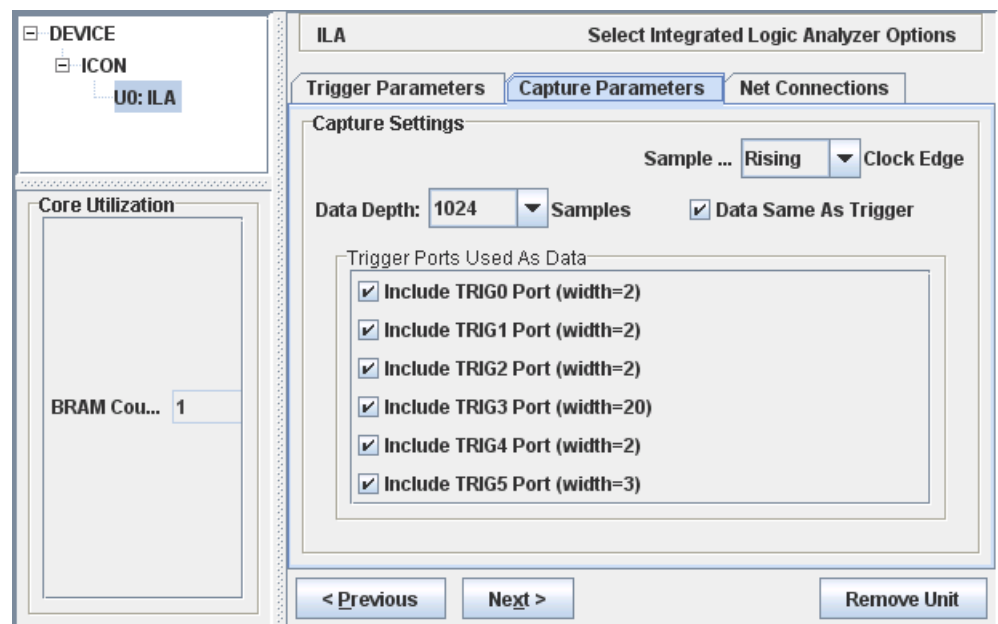


Figure 3-4: Capture Parameters Tab

6. Connect each port to debugging nets. To do this:
 - a. Select the **Net Connections** tab.
 - b. Click **Modify Connections**. The Select Net dialog window appears.
 - c. From the Select Net window, search for the CLK_BUFG net from the sinegen_demo hierarchy.
To search for the "CLK_BUFG" net, type the "clk_bufg" string in the **Pattern** field and click **Filter**.

- d. Select the `clk_BUFG` net from search results and click **Make Connections**

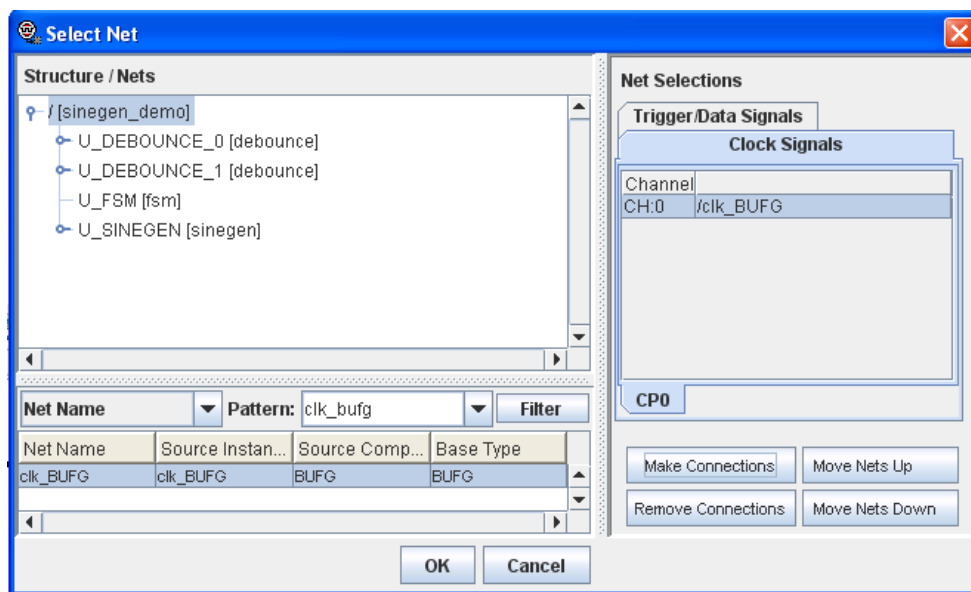


Figure 3-5: Select Net Window

7. Repeat Step 6 to connect the rest of trigger ports:
- TRIG0: search for `*sel*` from the `U_SINEGEN` hierarchy
 - TRIG1: search for `*GPIO_BUTTONS_re*` from the `sinegen_demo` hierarchy
 - TRIG2: search for `*GPIO_BUTTONS_dly*` from the `sinegen_demo` hierarchy
 - TRIG3: search for `*SINE*` from the `sinegen_demo` hierarchy
 - TRIG4: search for `*GPIO_BUTTONS_db<0>*` from the `sinegen_demo` hierarchy
 - TRIG4: search for `*GPIO_BUTTONS_db<1>*` from the `sinegen_demo` hierarchy
 - TRIG5: search for `*GPIO_BUTTONS_0_IBUF*` from the `sinegen_demo` hierarchy
 - TRIG5: search for `*GPIO_BUTTONS_1_IBUF*` from the `sinegen_demo` hierarchy
 - TRIG5: search for `*GPIO_SWITCH_IBUF*` from the `sinegen_demo` hierarchy

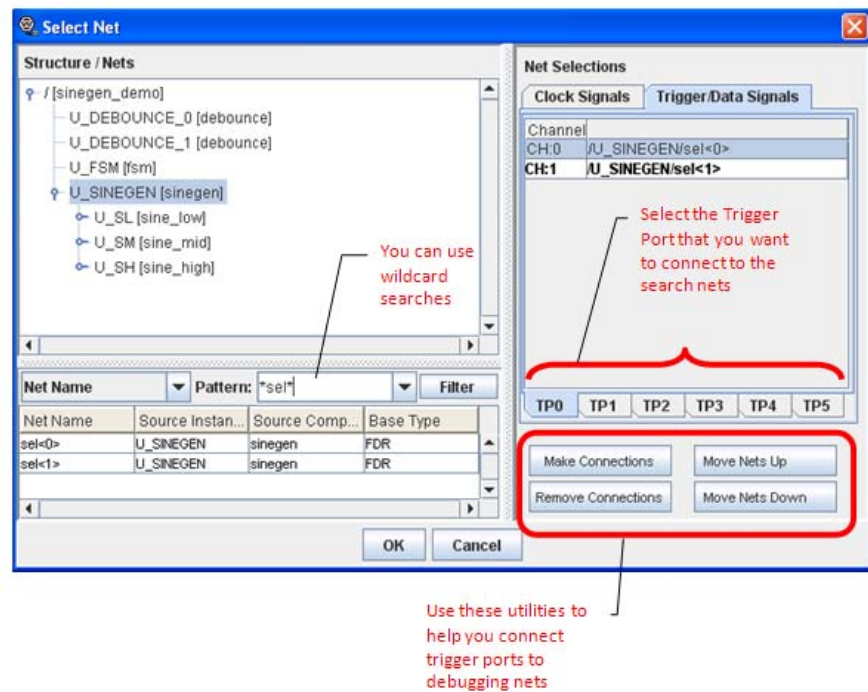


Figure 3-6: Trigger/Data Signals

8. Once complete, all ports turn from red to black, indicating that you have finished connecting all the clock and trigger ports to debugging nets.

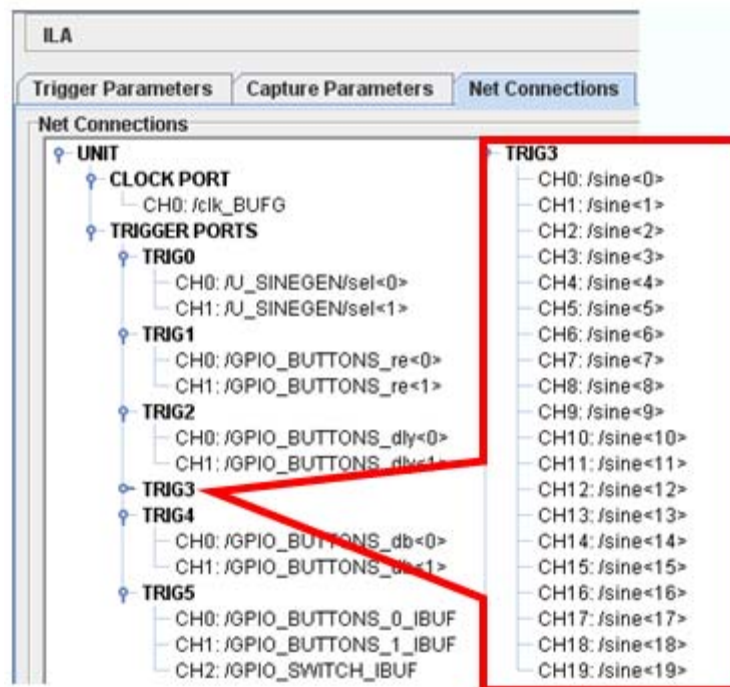


Figure 3-7: Complete Net Connections

9. Save and close the `pn_step1.cdc` file

Before you use the ChipScope Pro Analyzer tool to download your bitstream into your device, make sure the bitstream generation options are set properly.

10. Right-click on the **Generate Programming File** process and select **Properties**.
11. In the Startup Options category, set the `-g StartUpClk` switch to the **JTAG Clock** option.

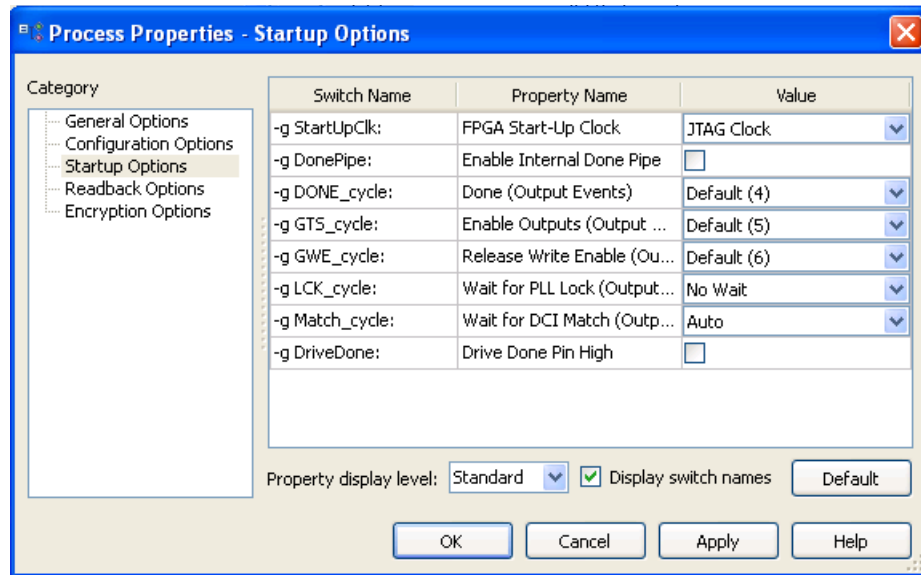


Figure 3-8: Startup Options Window

Now you can start generating a programming file.

12. Double-click the **Analyze Design Using ChipScope** process.
- When the process completes, the ChipScope Pro Analyzer tool opens.

Questions

4. What is the main advantage of inserting debug probes onto your post-synthesis netlist instead of adding them onto HDL design files? _____

You have just finished with inserting a ChipScope ILA core and now you are ready to debug the design using ChipScope Pro Analyzer.

Debugging Your Design using ChipScope Pro Analyzer

Debugging Your Design

This lab exercise shows how to debug a design using Xilinx® ChipScope™ Pro Analyzer and how to iterate the design once you have discovered and fixed errors. This step will also show some useful techniques of how to trigger and capture certain data from your design.

You will be using ChipScope Pro Analyzer to verify that Sine wave generator is working correctly. The two primary objectives will be to verify and check off the following two items.

- ☐ Verify that all sine wave selections look correct
- ☐ Verify that selection logic is working correctly

Do the following:

1. Configure JTAG Chain to USB cable and communication parameters by doing the following:
 - a. Select **JTAG Chain > Xilinx Platform USB Cable**.
 - b. In the ChipScope Pro Analyzer dialog box that opens, set the speed and port parameters.
For this tutorial, Speed is 3 MHz and Port is USB21.
 - c. Click **OK**.
 - d. In the ChipScope Pro Analyzer dialog box that opens, verify the device details and then click **OK**.

2. Confirm the connection to the JTAG chain.

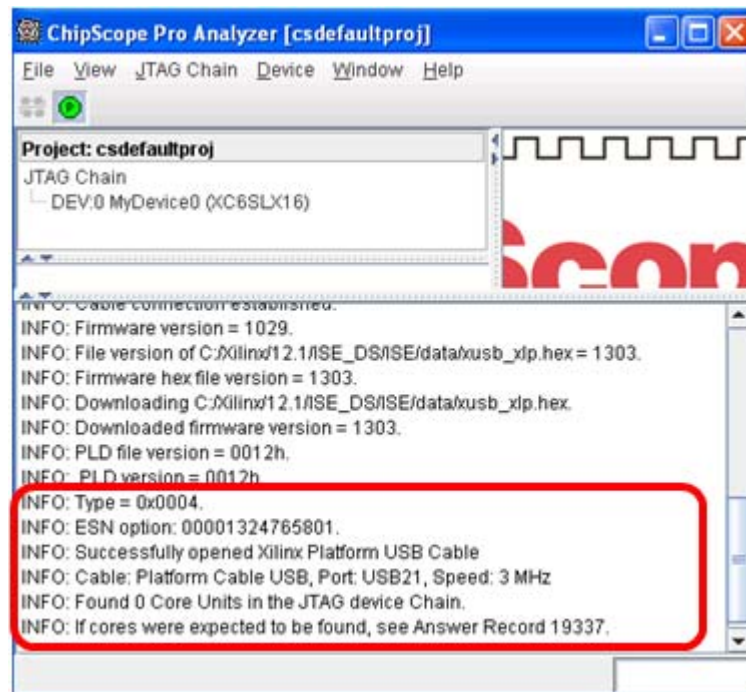


Figure 4-1: JTAG Chain Connection Details

3. Right-click the device name in the Project list and select **Configure** to configure the device.
4. In the Configuration window, select the default BIT and CDC files from Project Navigator.

5. Verify the device configuration and ILA core.

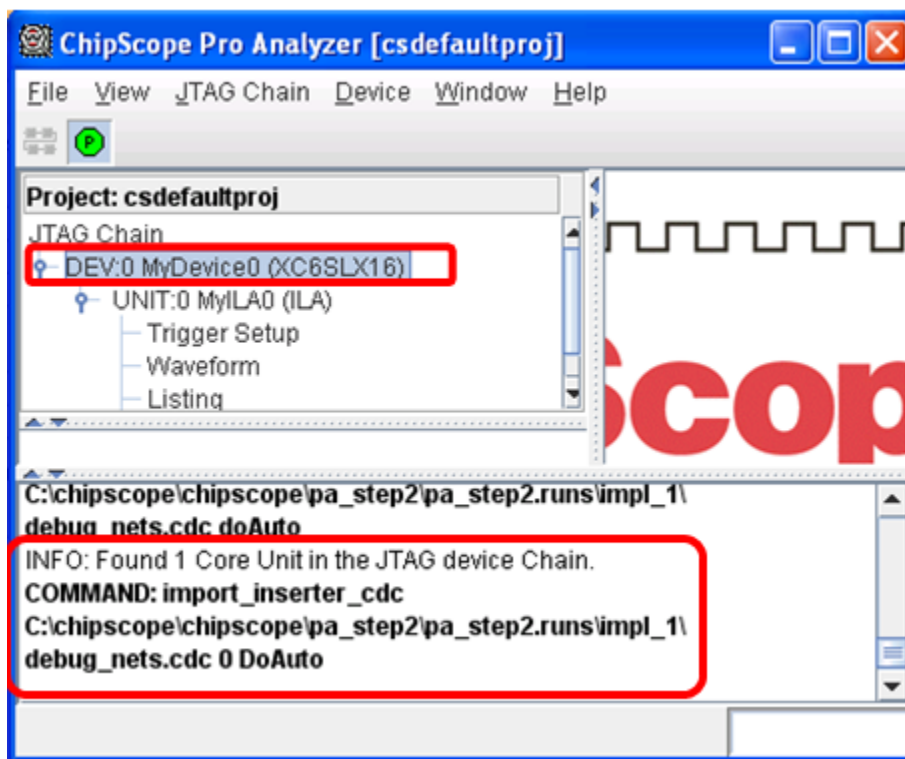


Figure 4-2: Device Configuration Details

6. Open the Trigger Setup and Waveform windows by double-clicking on each item: **Trigger Setup** and **Waveform**.

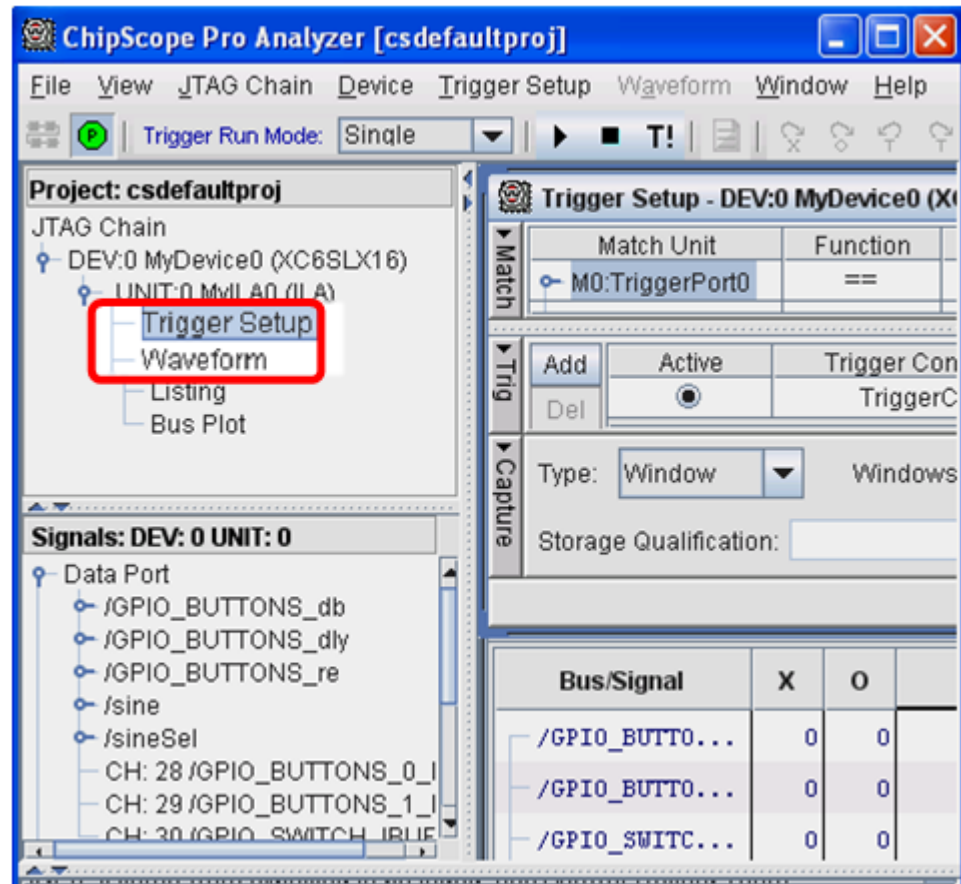


Figure 4-3: Location of Trigger Setup and Waveform

7. Select **Trigger Setup > Trigger Immediate**.
8. Verify that there is activity on the sine wave.

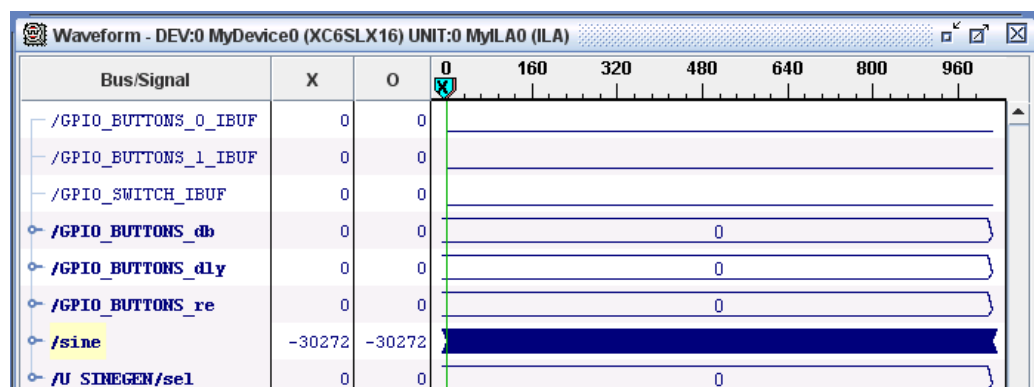


Figure 4-4: Sine Wave Activity

9. Double-click **Bus Plot** to open the Bus Plot viewer.

10. On the Bus Plot window, select **/sine** to display the sine wave.

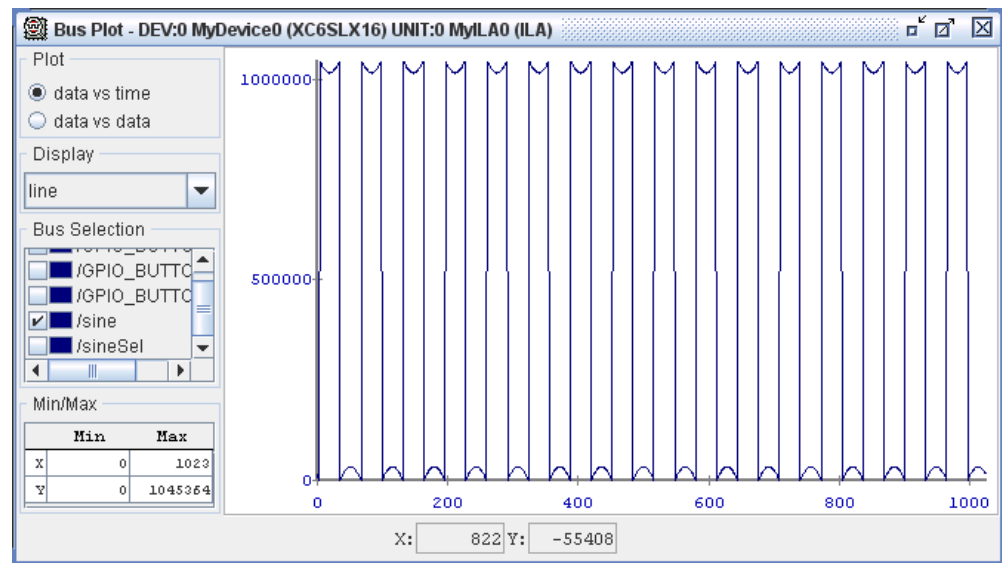


Figure 4-5: Sine Wave Display

Notice that this waveform does not look much like a sine wave. This is because the radix setting needs to be changed to from Hex to Sine Decimal.

11. In the Signal window, right-click **/sine** and select **Bus Radix**. Click to select the **Signed Decimal** check box.
12. Click the **Trigger Immediately** button **T!** and view the high-frequency sine wave bus plot.

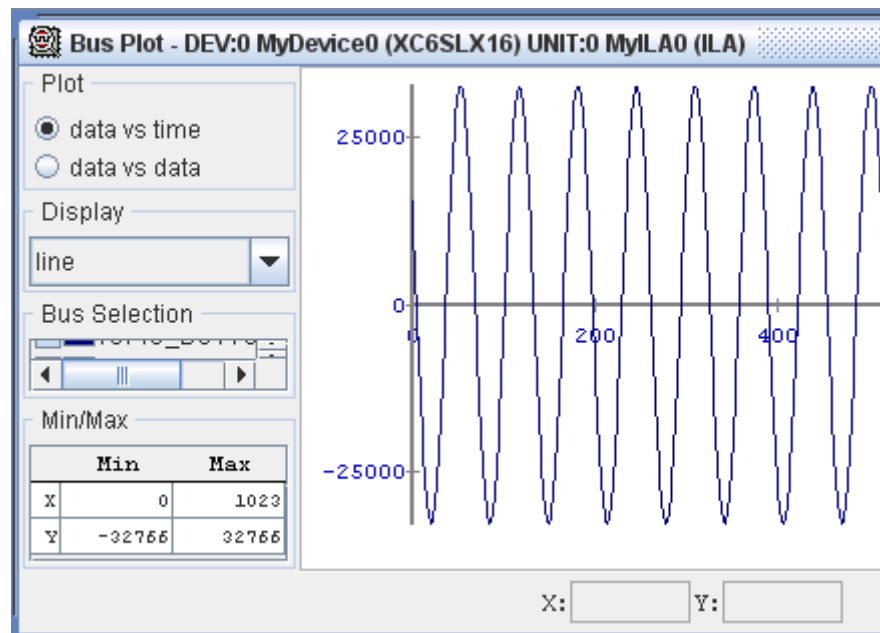


Figure 4-6: High-Frequency Sine Wave Bus Plot

13. On your board, push the Sine Wave Sequencer button (shown in [Figure 1-1](#)) until the Sine Wave Selection indicator LEDs on the board display "off,on, 01."

Note: Sequencer is not working correctly. The expected behavior is a simple 2-bit counter that counts for each press (00, 01, 10, 11...). You will debug the root cause for this later in this tutorial.

14. Click the **Trigger Immediately** button again, and view the mid-frequency bus plot.

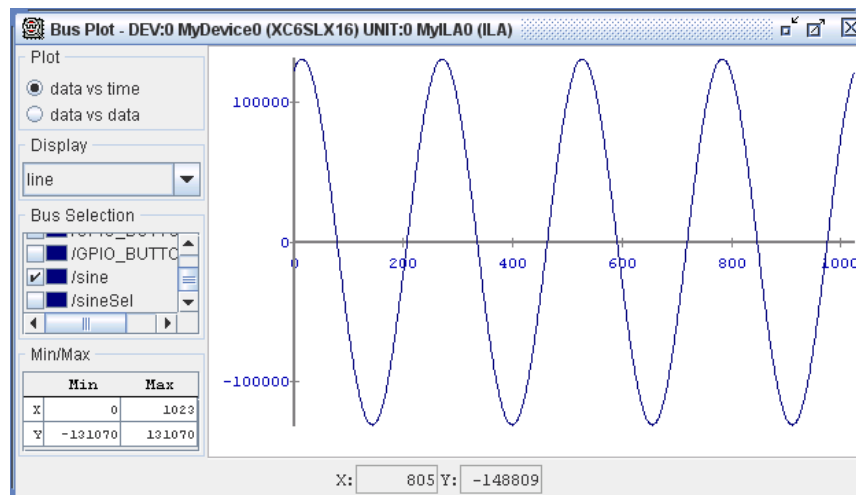


Figure 4-7: Mid-Frequency Sine Wave Bus Plot

15. Push the Sine Wave Sequencer button on the board until the Sine Wave Selection indicator LEDs on the board display "on, off, 10."
16. Click the **Trigger Immediately** button and view the low-frequency bus plot.

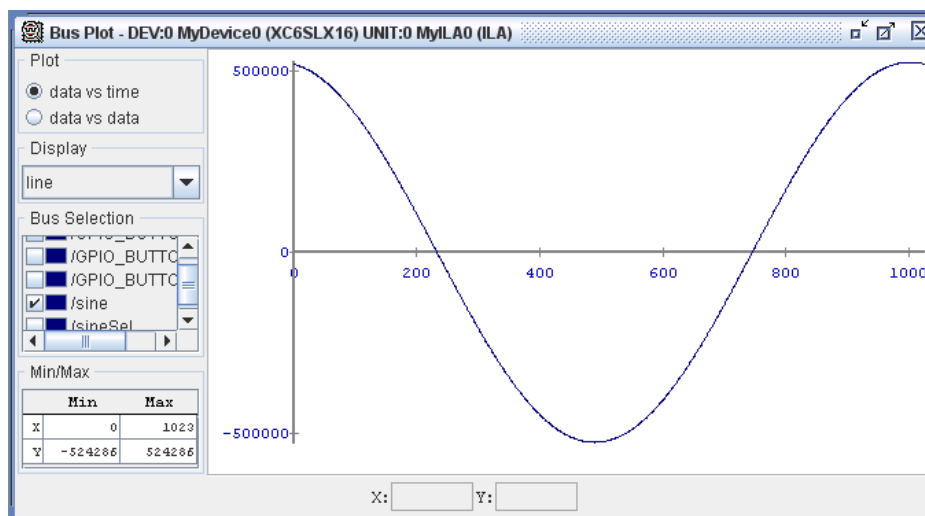


Figure 4-8: Low-Frequency Sine Wave Bus Plot

17. Push the Sine Wave Sequencer button on the board until the Sine Wave Selection indicator LEDs on the board display "on, on, 11."

18. Click the **Trigger Immediately** button and view the combined sine wave bus plot.

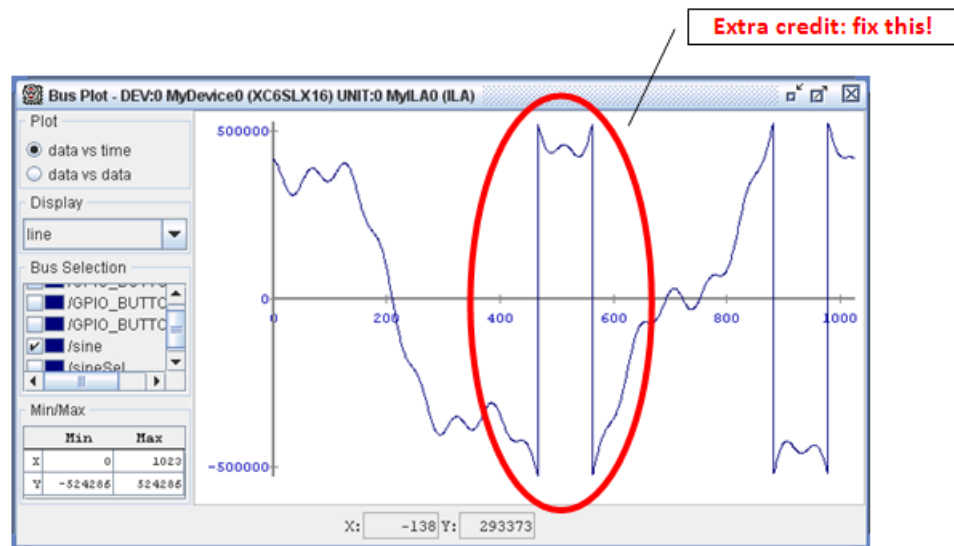


Figure 4-9: Combined Sine Wave Bus Plot

You just verified that all sine wave selections look correct. However, the selection logic circuit is still not working correctly.

- ☒ Verify that all sine wave selections look correct.
- ☐ Verify that selection logic is working correctly:
 - ☐ Verify that state machine is transitioning correctly and outputs are correct.
 - ☐ Verify that state machine inputs are correct.

Next, start debugging the selection logic circuit.

19. Set the following parameters:

Match: set TriggerPort1 to **RX** -- to look for rising edge on GPIO_BUTTONS_re[1] because this is what causes FSM to transition.

Trigger Conditions: M1 -- set up trigger equation to **M1**

Capture Settings: Windows = 10; Depth = 2

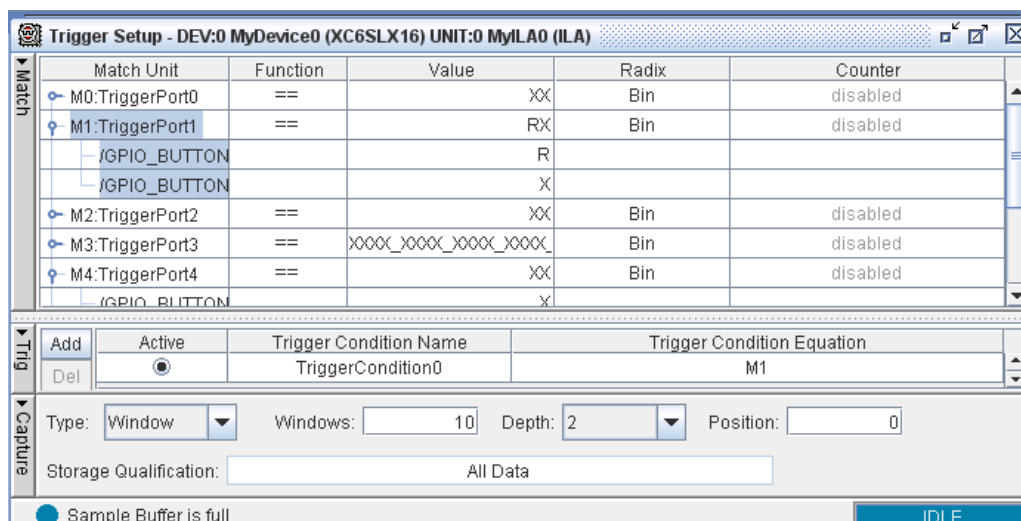


Figure 4-10: Trigger Setup Window

Note: The actual TriggerPort number for the `GPIO_BUTTONS_re[1]` net might not be the same as specified on this tutorial.

20. Click the **Run Trigger** button to arm the trigger and wait for the trigger condition. Information about the run progress displays at the bottom of the window.
21. Push the Sine Wave Sequencer button on the board.
22. Observe the number of windows captured.
 - If only one window was captured, repeat steps 21 and 22.
 - If more than one window was captured, go to the next step.

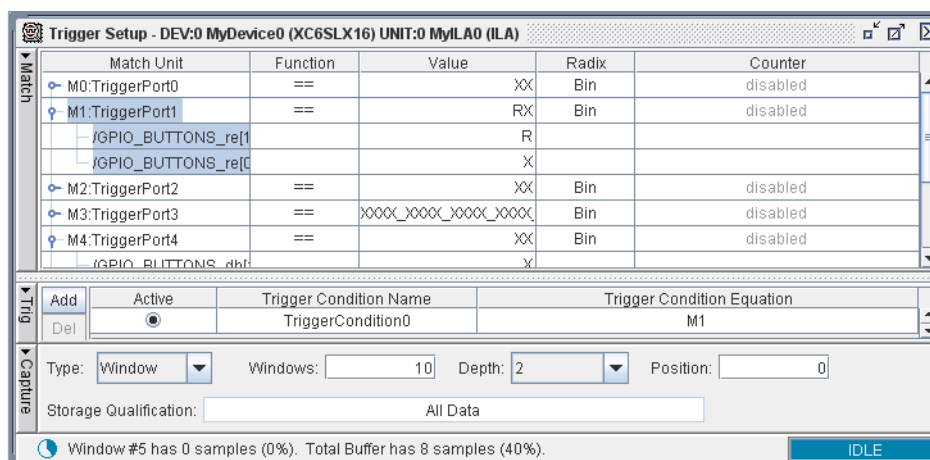


Figure 4-11: Observing the Windows Captured


23. Click the **Stop Trigger** button and view the captured data. Notice the multiple rising edges each time you pressed the Sine Wave Sequencer button. Also, note the correct transition of `sineSel`, which indicates that the state machine is working properly.

You just verified that sine wave generators are working correctly.

- ☒ Verify that all sine wave selections look correct.
- ☐ Verify that selection logic is working correctly:
 - ☒ Verify that state machine is transitioning correctly and outputs are correct.
 - ☐ Verify that state machine inputs are correct.

24. Set the following parameters for trigger modes and conditions:

- Trigger Run Mode: **Repetitive**
- Match: set **TriggerPort5** to **XX** -- to look for rising edge on GPIO_BUTTONS_1_IBUF, which is the input buffer for the Sine Wave Sequencer button on the board.
- Trigger Conditions: **M5**
- Capture Settings:
 - Windows = **1**
 - Depth = **1024**
 - Position = **512**

25. Click the **Run Trigger** button  and press the Sine Wave Sequencer button on the board until you see multiple transitions on the GPIO_BUTTONS_1_IBUF signal.

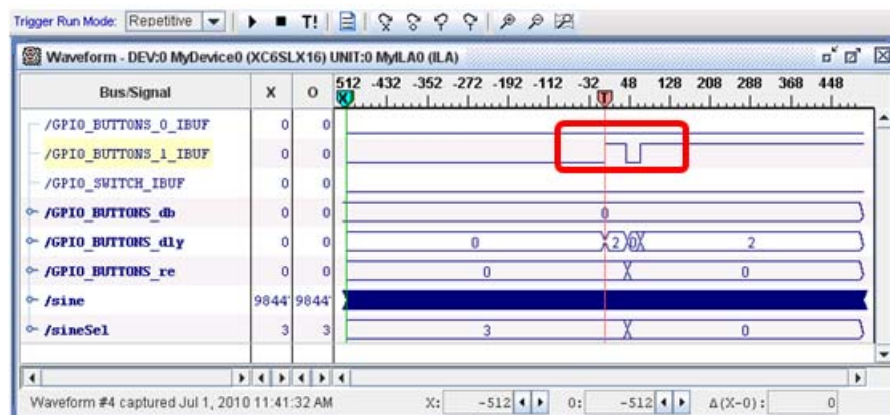


Figure 4-12: View Waveform

Note: Your waveform might not display signal glitches at exactly the same location as shown here. This is one of the advantages of the Repetitive triggering feature.

You just verified that sine wave generators are working correctly. However, the state machine inputs are not correct. These inputs are connected directly from Push Button switches.

- ☒ Verify that all sine wave selections look correct.
- ☒ Verify that selection logic is working correctly:
 - ☒ Verify that state machine is transitioning correctly and outputs are correct.
 - ☒ Verify that state machine inputs are correct.

As shown in [Figure 4-12](#), the problem seems to point to Push Button switches, which generate glitches every time the Push Button switch is pressed and released. A debounce circuit is required for each push button switch to eliminate these glitches that result in multiple transitions.

The debounce circuits have already been integrated in the provided design. To enable debounce circuits, turn on Dip switch-1, repeat steps [24](#) and [25](#), and verify there is only one transition for each button push.

Questions

5. Did you have time to resolve the problem on Step [18](#) for extra credit? _____
6. Why is a debounce circuit required for this lab? _____

Tutorial Conclusion

Conclusion

This tutorial introduced you to the tightly integrated design flow between ChipScope Pro ILA Core Inserter and Project Navigator. It showed you how to generate IP core netlist in Project Navigator and synthesize the design. Secondly, it illustrated how to add ChipScope ILA core to the design using the ILA Core Inserter. More importantly, this tutorial guides you through a debugging process. It showed you how to validate and debug your design using ChipScope Pro Analyzer using various triggering setups.

You should now be familiar with some basic design flows and integration between Project Navigator and ChipScope Pro.

Question Answers

1. Describe briefly of what you did in Step1.
You just created an RTL PlanAhead project, loaded a VHDL ChipScope™ design, and implemented the design.
2. What are some major circuits used in this lab?
 - *debounce.vhdl - Debounce circuit*
 - *fsm.vhdl - Control state machine*
 - *sinegen_demo - Wrapper for Sine wave generators*
3. Which source file would you have to modify if you were to target other Xilinx® boards?
UCF constraint file
4. What is the main advantage of inserting debug probes onto your post-synthesis netlist instead of adding them onto HDL design files?
You don't have to directly touch and / or modify original HDL source files, thereby eliminating the risk of making any unintentional changes
5. Did you have time to resolve the problem on Step 18 for extra credit?
Hint: *Judging by the waveform from Analyzer, it appeared that the output Sine wave got truncated possible due to insufficient bit vector specified in "sinegen_demo.vhd and sinegen.vhd" modules. It's currently specified as a 20-bit vector. It should be expanded to 22-bit. Feel free to modify these two modules and iterate the design.*
6. Why is a debounce circuit required for this lab?
It provides a clean pulse or transition from a high to low on this particular example. It eliminates series of spikes or glitches when a button is pressed and released.

