

PlanAhead Software Tutorial

Debugging with ChipScope

UG677 (v13.3) October 19, 2011



The information disclosed to you hereunder (the "Information") is provided "AS-IS" with no warranty of any kind, express or implied. Xilinx does not assume any liability arising from your use of the Information. You are responsible for obtaining any rights you may require for your use of this Information. Xilinx reserves the right to make changes, at any time, to the Information without notice and at its sole discretion. Xilinx assumes no obligation to correct any errors contained in the Information 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. XILINX MAKES NO OTHER WARRANTIES, WHETHER EXPRESS, IMPLIED, OR STATUTORY, REGARDING THE INFORMATION, INCLUDING ANY WARRANTIES OF MERCHANTABILITY, FITNESS FOR A PARTICULAR PURPOSE, OR NONINFRINGEMENT OF THIRD-PARTY RIGHTS.

© Copyright 2010- 2011 Xilinx, Inc. XILINX, the Xilinx logo, Virtex, Spartan, ISE, and other designated brands included herein are trademarks of Xilinx in the United States and other countries. All other trademarks are the property of their respective owners.

Revision History

The following table shows the revision history for this document.

Date	Revision
03/01/2011	Updated for a new debug flow in PlanAhead™ software.
07/06/2011	Minor updates for clarification. No technical content updates.
10/19/2011	Updated for 13.3 software. Made updates for usability and clarity. Updated links in Additional Resources.

Table of Contents

Revision History	2
Tutorial: Debugging with ChipScope Pro	
Prerequisites	5
Objectives	5
Getting Started	5
Step 1: Creating and Implementing an RTL Project in the PlanAhead Software	8
Step 2: Using ChipScope Tools to Debug the PlanAhead Software Design. . . .	10
Step 3: Using ChipScope Tools to Debug the Hardware	12
Additional Resources	
Xilinx Resources	21
ChipScope Documentation	21
PlanAhead Documentation	21
Board Documentation	21

Tutorial: Debugging with ChipScope Pro

Prerequisites

A basic knowledge of Xilinx® ISE® Design Suite tool flows.

Objectives

This tutorial:

- Shows you how to take advantage of enhanced ChipScope™ Pro Analyzer features in the PlanAhead™ software that make the debug process faster and more simple.
- Provides specifics on how to use the PlanAhead software and the ChipScope Analyzer to debug some common problems in FPGA logic designs.

After completing this tutorial, you will be able to:

- Validate and debug your design using the PlanAhead software and the integrated ChipScope Analyzer with an ILA (Integrated Logic Analyzer) core.
- Understand how to create an RTL project, probe your design using an ILA core, and implement the design in the PlanAhead software.
- Generate and customize an IP core netlist in the PlanAhead software design environment.
- Debug the design using ChipScope Pro Analyzer and iterate the design using the PlanAhead software design environment and a Spartan®-6 FPGA SP601 Evaluation Kit Base Board (SP601 Platform).

Getting Started

Before you start this tutorial, make sure you have and understand the hardware and software components needed to perform the steps. The following subsection lists the requirements.

Set Up Requirements

The following software and hardware are required to follow the steps in this tutorial:

- Xilinx ISE® Design Suite 13.3 (Logic, DSP, Embedded, or System Edition).
- Spartan-6 FPGA SP601 Evaluation Kit Base Board. For a link to information about the SP601 board, see [Additional Resources, page 21](#).
You can also use the SP605 or ML605 board with this tutorial. See [Board Support and Pinout Information, page 7](#) for more information.
- USB and power cables that come with the SP601 Evaluation Kit.

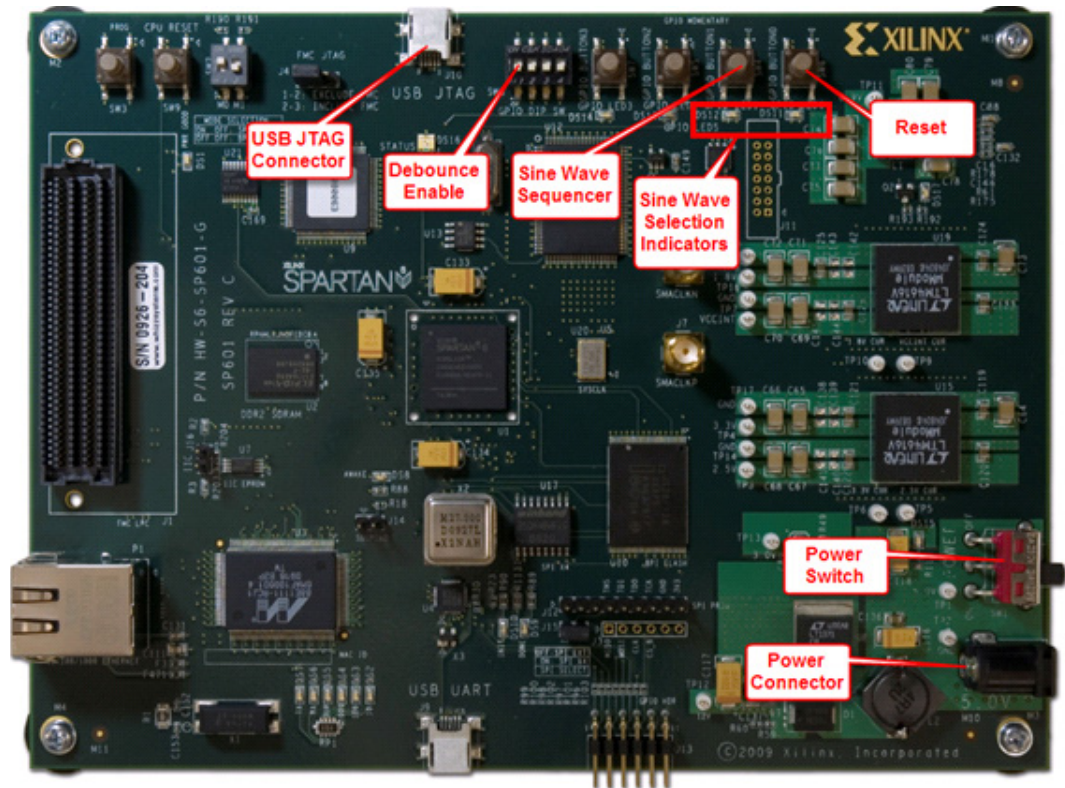


Figure 1: SP601 Board Showing Key Components

Tutorial Design Components

The design includes:

- A simple control state machine
- Multiple sine wave generators
- Common push buttons (GPIO_BUTTON)
- DIP switches (GPIO_SWITCH)
- LED displays (GPIO_LED)
- Push Button Switches: Serve as inputs to the debounce and control state machine circuits. Pushing a button generates a high-to-low transition pulse. Each generated output pulse is then used as an input into the state machine.
- DIP Switch: Enables or disables a debounce circuit.
- Debounce Circuit: In this example, when enabled, provides a clean pulse or transition from high to low. Eliminates a series of spikes or glitches when a button is pressed and released.
- Sine Wave Sequencer State Machine: Captures and decodes input pulses from the two push button switches. 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_1 display selection status from the state machine outputs, each of which represents a different sine wave frequency: high, medium, and low.
- Tutorial design files: For details on locating design files, see [Getting Started, page 8](#).

Board Support and Pinout Information

Note: This tutorial also supports two other Xilinx platforms: SP605 and ML605. Use the pin-out information in [Table 1](#) to retarget this tutorial to the SP605 or ML605 board.

Table 1: Pinout Information for the SP605 or ML605 Board

	Pinout Locations			Function
	SP601	SP605	ML605	
CLK_N	K16	K22	H9	Clock
CLK_P	K15	K21	J9	Clock
GPIO_BUTTONS[0]	P4	F3	A19	Reset
GPIO_BUTTONS[1]	F6	G6	G26	Sine Wave Sequencer
GPIO_SWITCH	D14	C18	D22	Debounce Circuit Selector
LEDS_n[0]	E13	D17	AC22	Sine Wave Selection[0]
LEDS_n[1]	C14	AB4	AC24	Sine Wave Selection[1]
LEDS_n[2]	C4	D21	AE22	Reserved
LEDS_n[3]	A4	W15	AE23	Reserved

Step 1: Creating and Implementing an RTL Project in the PlanAhead Software

To create and implement an RTL project you will:

- Get started by unzipping the tutorial source files and opening the PlanAhead software.
- Create a New Project with the New Project Wizard.
- Synthesize the design.

Getting Started

1. In your C:/drive, create a folder called /ChipScope_PlanAhead.
2. Find the tutorial design source files.

This tutorial uses the sample design data included in the supplied example projects in the PlanAhead software or in a downloadable compressed ZIP file.

Design data location: http://www.xilinx.com/support/documentation/dt_planahead_planahead13-1_tutorials.htm

The tutorial and design files might be updated or modified in between software releases on the Xilinx website, where you can download the latest version of the materials.

3. Unzip the tutorial source file to the /ChipScope_PlanAhead folder.
4. When unzipped, look in ChipScope_PlanAhead/PlanAhead_Tutorial/Sources/chipscope/pa_cs_tutorial/src for the files and folder shown in [Figure 2](#).

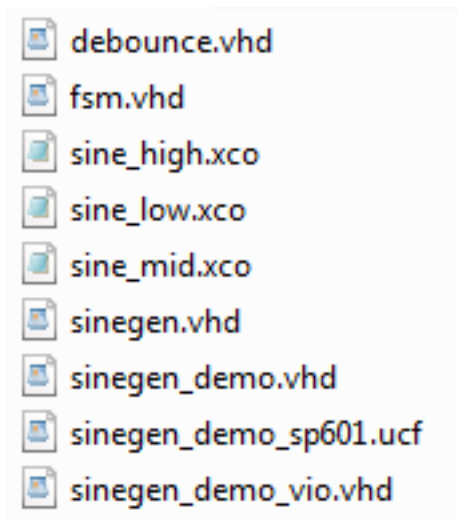


Figure 2: Tutorial Design File Set

Creating a Project with the PlanAhead Software New Project Wizard

To create a project, you will use the New Project wizard to name the project, to add RTL source files and constraints, and to specify the target device.

1. Invoke the PlanAhead software.
2. In the Getting Started screen, click **Create New Project** to start the New Project wizard.
3. In the Project Name screen, name the new project `pa_step1` and provide the project location (`C:\ChipScope_PlanAhead`).
4. In the **Design Source** dialog box, select **Specify RTL Sources**.
5. In the **Add Sources** dialog box:
 - a. Click the **Add Files** button.
 - b. In the **Add Source Files** dialog box, navigate to the `/src` directory where you saved your project design files.
 - c. Select all VHD source files, and click **OK**.
 - d. Verify that the files are added, and click **Next**.
6. In the **Add Existing IP** dialog box:
 - a. Click the **Add Files** button.
 - b. In the **Add Configurable IP** dialog box, navigate to the `/src` directory.
 - c. Select all XCO source files, and click **OK**.
 - d. Verify that the files are added, and click **Next**.
7. In the **Add Constraints** (optional) dialog box, if the UCF file does not automatically appear in the main window, select it and click **Next** to continue.
8. In the **Default Part** dialog box, specify the `xc6slx16csg324-2` part for the SP601 platform. It is easiest to use the search tool just above the parts list to find the correct item.
9. Review the New Project Summary screen. Verify that the data appears as expected, per the steps above.

Note: It might take a moment for the project to initialize.

After you exit the New Project wizard, you use the Project Manager in the PlanAhead software main window to add IP and to synthesize the design.

Synthesizing the Design

1. In the left panel, click the **Synthesize** button.

Note: When synthesis runs, a progress indicator appears, showing that synthesis is occurring. This could take a few minutes.
2. In the **Synthesis Completed** dialog box, click **Cancel**. You will implement the design later.
3. Select **File > Save Project As** and save the project as `pa_step2`. (Saving the project to a new file allows you to more easily resume the tutorial if you do not wish to complete all the steps in one sitting.)

Step 2: Using ChipScope Tools to Debug the PlanAhead Software Design

To add a ChipScope Analyzer ILA core to the design, you will take advantage of integration flows between the PlanAhead software and ChipScope Analyzer.

With the simplified workflow, you probe the design without modifying the original RTL source code. You will accomplish the following tasks:

- Add debug nets to the project
- Run the Set Up ChipScope Wizard
- Implement and open the design
- Generate the Bitstream

Adding Debug Nets to the Project

Working in the `pa_step2` project:

1. From the Netlist Design dropdown, click **Open Netlist Design** and accept the defaults.
2. Click the Netlist tab if it is not already selected.
3. Expand **Nets**. Select the following nets for debugging (also shown in [Figure 3](#)):
 - GPIO_BUTTONS_db (2)
 - GPIO_BUTTONS_dly (2)
 - GPIO_BUTTONS_re (2)
 - sine (20)
 - sineSel (2)
 - GPIO_BUTTONS_0_IBUF
 - GPIO_BUTTONS_1_IBUF

Note: These signals represent the significant behavior of this design and will be used to verify and debug the design in subsequent steps.

4. Right-click the selected nets and select **Add to ChipScope Unassigned nets**. If a confirmation dialog box opens, click **OK** to close it.

Note: With the ChipScope tab selected, you can see the unassigned nets you just selected.

The steps above are illustrated in [Figure 3, page 11](#).

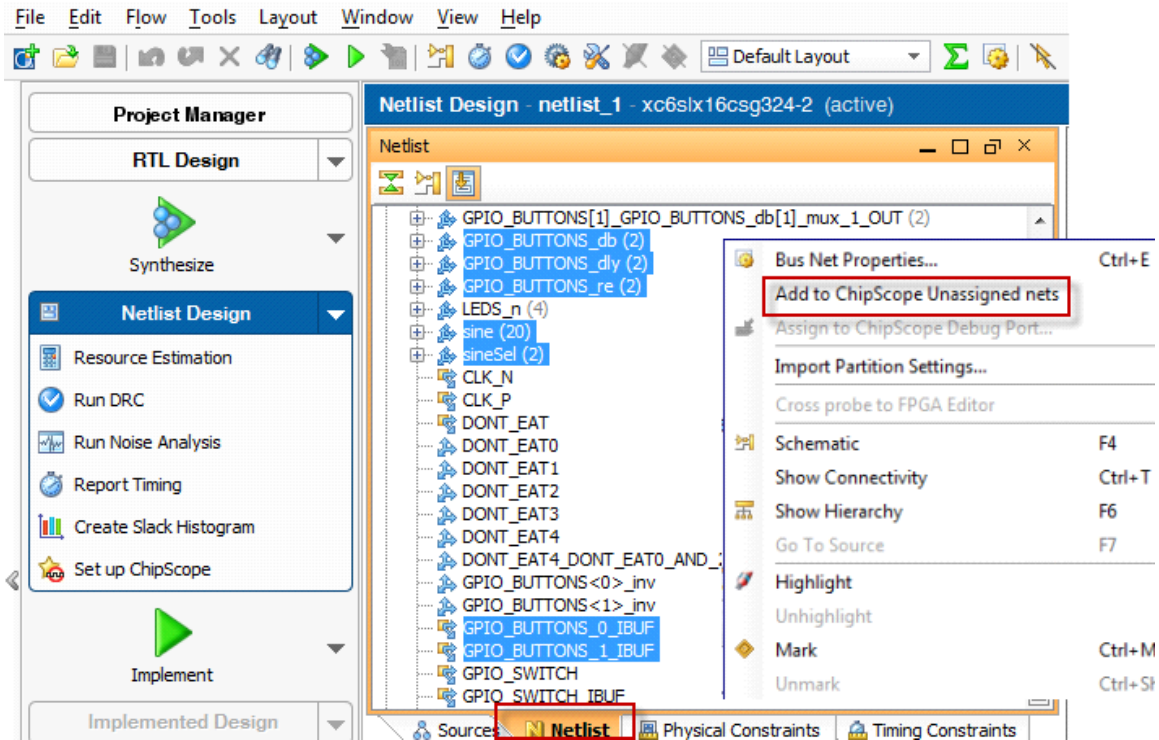


Figure 3: Adding Nets from the Netlist Tab

Running the Set Up ChipScope Wizard

1. From the Netlist Design dropdown, select **Set Up ChipScope**.
The Set Up ChipScope wizard opens.
2. Click through the wizard to create ChipScope Analyzer debug cores, keeping the default settings.

Implementing the Design and Generating the Bitstream

1. In the left panel, main window of the PlanAhead software, click the **Implement** button.
2. In the Save Project pop-up menu, select **Save**.
When the implementation process ends, an Implementation Completed dialog box opens.
Note: Implementation could take a few minutes.
3. Select the **Generate Bitstream** option and click **OK**.
4. In the **Generate Bitstream** dialog box, click **OK** to start generating the bitstream.
A Bitstream Generation Completed pop-up appears to let you know the process is finished.

Step 3: Using ChipScope Tools to Debug the Hardware

In this step, you learn:

- How to debug the design using ChipScope tools.
- How to discover and correct a circuit problem by making a small adjustment to the design.
- Some useful techniques for triggering and capturing design data.

Verifying Operation of the Sine Wave Generator

After doing some setup work, you will use ChipScope Pro Analyzer to verify that the sine wave generator is working correctly. The two primary objectives are to verify that:

- All sine wave selections are correct.
- The selection logic works correctly.

Setting Up

1. Ensure that your SP601 board is correctly set up:
 - USB cable should run from the USB JTAG connector on the board to the USB port on your system. (*Do not* use the *USB UART* connector on the board.)
 - The board is plugged in and powered on.
 - Turn DIP switch positions on SW8 (Debounce Enable) to the OFF position.
2. In the PlanAhead software, from the **Program and Debug** drop-down list, select **ChipScope Analyzer**.
3. In ChipScope Analyzer, configure the JTAG Chain to the USB cable and communication parameters:
 - a. Select **JTAG Chain > Xilinx Platform USB Cable**.
 - b. The **Platform USB Cable Parameters** dialog box opens. Verify that the speed is set to **3 MHz** and the Port to **USB21**.
4. Confirm the connection to the JTAG chain by verifying the information that appears in the bottom pane of the ChipScope Pro Analyzer window, as illustrated in [Figure 4, page 13](#).

Note: The ESN option number varies from what is shown. The number is unique to each board.

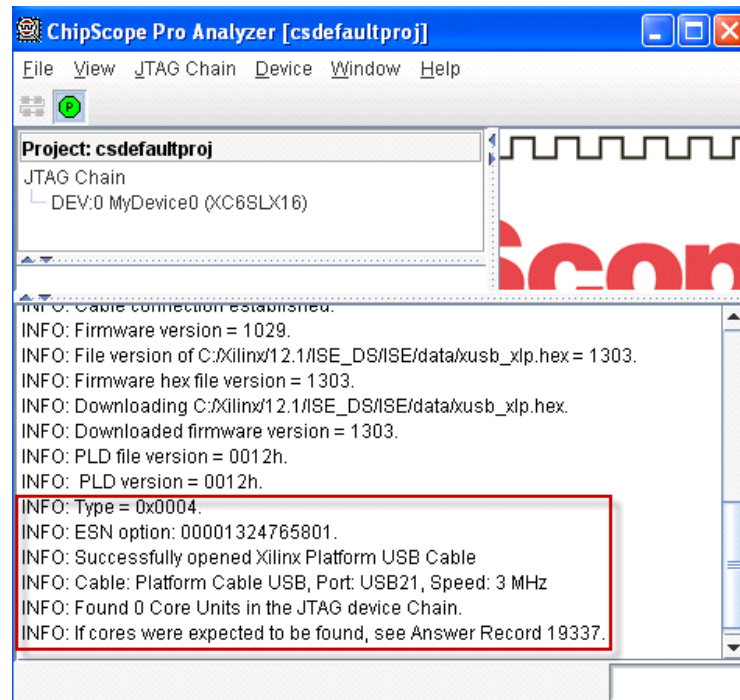


Figure 4: JTAG Chain Connection Data

5. Configure the device.
 - a. In the upper left pane of ChipScope Pro Analyzer, right-click the JTAG Chain device listed under **Project: csdefaultproj** and select **Configure**.

The JTAG Configuration dialog box opens. This dialog box lets you program the device with the BIT file you created earlier. The PlanAhead software tools provide the location of the BIT and CDC files to the ChipScope Pro Analyzer; consequently, leave all settings at their default values.

- b. You can now verify the device configuration and ILA core in the ChipScope Pro Analyzer main window, as shown in [Figure 5, page 14](#).

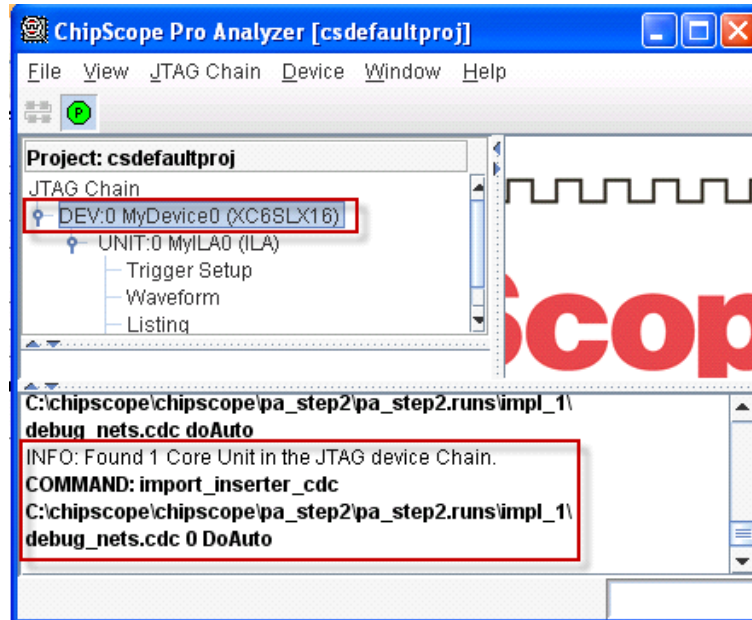


Figure 5: Device Configuration and ILA Core Information

Verifying Sine Wave Activity

1. With the item **DEV:0 MyDevice0 (XC6SLX16)** expanded, as shown in [Figure 5](#), double-click **Trigger Setup**. The Trigger Setup display appears in the upper right pane.
2. Double-click **Waveform** to add the Waveform display to the upper right pane.
3. In the tool bar menu, click **T!** to trigger immediately and capture data.
4. In the **Waveform** window, verify that there is activity on the sine signal.

Displaying the Sine Wave

1. With the item **DEV:0 MyDevice0 (XC6SLX16)** expanded, double-click **Bus Plot** to open the **Bus Plot** window.
2. In the **Bus Plot** window, select the **/sine** checkbox to display sine wave.
Notice that the waveform does not look like a sine wave. This is because you must change the radix setting from Hex to Signed Decimal, as described in the following subsection.

Correcting Display of the Sine Wave

To change the radix setting and correct the display, use the Trigger Immediate function (T! button) to view the high, mid, and low frequency sine wave bus plots and change the setting for each.

1. Select the waveform in the project panel.
2. In the **Signals** window, on the left side of the screen, select **/sine** and right-click to select **Bus Radix > Signed Decimal**. Review the information in the resulting **Decimal Values** dialog box, and then click **OK** to close it because scaling is not required in this case.
3. To verify that the sine wave selection state machine is working correctly, perform the steps shown in [Table 2, page 15](#), on your SP601 board. Refer to [Figure 1, page 6](#) to identify the board components cited in the table.

Note: As you sequence through the sine wave selections, you might notice that the LEDs do not light up in the expected order. You will debug this in the next section of this tutorial. For now, you will verify, for each LED selection, that the correct sine wave is displayed.

Table 2: Sequencing Through the Sine Wave Selections

Test Setup on the SP601 Board	Trigger and Capture Data in ChipScope Pro Analyzer	Verify Test Results
1. Verify that the sine wave selection indicators (LEDs) are both <i>off</i> (0,0). If they are not, push the <i>Sine Wave Sequencer</i> button until they are off.	Click T! (Trigger Immediately) to view the high frequency sine wave bus plot.	Verify that you see the high frequency sine wave in the Bus Plot viewer.
2. Push the <i>Sine Wave Sequencer</i> button on the board until the two <i>Sine Wave Selection</i> indicator LEDs are <i>off,on</i> (0,1).	Click T! to view the mid frequency sine wave bus plot.	Verify that you see the mid frequency sine wave in the Bus Plot viewer.
3. Push the <i>Sine Wave Sequencer</i> until the until the two <i>Sine Wave Selection</i> indicator LEDs are <i>on/off</i> (1,0).	Click T! and view the low frequency sine wave bus plot.	Verify that you see the low frequency sine wave in the Bus Plot viewer.
4. Push the <i>Sine Wave Sequencer</i> button on board until Sine Wave Selection indicator LEDs display on, on (1,1).	Click T! to view the combined sine wave bus plot.	Verify that you see the combined sine wave in the Bus Plot viewer.

Debugging the Sine Wave Sequencer State Machine

As you were correcting the sine wave display, the LEDs might not have lit up in sequence as you pressed the *Sine Wave Sequencer* button.

With each push of the button, there should be a single, cycle-wide pulse on the `GPIO_BUTTONS_re[1]` signal. If there is more than one, the behavior of the LEDs becomes irregular. In this section of the tutorial, you will use ChipScope to probe the sine wave sequencer state machine, and view and repair the root cause of the glitch.

Before starting the actual debug process, it is important to understand more about the sine wave sequencer state machine.

Sine Wave Sequencer State Machine Overview

The sine wave sequencer state machine selects one of the four sine waves to be driven onto the `sine` signal at the top-level of the design. The state machine has one input and one output. [Figure 6, page 16](#) shows the schematic elements of the state machine.

Refer to this diagram as you read the following description and as you perform the steps to view and repair the state machine glitch.

- The input is a scalar signal called `button`. When the button input equals '1', the state machine advances from one state to the next.
- The output is a two-bit signal vector called `Y`, and it indicates which of the four sine wave generators is selected.

The input signal `button` connects to the top-level signal `GPIO_BUTTONS_re[1]`, which is a low-to-high transition indicator on the *Sine Wave Sequencer* button (shown in [Figure 1, page 6](#)). The output signal `Y` connects to the top-level signal, `sineSel`, which selects the sine wave.

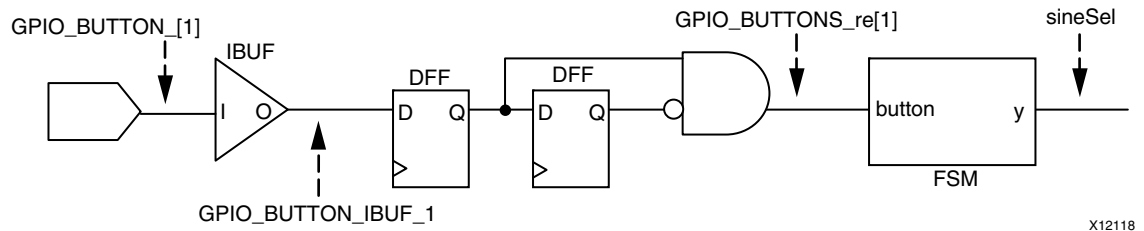


Figure 6: Sine Wave Sequencer Button Schematic

Viewing the State Machine Glitch

Viewing the Input to the State Machine

In this step, you will examine the input to the state machine (`GPIO_BUTTONS_re[1]`), shown in [Figure 6](#) to verify that this signal is causing the glitch. First, you must configure the trigger setup to take a measurement:

1. In the **Match** area of the **Trigger Setup** window, locate and expand the Match Unit row that contains `GPIO_BUTTONS_re[1]` and `GPIO_BUTTONS_re[0]`.
2. Select the **Value** column in the `GPIO_BUTTONS_re[1]` row, type **R** (rising edge), and press **Enter**. This sets the trigger port match unit to look for a single cycle-wide pulse on the `GPIO_BUTTONS_re[1]` signal.

Note: When the `GPIO_BUTTONS_re[1]` signal is assigned a value of '1', the sine wave sequencer state machine transitions from state to state.

3. In the **Trigger Conditions** area of the **Trigger Setup** window, click inside the **Trigger Condition Equation** field. In the resulting dialog box, ensure that the appropriate match unit is enabled. You can tell that the correct match unit is enabled if it is highlighted in blue, as shown in [Figure 7, page 17](#).

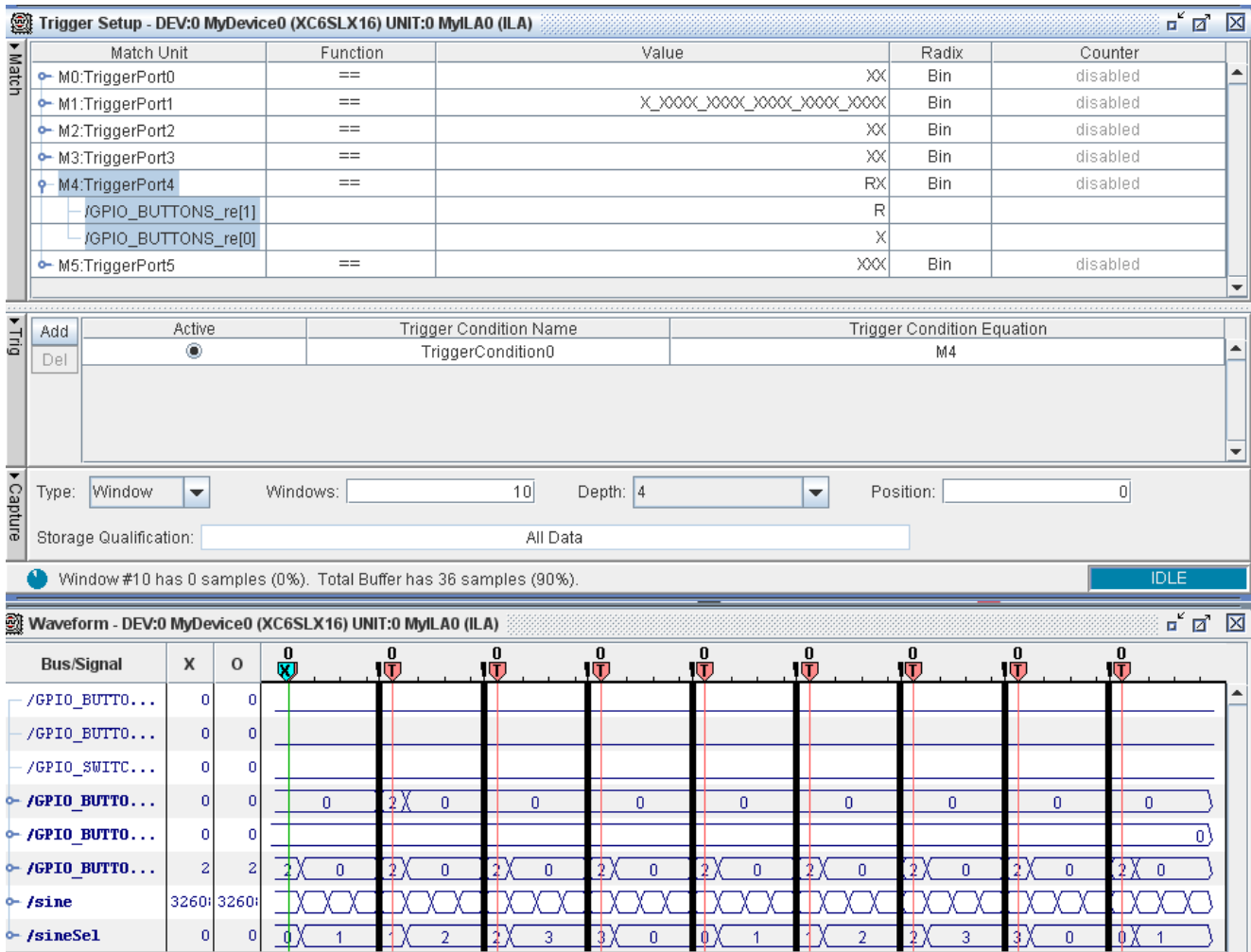


Figure 7: Sample View of Display Setting and State Machine Glitch

4. In the **Capture** area of the **Trigger Setup** window, shown in Figure 7, change the settings as follows:
 - **Windows** = 10
 - **Depth** = 4

Note: The Depth setting controls the number of samples per window.

Capturing and Viewing the Data

Note the toolbar buttons and names, shown in [Figure 8](#), when following the steps below.



Figure 8: Toolbar Buttons: Apply Settings and Arm Trigger, Stop Acquisition

1. In the toolbar, click the **Apply Settings** and the **Arm Trigger** button.
2. Ensure you have the **Trigger Setup** and **Waveform** windows displayed.
3. Push the *Sine Wave Sequencer* button on the SP601 board. Each button push generates one or more “windows” that show segments of time on the time axis of the waveform. Observe the number of windows captured in the status bar that appears just below the capture settings. (The status bar is shown in [Figure 7](#), page 17.) If more than one window is captured with a single push of the button, you have determined that there is an issue on the GPIO_BUTTON[1] input. If only one window is captured, try again. You might have to push the button several times to reproduce this intermittent glitch. As soon as a single button push generates multiple windows, go on to the next step.
4. In the toolbar, press the **Stop Acquisition** button and view the captured data. Notice that each of the multiple windows shows a low-to-high transition on the GPIO_BUTTON_re[1] signal. A single button push should only result in a single low-to-high transition. Again, this indicates that something is wrong with the GPIO_BUTTON[1] input signal.

Viewing the Button Input to the Design

You cannot troubleshoot the issue you identified above by connecting a debug probe to the GPIO_BUTTON[1] input signal itself. The GPIO_BUTTON[1] input signal is a PAD signal that is not directly accessible from the FPGA fabric. Instead, you must trigger on low-to-high transitions (rising edges) on the GPIO_BUTTON_IBUF signal (shown in [Figure 6](#), page 16), which is connected to the output of the input buffer of the GPIO_BUTTON[1] input signal.

1. In the **Match** area of the Trigger Setup window, locate and expand the Match Unit row that contains GPIO_BUTTONS_1_IBUF and GPIO_BUTTONS_0_IBUF.
2. Select the **Value** column in the GPIO_BUTTONS_1_IBUF row, type **R**, and press **Enter** to set the trigger port match unit to look for a single cycle-wide pulse on the GPIO_BUTTONS_IBUF signal.
3. In the **Trigger Conditions** area of the **Trigger Setup** window, click inside the **Trigger Condition Equation** field. In the resulting dialog box, ensure that the appropriate match unit is enabled.

As described earlier, the glitch reveals itself as multiple low-to-high transitions on the GPIO_BUTTONS_1_IBUF signal, but it occurs intermittently. Because it could take several button presses to detect it, you will now set up the ChipScope Pro Analyzer tool to **Repetitive Trigger Run Mode**. This setting makes it easier to repeat the button presses and look for the event in the **Waveform** viewer.

4. In the toolbar drop-down menu, set the **Trigger Run Mode** to **Repetitive**.
5. In the **Capture** area of the Trigger Setup window, change the settings as follows:
 - **Windows** = 1
 - **Depth** = 1024
 - **Position** = 512

6. Click the **Apply Settings and Arm Trigger** button.
7. On the board, press the *Sine Wave Sequencer* button until you see multiple transitions on the `GPIO_BUTTONS_1_IBUF` signal (this could take 20 or more tries). This is a visualization of the glitch that is occurring on the input. An example of the glitch is shown in [Figure 9](#).

Note: You might not observe signal glitches at exactly the same location as shown in the figure.

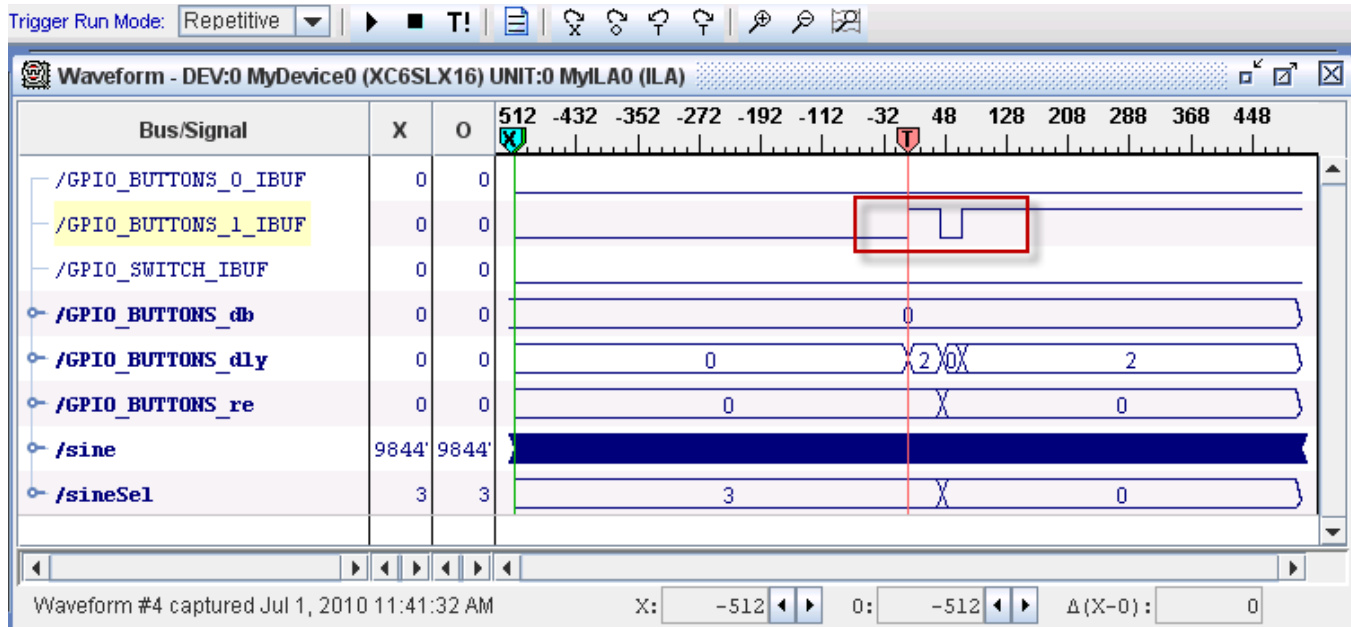


Figure 9: GPIO Buttons_1_re Signal Glitch

Fixing the Signal Glitch and Verifying the Correct State Machine Behavior

The multiple transition glitch or “bounce” occurs because the mechanical button is making and breaking electrical contact just as you press it. To eliminate this signal bounce, a “debouncer” circuit is required.

1. Enable the debouncer circuit by setting *DIP* switch position 1 on the SP601 board (labeled *Debounce Enable* in [Figure 1](#), page 6) to the *ON* position.
2. Repeat steps 6 and 7, above, to:
 - Ensure that you no longer see multiple transitions on the `GPIO_BUTTON_re[1]` signal on a single press of the Sine Wave Sequencer button.
 - Verify that the state machine is working correctly by ensuring that the `sineSel` signal transitions from 00 to 01 to 10 to 11 and back to 00 with each successive button press.

Additional Resources

Xilinx Resources

- *Spartan®-6 PCB Design Guide (UG393)*:
http://www.xilinx.com/support/documentation/user_guides/ug393.pdf
- **Xilinx® Glossary**:
http://www.xilinx.com/support/documentation/sw_manuels/glossary.pdf
- **Xilinx Support**: <http://www.xilinx.com/support>

ChipScope Documentation

- *ChipScope™ Pro Software and Cores User Guide (UG029)*:
http://www.xilinx.com/support/documentation/sw_manuels/xilinx13_3/chipscope_pro_sw_cores_ug029.pdf
- *Using Xilinx ChipScope Pro ILA Core with Project Navigator to Debug FPGA Applications (UG750)*:
http://www.xilinx.com/support/documentation/sw_manuels/xilinx13_3/ug750.pdf

PlanAhead Documentation

- *PlanAhead™ Software User Guide (UG632)*:
http://www.xilinx.com/support/documentation/sw_manuels/xilinx13_3/PlanAhead_UserGuide.pdf
- **PlanAhead Tutorials**:
http://www.xilinx.com/support/documentation/dt_planahead_planahead13-3_tutorials.htm
- *Quick Front-to-Back Flow Overview (UG673)*:
http://www.xilinx.com/support/documentation/sw_manuels/xilinx13_3/PlanAhead_Tutorial_Quick_Front-to-Back_Overview.pdf

Board Documentation

- Spartan-6 FPGA SP601 Evaluation Kit information:
<http://www.xilinx.com/products/devkits/EK-S6-SP601-G.htm>

