

Vivado Design Suite User Guide:

Programming and Debugging

UG908 (v2012.2) July 25, 2012



Notice of Disclaimer

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

© Copyright 2012 Xilinx, Inc. Xilinx, the Xilinx logo, Artix, ISE, Kintex, Spartan, Virtex, Vivado, Zynq, and other designated brands included herein are trademarks of Xilinx in the United States and other countries. All other trademarks are the property of their respective owners.

Revision History

The following table shows the revision history for this document.

Date	Version	Revision
07/25/12	1.0	Initial Xilinx release.

Table of Contents

Revision History	2
Chapter 1: Introduction	
Getting Started	5
Chapter 2: Programming the Device	
Introduction	6
Generating the Bitstream	6
Changing the Bitstream File Format Settings	7
Changing Device Configuration Bitstream Settings	8
Programming the FPGA Device	8
Launching iMPACT	8
Using a Vivado Hardware Session to Program an FPGA Device	9
Chapter 3: Debugging the Design	
Introduction	15
RTL-level Design Simulation	15
Post-Implemented Design Simulation	16
In-System Debugging	16
Chapter 4: In-System Debugging Flows	
Introduction	17
Probing the Design for In-System Debugging	17
Using the Netlist Insertion Debug Probing Flow	18
HDL Instantiation Debug Probing Flow Overview	27
Using the HDL Instantiation Debug Probing Flow	27
Implementing the Design Containing the Debug Cores	31
Chapter 5: Debugging the Design in Hardware	
Introduction	33
Launching ChipScope Pro Analyzer to Debug the Design	33
Using Vivado Logic Analyzer to Debug the Design	34
Connecting to the Hardware Target and Programming the FPGA Device	34

Setting up the ILA Core to Take a Measurement	35
Setting the ILA Core Trigger Condition	35
Setting the ILA Core Trigger Position	35
Writing ILA Probes Information	36
Reading ILA Probes Information	36
Viewing ILA Probes	37
Setting Up the ILA Probe Trigger Compare Values	37
ILA Probe Compare Value Settings	38
Running or Arming the ILA Core Trigger	39
Stopping the ILA Core Trigger	40
Viewing the ILA Core State	40
Viewing Captured Data from the ILA Core in the Waveform Viewer	41
Saving and Restoring Captured Data from the ILA Core	41
Using Vivado Logic Analyzer in a Lab Environment	41
Using Vivado Logic Analyzer and ChipScope™ Pro Analyzer Simultaneously	43
Description of Hardware Session Tcl Objects and Commands	45
Using Hardware Session Tcl Commands	48

Chapter 6: Viewing ILA Probe Data Using Waveform Viewer

Introduction	49
Waveform Viewer Features and Limitations	49
Waveform Viewer Configuration Signals and Buses	50
ILA Probes in Waveform Configuration	51
Customizing the Wave Configuration	52
Renaming Objects	56
About Radixes and Analog Waveforms	59
Zoom Gestures	63

Appendix A: Device Configuration Bitstream Settings

Scope	64
-----------------	----

Appendix B: Additional Resources

Xilinx Resources	70
Solution Centers	70
References	70

Introduction

Getting Started

After successfully implementing your design, the next step is to run it in hardware by programming the FPGA device and debugging the design in-system. All of the necessary commands to perform programming of FPGA devices and in-system debugging of the design are in the **Program and Debug** section of the **Flow Navigator** window in the Vivado™ Integrated Design Environment (IDE) (see [Figure 1-1](#))

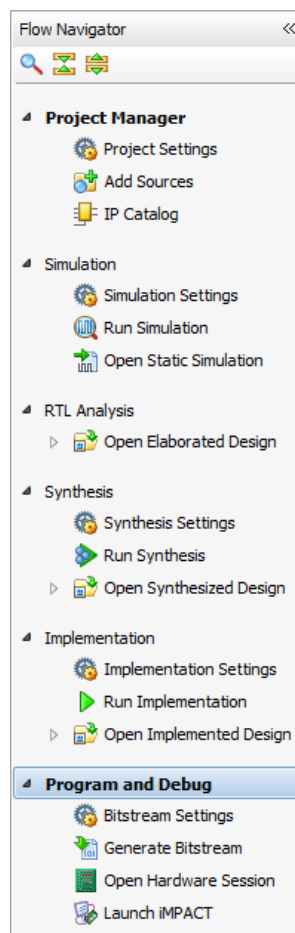


Figure 1-1: Program and Debug section of the Flow Navigator panel

Programming the Device

Introduction

The hardware programming phase is broken into two steps:

1. Generating the bitstream data programming file from the implemented design.
 2. Connecting to hardware and downloading the programming file to the target FPGA device.
-

Generating the Bitstream

Before generating the bitstream data file, it is important to review the bitstream settings to make sure they are correct for your design.

There are two types of bitstream settings in Vivado™ IDE:

1. Bitstream file format settings.
2. Device configuration settings.

The **Bitstream Settings** button in the **Flow Navigator** and the **Flow > Bitstream Settings** menu selection opens the **Bitstream** section in the **Project Settings** popup window (see [Figure 2-1](#)). Once the bitstream settings are correct, the bitstream data file can be generated using the `write_bistream` Tcl command or by using the **Generate Bitstream** button in the **Flow Navigator**.

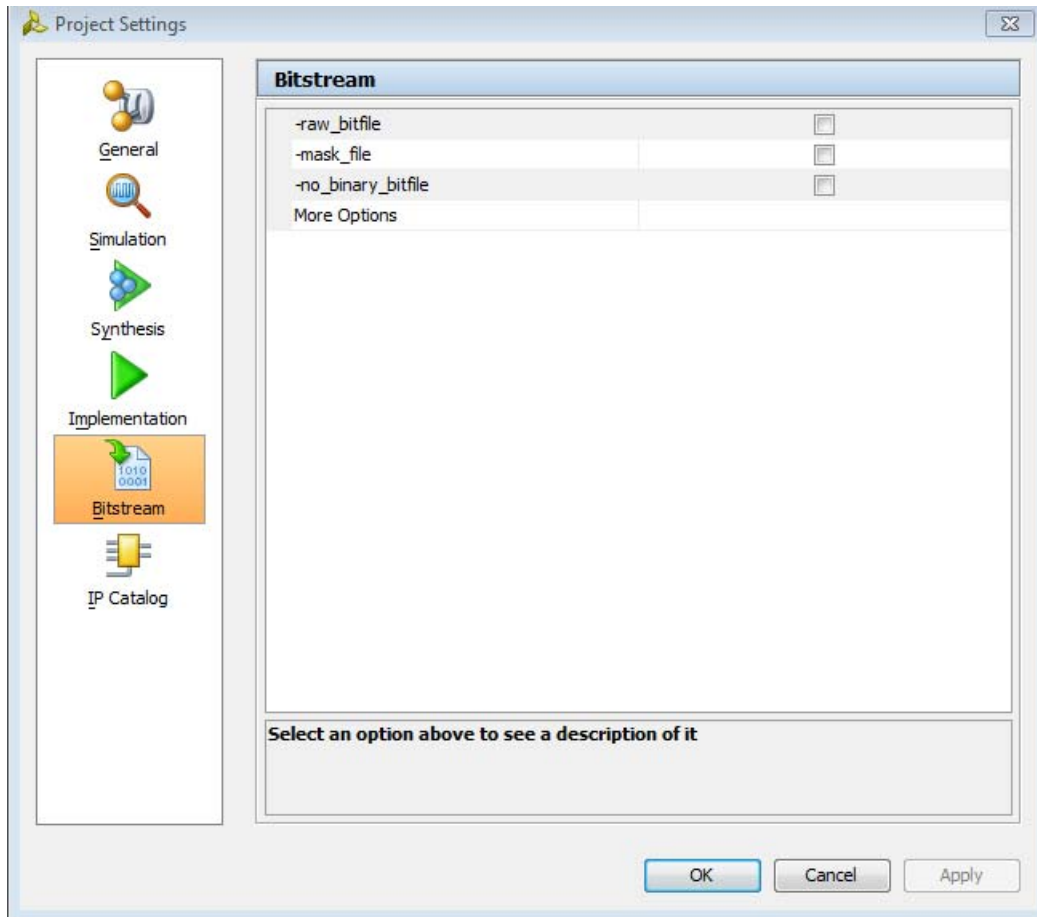


Figure 2-1: Bitstream settings panel

Changing the Bitstream File Format Settings

By default, the `write_bitstream` Tcl command will generate a binary bitstream (`.bit`) file only. You can optionally change the file format(s) written out by the `write_bitstream` Tcl command by using the following command switches:

- `-raw_bitfile`: Causes `write_bitstream` to write a raw bit file (`.rbt`) which contains the same information as the binary bitstream file, but is in ASCII format. The output file will be named `filename.rbt`
- `-mask_file`: Write a mask file (`.msk`), which has mask data where the configuration data is in the bitstream file. This file determines which bits in the bitstream should be compared to readback data for verification purposes. If a mask bit is 0, that bit should be verified against the bitstream data. If a mask bit is 1, that bit should not be verified. The output file will be named `file.msk`.

- `-no_binary_bitfile`: Do not write the binary bitstream file (.bit). Use this command when you want to generate the ASCII bitstream or mask file, or to generate a bitstream report, without generating the binary bitstream file.

Changing Device Configuration Bitstream Settings

The most common configuration settings that you can change fall into the device configuration settings category. These settings are properties on the device model and you change them by using the `set_property` command in an XDC file. For instance, here is an example on how to change the start-up DONE cycle property:

```
set_property BITSTREAM.STARTUP.DONE_CYCLE 4 [current_design]
```

Additional examples and templates are provided in the Vivado Templates. [Appendix A, Device Configuration Bitstream Settings](#) describes all of the device configuration settings.

Programming the FPGA Device

The next step after generating the bitstream data programming file is to download it into the target FPGA device. The Vivado tool offers two different ways to do this:

- Launch the iMPACT device programmer tool using the **Launch iMPACT** button in the **Flow Navigator** or **Flow > Launch iMPACT** menu option.
- Open a hardware session to use the native in-system device programming capabilities that are built into the Vivado IDE.

Launching iMPACT

The iMPACT tool lets you perform device configuration and file generation.

- Device Configuration lets you directly configure Xilinx® FPGAs and PROMs with the JTAG download cables (Xilinx Parallel Cable IV, Xilinx Platform Cable USB, Xilinx Platform Cable USB II, or Digilent JTAG cables).
- Operating in Boundary-Scan mode, iMPACT can configure or program Xilinx FPGAs, CPLDs, and PROMs.
- File generation enables you to create the following programming file types: System ACE™ CF, PROM, SVF, STAPL, and XSVF files.

iMPACT also lets you:

- Readback and verify design configuration data.
- Debug configuration problems.
- Execute SVF and XSVF files.

You can launch the iMPACT software tool directly from the Vivado IDE on any implemented design on which the Generate Bitstream command has been run. To invoke iMPACT, in the **Flow Navigator**, select **Launch iMPACT**.

The BIT bitstream file is passed automatically to iMPACT when launched from the Vivado tool. For more information on using iMPACT, see the iMPACT Help.

Using a Vivado Hardware Session to Program an FPGA Device

The Vivado IDE tool includes functionality that allows you to connect to hardware containing one or more FPGA devices in order to program and interact with those FPGA devices. Connecting to hardware can be done from either the Vivado IDE graphical user interface or by using Tcl commands. In either case, the steps to connect to hardware and program the target FPGA device are the same:

1. Open a hardware session.
2. Open a hardware target that is managed by a hardware server running on a host computer.
3. Associate the bitstream data programming file with the appropriate FPGA device.
4. Program or download the programming file into the hardware device.

Opening a Hardware Session

Opening a hardware session is the first step in programming and/or debugging your design in hardware. To open a hardware session, do one of the following:

- Click on the **Open Hardware Session** button in the **Program and Debug** section of the **Flow Navigator**.
- Select the **Flow > Open Hardware Session** menu option.

Opening Hardware Target Connections

The next step in opening a hardware target (for instance, a hardware board containing a JTAG chain of one or more FPGA devices) is connecting to the hardware server (also called the CSE server) that is managing the connection to the hardware target. You can do this one of three ways:

- Use the **Open New Hardware Target** selection in the **Hardware** view to use a wizard-based flow to open a new connection to a hardware target.
- Use the **Open Recent Hardware Target** selection in the **Hardware** view to re-open a connection to a previously connected hardware target.
- Use Tcl commands to open a connection to a hardware target.

Opening a New Hardware Target

The **Open New Hardware Target** wizard provides an interactive way for you to connect to a hardware server and target. The wizard is a three-step process:

1. Specify or select the host name and port of the CSE server (also called hardware server or cse_server) that is managing the hardware targets on the machine to which the target board is connected (see [Figure 2-2](#)).

Note: If you use “localhost” as the host name, a `cse_server` process will automatically be started on the machine on which you are running the Vivado tool and will be used in the subsequent panels of the wizard.

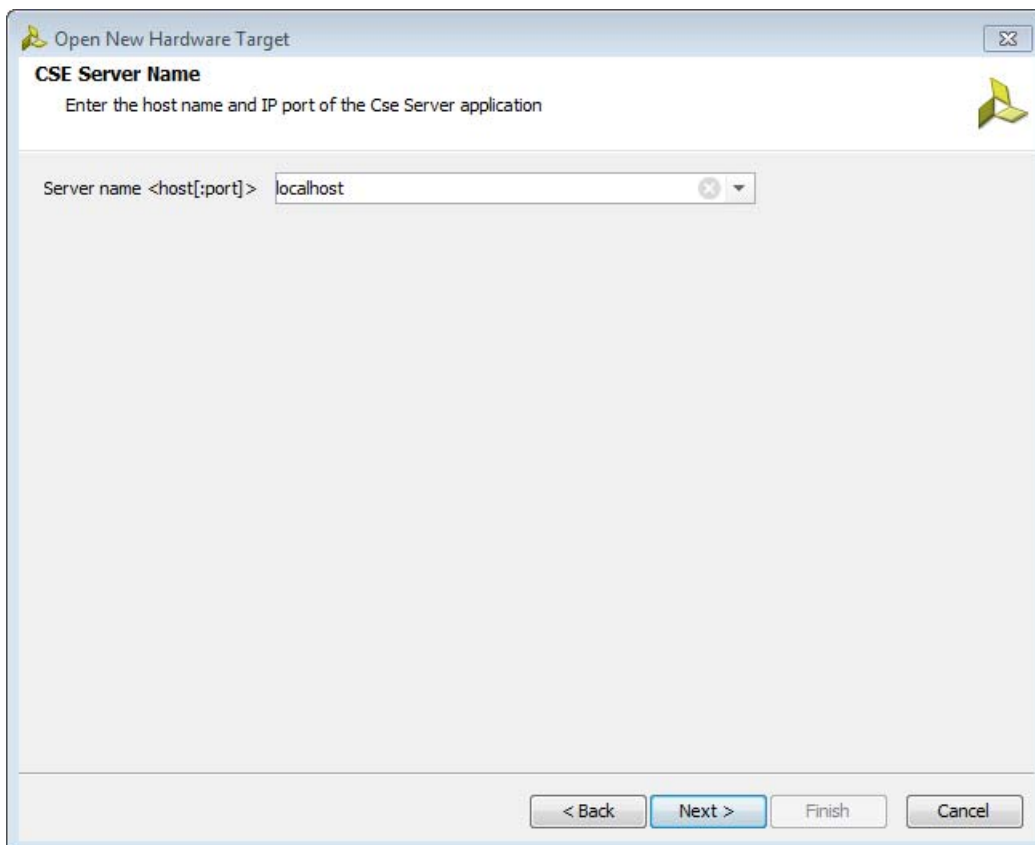


Figure 2-2: Specifying the CSE Server Name

2. Select the appropriate hardware target from the list of targets that are managed by the hardware server. Note that when you select a target, you will see the various hardware

devices that are available on that hardware target (see [Figure 2-3](#)).

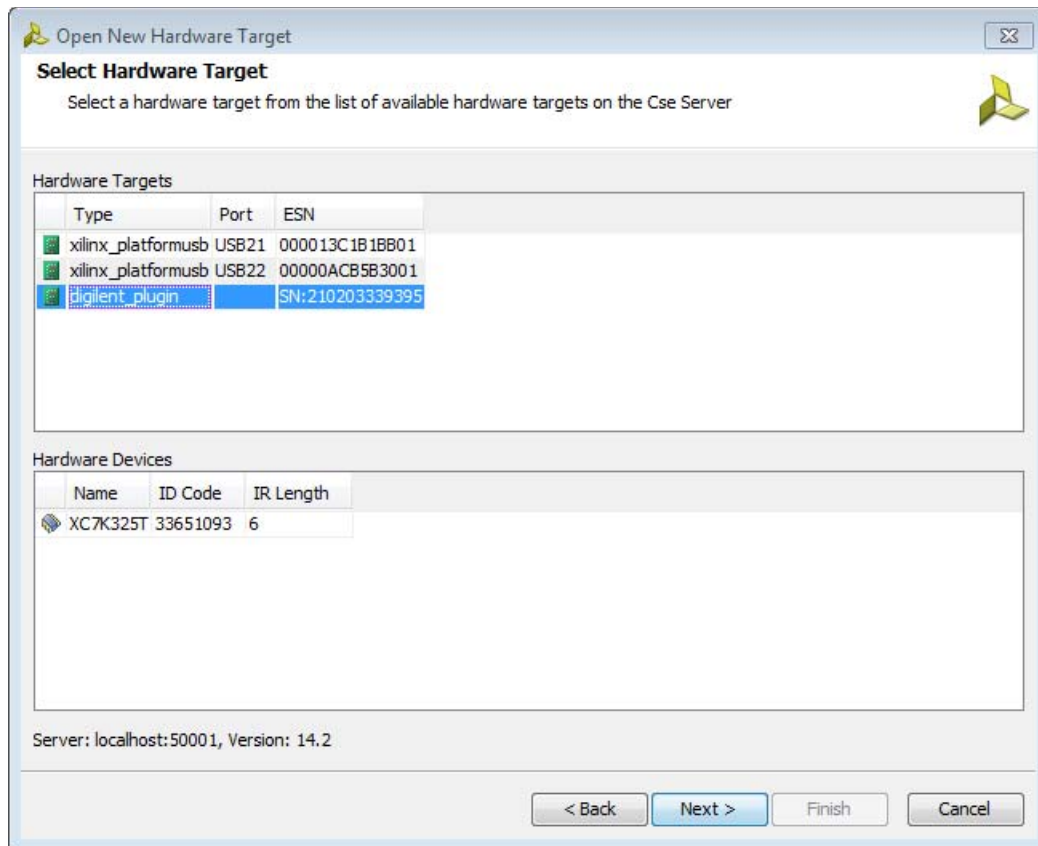


Figure 2-3: Selecting the Hardware Target

- Set the properties of the hardware target, such as the frequency of the TCK clock pin, etc. Note that each type of hardware target may have different properties. Refer to the [documentation](#) of each hardware target for more information about these properties.

Opening a Recent Hardware Target

The **Open New Hardware Target** wizard is also what populates the list of previously connected hardware targets to be used by the **Open Recent Hardware Target** selection. Instead of connecting to a hardware target by going through the wizard, you can also re-open a connection to a previously connected hardware target by selecting the **Open Recent Hardware Target** link in the **Hardware** window and selecting one of the recently connected hardware server/target combinations in the list.

Opening a Hardware Target Using Tcl Commands

You can also use Tcl commands to connect to a hardware server/target combination. For instance, to connect to the `digilent_plugin` target (serial number 210203339395) that is managed by the `cse_server` running on `localhost:50001`, use the following Tcl commands:

```
connect_hw_server -host localhost -port 50001
current_hw_target [get_hw_targets */digilent_plugin/SN:210203339395]
open_hw_target
```

Once you finish opening a connection to a hardware target, the **Hardware** window will be populated with the hardware server, hardware target, and various hardware devices for the open target (see [Figure 2-4](#)).

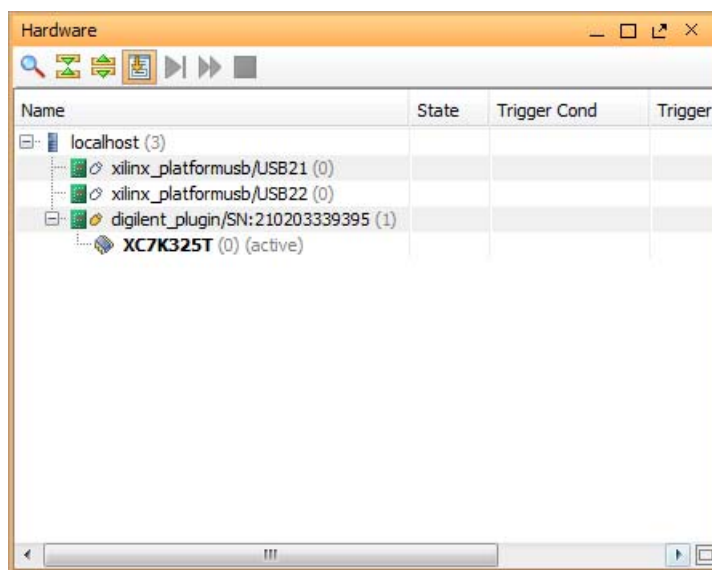


Figure 2-4: Hardware View after Opening a Connection to the Hardware Target

Associating a Programming File with the Hardware Device

After connecting to the hardware target and before you program the FPGA device, you need to associate the bitstream data programming file with the device. Select the hardware device in the **Hardware** window and make sure the **Programming file** property in the **Properties** window is set to the appropriate bitstream data (`.bit`) file.

Note: As a convenience, Vivado IDE automatically uses the `.bit` file for the current implemented design as the value for the **Programming File** property of the first matching device in the open hardware target.

You can also use the `set_property` Tcl command to set the `PROGRAM.FILE` property of the hardware device:

```
set_property PROGRAM.FILE {C:/design.bit} [lindex [get_hw_devices] 0]
```

Programming the Hardware Device

Once the programming file has been associated with the hardware device, you can program the hardware device using by right-clicking on the device in the **Hardware** view and selecting the **Program Device** menu option. You can also use the `program_hw_device` Tcl command. For instance, to program the first device in the JTAG chain, use the following Tcl command:

```
program_hw_devices [lindex [get_hw_devices] 0]
```

Once the progress dialog has indicated that the programming is 100% complete, you can check that the hardware device has been programmed successfully by examining the DONE status in the Hardware Device Properties view (see [Figure 2-5](#)).

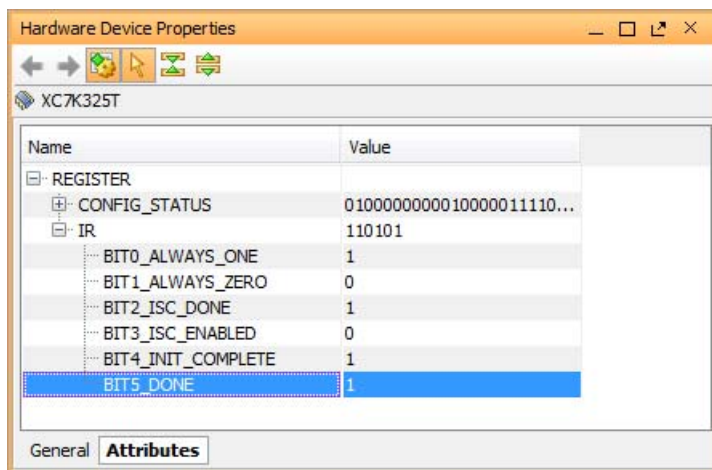


Figure 2-5: Checking the DONE status of an FPGA device

You can also use the `get_property` Tcl command to check the DONE status. For instance, to check the DONE status of a Kintex™-7 device that is the first device in the JTAG chain, use the following Tcl command:

```
get_property REGISTER.IR.BIT5_DONE [lindex [get_hw_devices] 0]
```

If you use another means to program the hardware device (for instance, a Flash device or external device programmer such as the iMPACT tool), you can also refresh the status of a hardware device by right-clicking the **Refresh Device** menu option or by running the `refresh_hw_device` Tcl command. This will refresh the various properties for the device, including but not limited to the DONE status.

Closing the Hardware Target

You can close a hardware target by right-clicking on the hardware target in the **Hardware** window and selecting **Close Target** from the popup menu. You can also close the hardware target using a Tcl command. For instance, to close the xilinx_platformusb/USB21 target on the localhost server, use the following Tcl command:

```
close_hw_target {localhost/xilinx_platformusb/USB21}
```

Closing a Connection to the Hardware Server

You can close a hardware server by right-clicking on the hardware server in the **Hardware** window and selecting **Close Server** from the popup menu. You can also close the hardware server using a Tcl command. For instance, to close the connection to the localhost server, use the following Tcl command:

```
disconnect_hw_server localhost
```

Debugging the Design

Introduction

Debugging an FPGA design is a multistep, iterative process. Like most complex problems, it is best to break the FPGA design debugging process down into smaller parts by focusing on getting smaller sections of the design working one at a time rather than trying to get the whole design to work at once. Iterating through the design flow by adding one module at a time and getting it to function properly in the context of the whole design is one example of a proven design and debug methodology. You can use this design and debug methodology in any combination of the following design flow stages:

- RTL-level design simulation
 - Post-implemented design simulation
 - In-system debugging
-

RTL-level Design Simulation

The design can be functionally debugged during the simulation verification process. Xilinx provides a full design simulation feature in the Vivado™ IDE. The Vivado design simulator can be used to perform RTL simulation of your design. The benefits of debugging your design in an RTL-level simulation environment include full visibility of the entire design and ability to quickly iterate through the design/debug cycle. The limitations of debugging your design using RTL-level simulation includes the difficulty of simulating larger designs in a reasonable amount of time in addition to the difficulty of accurately simulating the actual system environment. For more information about using the Vivado simulator, please refer to the *Vivado Design Suite User Guide: Logic Simulation (UG937)* [Ref 1].

Post-Implemented Design Simulation

The Vivado simulator can also be used to simulate the post-implemented design. One of the benefits of debugging the post-implemented design using the Vivado simulator includes having access to a timing-accurate model for the design. The limitations of performing post-implemented design simulation include those mentioned in the previous section: long run-times and system model accuracy.

In-System Debugging

The Vivado IDE also includes a logic analysis feature that enables you to perform in-system debugging of the post-implemented design on an FPGA device. The benefits for debugging your design in-system include debugging your timing-accurate, post-implemented design in the actual system environment at system speeds. The limitations of in-system debugging include somewhat lower visibility of debug signals compared to using simulation models and potentially longer design/implementation/debug iterations, depending on the size and complexity of the design.

In general, the Vivado tool provides several different ways to debug your design. You can use one or more of these methods to debug your design, depending on your needs. [Chapter 4, In-System Debugging Flows](#) will focus on the in-system logic debugging capabilities of the Vivado IDE.

In-System Debugging Flows

Introduction

The Vivado tool provides many features to debug a design in-system in an actual hardware device. The in-system debugging flow has three distinct phases:

1. **Probing phase:** Identifying what signals in your design you want to probe and how you want to probe them.
2. **Implementation phase:** Implementing the design that includes the additional debug IP that is attached to the probed nets.
3. **Analysis phase:** Interacting with the debug IP contained in the design to debug and verify functional issues.

This in-system debug flow is designed to work using the iterative design/debug flow described in the previous section. If you choose to use the in-system debugging flow, it is advisable to get a part of your design working in hardware as early in the design cycle as possible. The rest of this chapter describes the three phases of the in-system debugging flow and how to use the Vivado™ logic debug feature to get your design working in hardware as quickly as possible.

Probing the Design for In-System Debugging

The probing phase of the in-system debugging flow is split into two steps:

1. Identifying signals or nets that you want to probe
2. Deciding how you want to add debug core(s) to your design

In many cases, the decision you make on what signals to probe or how to probe them can affect one another. It helps to start by deciding if you want to manually add the debug IP component instances to your design source code (called the HDL instantiation probing flow) or if you want the Vivado tool to automatically insert the debug cores into your post-synthesis netlist (called the netlist insertion probing flow). [Table 4-1](#) describes some of the advantages and trade-offs of the different debugging approaches.

Table 4-1: Debugging Strategies

Debugging Goal	Recommended Debug Programming Flow
Identify debug signals in the HDL source code while retaining flexibility to enable/disable debugging later in the flow.	<ul style="list-style-type: none"> Use mark_debug property to tag signals for debugging in HDL. Use the Set up Debug wizard to guide you through the Netlist Insertion probing flow.
Identify debug nets in synthesized design netlist without having to modify the HDL source code.	<ul style="list-style-type: none"> Use the Mark Debug right-click menu option to select nets for debugging in the synthesized design netlist. Use the Set up Debug wizard to guide you through the Netlist Insertion probing flow.
Automated debug probing flow using Tcl commands.	<ul style="list-style-type: none"> Use set_property Tcl command to set the mark_debug property on debug nets. Use Netlist Insertion probing flow Tcl commands to create debug core(s) and connect to them to debug nets.
Explicitly attach signals in the HDL source to an ILA debug core instance.	<ul style="list-style-type: none"> Identify HDL signals for debugging. Use the HDL Instantiation probing flow to generate and instantiate an Integrated Logic Analyzer (ILA) core and connect it to the debug signals in the design.

Using the Netlist Insertion Debug Probing Flow

Insertion of debug cores in the Vivado tool is presented in a layered approach to address different needs of the diverse group of Vivado users:

- The highest level is a simple wizard that creates and configures Integrated Logic Analyzer (ILA) v2.0 cores automatically based on the selected set of nets to debug.
- The next level is the main **Debug** window allowing control over individual debug cores, ports and their properties. The **Debug** window can be displayed by selecting the **Debug layout** from the **Layout Selector** or the **Layers** menu, or can be opened directly using **Window > Debug**.
- The lowest level is the set of Tcl debug commands that you can enter manually or replay as a script.

You can also use a combination of the modes to insert and customize debug cores.

Marking HDL Signals for Debug

You can identify signals for debugging at the HDL source level prior to synthesis by using the mark_debug constraint. Nets corresponding to signals marked for debug in HDL are automatically listed in the **Debug** window under the **Unassigned Debug Nets** view.

The procedure for marking nets for debug depends on whether you are working with an RTL source-based project or a synthesized netlist-based project. For an RTL netlist-based project:

- Using the Vivado synthesis feature you can optionally mark HDL signals for debug using the `mark_debug` constraint in VHDL and Verilog source files. The valid values for the `mark_debug` constraint are "TRUE" or "FALSE". The Vivado synthesis feature does not support the "SOFT" value.
- Using Xilinx Synthesis Technology (XST) you can optionally mark nets for debug using the `mark_debug` constraint in VHDL and Verilog sources. In addition to the boolean string values of, "TRUE" or "FALSE," a value of "SOFT" allows the software to optimize the specified net, if possible.

For a synthesized netlist-based project:

- Using the Synopsys® Synplify® synthesis tool, you can optionally mark nets for debug using the `mark_debug` and `syn_keep` constraints in VHDL or Verilog, or using the `mark_debug` constraint alone in the Synopsys Design Constraints (SDC) file. Synplify does not support the "SOFT" value, as this behavior is controlled by the `syn_keep` attribute.
- Using the Mentor Graphics® Precision® synthesis tool, you can optionally mark nets for debug using the `mark_debug` constraint in VHDL or Verilog.

The following subsections provide syntactical examples for Vivado synthesis, XST, Synplify, and Precision source files.

Vivado Synthesis `mark_debug` Syntax Examples

The following are examples of VHDL and Verilog syntax when using Vivado synthesis.

- VHDL Syntax Example

```
attribute mark_debug : string;
attribute mark_debug of char_fifo_dout: signal is "true";
```

- Verilog Syntax Example

```
(* mark_debug = "true" *) wire [7:0] char_fifo_dout;
```

XST `mark_debug` Syntax Examples

The following are examples of VHDL and Verilog syntax when using XST.

- VHDL Syntax Example

```
attribute mark_debug : string;
attribute mark_debug of char_fifo_dout: signal is "true";
```

- Verilog Syntax Example

```
(* mark_debug = "true" *) wire [7:0] char_fifo_dout;
```

Synplify mark_debug Syntax Examples

The following are examples of Synplify syntax for VHDL, Verilog, and SDC.

- VHDL Syntax Example

```
attribute syn_keep : boolean;
attribute mark_debug : string;
attribute syn_keep of char_fifo_dout: signal is true;
attribute mark_debug of char_fifo_dout: signal is "true";
```

- Verilog Syntax Example

```
(* syn_keep = "true", mark_debug = "true" *) wire [7:0] char_fifo_dout;
```

- SDC Syntax Example

```
define_attribute {n:char_fifo_din[*]} {mark_debug} {"true"}
```



IMPORTANT: Net names in an SDC source must be prefixed with the "n:" qualifier.

Note: Synopsys Design Constraints (SDC) is an accepted industry standard for communicating design intent to tools, particularly for timing analysis. A reference copy of the SDC specification is available from Synopsys by registering for the TAP-in program at:

<http://www.synopsys.com/Community/Interoperability/Pages/TapinSDC.aspx>

Precision mark_debug Syntax Examples

The following are examples of VHDL and Verilog syntax when using Precision.

- VHDL Syntax Example

```
attribute mark_debug : string;
attribute mark_debug of char_fifo_dout: signal is "true";
```

- Verilog Syntax Example

```
(* mark_debug = "true" *) wire [7:0] char_fifo_dout;
```

Synthesizing the Design

The next step is to synthesize the design containing the debug core(s) by clicking **Run Synthesis** in the Vivado IDE or by running the following Tcl commands:

```
launch_runs synth_1
wait_on_run synth_1
```

You can also use the synth_design Tcl command to synthesize the design. Refer to the *Vivado Design Suite User Guide: Synthesis (UG901)* [Ref 2] for more details on the various ways you can synthesize your design.

Marking Nets for Debug in the Synthesized Design

Open the synthesized design by clicking **Open Synthesized Design** in the **Flow Navigator** and select the **Debug** window layout to see the **Debug** window. Any nets that correspond to HDL signals that were marked for debugging are shown in the **Unassigned Debug Nets** folder in the **Debug** window (see [Figure 4-1](#)).

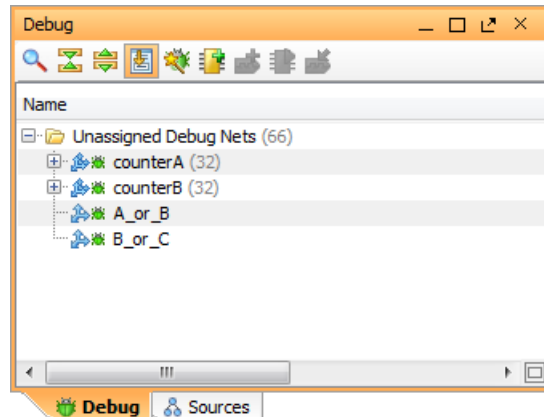


Figure 4-1: Unassigned Debug Nets

You can mark additional nets in the synthesized design netlist for debugging using any of three methods:

1. Selecting a net in any of the design views (such as the **Netlist** or **Schematic** windows), then right-click select the **Mark Debug** option.
2. Selecting a net in any of the design views, then dragging and dropping the nets into the **Unassigned Debug Nets** folder.
3. Using the net selector in the Set up Debug wizard (see [Using the Set Up Debug Wizard to Insert Debug Cores](#) for details).

Using the Set Up Debug Wizard to Insert Debug Cores

The next step after marking nets for debugging is to assign them to debug cores. The Vivado IDE provides an easy to use Set up Debug wizard to help guide you through the process of automatically creating the debug cores and assigning the debug nets to the inputs of the cores.

To use the Set up Debug wizard to insert the debug cores:

1. Optionally, select a set of nets for debugging either using the unassigned nets list or direct net selection.
2. Select **Tools > Set up Debug** from the Vivado IDE main menu.
3. Click **Next** to get to the **Specify Nets to Debug** panel (see [Figure 4-2](#)).

4. Optionally, click **Add/Remove Nets** to add more nets or remove existing nets from the table.
5. Right-click on a debug net and select **Select Clock Domain** to change the clock domain that will be used to sample value on the net.

Note: The Set up Debug wizard will attempt to automatically select the appropriate clock domain for the debug net by searching the path for synchronous elements. Use the **Select ClockDomain** dialog window to modify this selection as needed, but be aware that each clock domain present in the table will result in a separate ILA v2.0 core instance.

6. Once you are satisfied with the debug net selection, click **Next**.

Note: The ChipScope wizard inserts one ILA core per clock domain. The nets that were selected for debug are assigned automatically to the probe ports of the inserted ILA v2.0 cores. The last wizard screen shows the core creation summary displaying the number of clocks found and ILA cores to be created and/or removed.

7. If you are satisfied with the results, click **Finish** to insert and connect the ILA v2.0 cores in your synthesized design netlist.

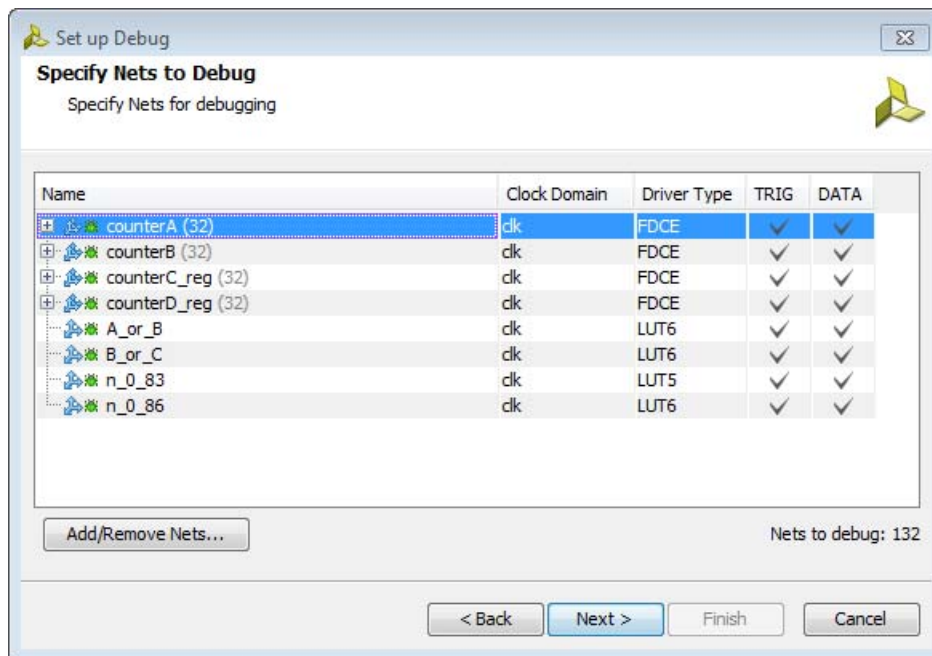


Figure 4-2: Set Up Debug Wizard

Using the Debug Window to Add and Customize Debug Cores

The **Debug** window provides more fine-grained control over ILA v2.0 core insertion than what is available in the Set up Debug wizard. The controls available in this window allow core creation, core deletion, debug net connection, and core parameter changes.

The **Debug** window:

- Shows the list of debug cores that are connected to the `debug_core_hub` core.
- Maintains the list of unassigned debug nets at the bottom of the window.

You can manipulate debug cores and ports from the popup menu or the toolbar buttons on the top of the window.

Creating and Removing Debug Cores

To create debug cores in the **Debug** window, click **Create Debug Core**. Using this interface (see [Figure 4-3](#)), you can change the parent instance, debug core name, and set parameters for the core. To remove an existing debug core, right-click on the core in the **Debug** window and select **Delete**. Click **Finish** to insert and connect the ILA v2.0 cores in your synthesized design netlist.

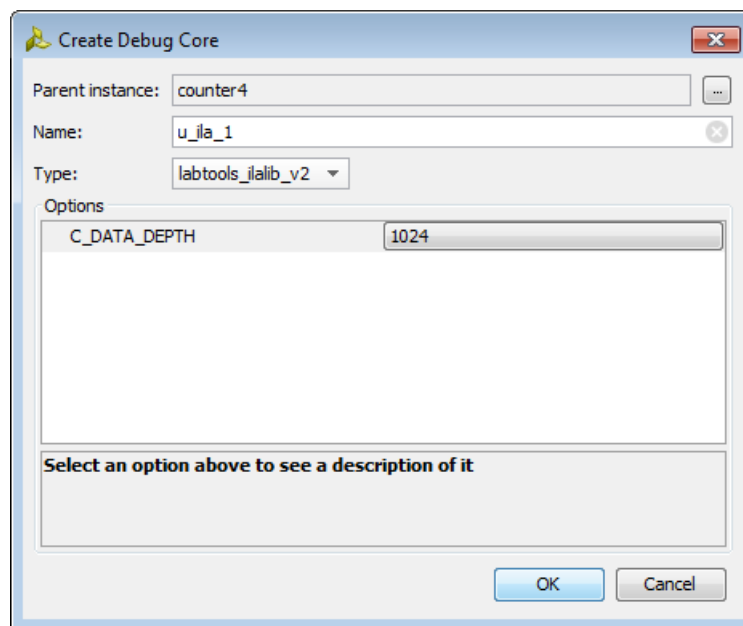


Figure 4-3: Creating a New Debug Core

Adding, Removing, and Customizing Debug Core Ports

In addition to adding and removing debug cores, you can also add, remove, and customize ports of each debug core to suit your debugging needs. To add a new debug port:

1. Select the debug core in the **Debug** window.
2. Click **Create Debug Port** to open the dialog shown in Figure 4-4.
3. Select or type in the port width
4. Click **OK**.
5. To remove a debug port, first select the port on the core in the **Debug** window, then select **Delete**.

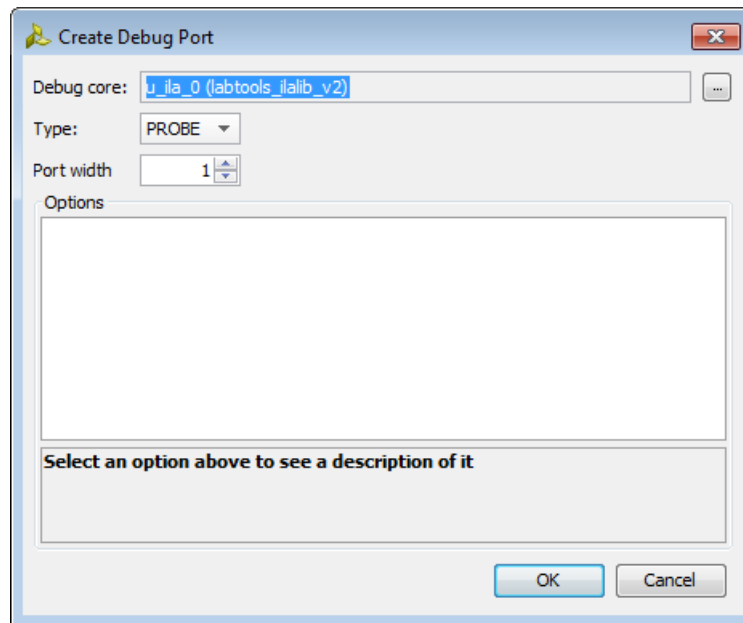


Figure 4-4: Creating a New Debug Port

Connecting and Disconnecting Nets to Debug Cores

You can select, drag, and drop nets and buses (also called bus nets) from the **Schematic** or **Netlist** windows onto the debug core ports. This expands the debug port as needed to accommodate the net selection. You can also right-click on any net or bus, and select **Assign to Debug Port**.



IMPORTANT: You should avoid connecting multiple different nets or buses to the same probe port of the ILA v2.0 core due to a limitation in the way these aggregated probe ports are handled in the Vivado logic analyzer. Instead, you should allocate a single probe port for each unique net or bus.

To disconnect nets from the debug core port, select the nets that are connected to the debug core port, and click **Disconnect Net**.

Modifying Properties on the Debug Cores

Each debug core has properties you can change to customize the behavior of the core. To learn how to change properties on the `debug_core_hub` debug core, refer to [Changing the BSCAN User Scan Chain of the Debug Core Hub](#), page 30.

You can also change properties on the ILA v2.0 debug core. For instance, to change the number of samples captured by the ILA v2.0 debug core (see [Figure 4-5](#)), do the following:

1. In the **Debug** window, select the desired ILA core (such as `u_ila_0`).
2. In the **Instance Properties** window, select the **Debug Core Options** view.
3. Using the `C_DATA_DEPTH` pull-down list, select the desired number of samples to be captured.

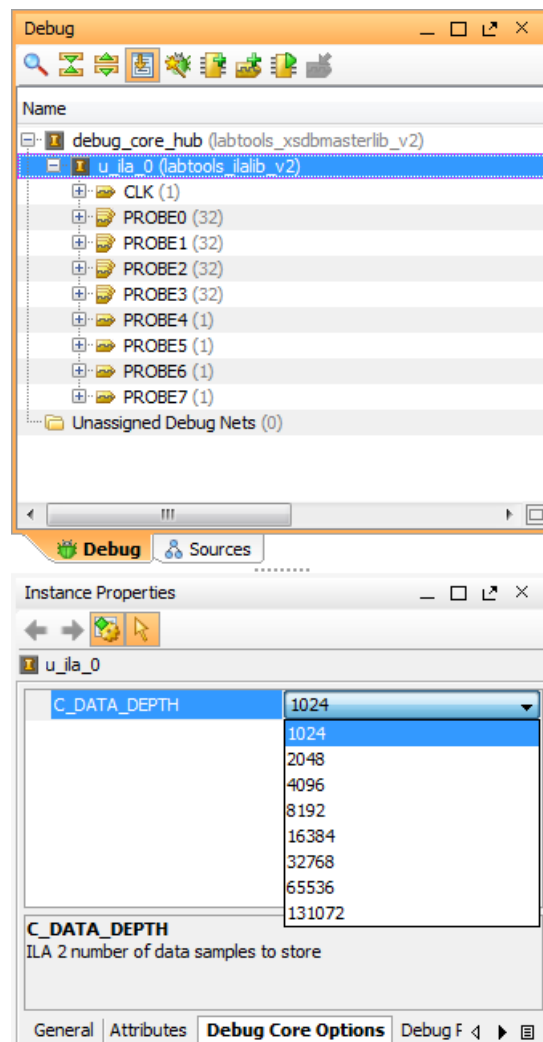


Figure 4-5: Changing the Data Depth of the ILA v2.0 Core

Using Tcl Commands Insert Debug Cores

In addition to using the Set up Debug wizard, you can also use Tcl commands to create, connect, and insert debug cores into your synthesized design netlist:

1. Create the ILA v2.0 core black box.

```
create_debug_core u_ila_0 labtools_ilalib_v2
```

2. Set the data depth property of the ILA v2.0 core.

```
set_property C_DATA_DEPTH 1024 [get_debug_cores u_ila_0]
```

3. Set the width of the CLK port of the ILA v2.0 core to 1 and connect it to the desired clock net.

```
set_property port_width 1 [get_debug_ports u_ila_0/CLK]
connect_debug_port u_ila_0/CLK [get_nets [list clk ]]
```

Note: You do not have to create the CLK port of the ILA v2.0 core because it is automatically created by the `create_debug_core` command.

4. Set the width of the PROBE0 port to the number of nets you plan to connect to the port.

```
set_property port_width 1 [get_debug_ports u_ila_0/PROBE0]
```

Note: You do not have to create the first probe port (PROBE0) of the ILA v2.0 core because it is automatically created by the `create_debug_core` command.

5. Connect the PROBE0 port to the nets you want to attach to that port.

```
connect_debug_port u_ila_0/PROBE0 [get_nets [list A_or_B]]
```

6. Optionally, create more probe ports, set their width, and connect them to the nets you want to debug.

```
create_debug_port u_ila_0 PROBE
set_property port_width 2 [get_debug_ports u_ila_0/PROBE1]
connect_debug_port u_ila_0/PROBE1 [get_nets [list {A[0]} {A[1]}]]to debug
```

7. Optionally, generate and synthesize the debug core(s) so you can floorplan them with the rest of your synthesized design.

```
implement_debug_cores [get_debug_cores]
```

For more information on these and other related Tcl commands, type `help -category ChipScope` in the Tcl Console of the Vivado IDE.

Implementing the Design

After inserting, connecting and customizing your debug cores, you are now ready for implementing your design (refer to the section called [Implementing the Design Containing the Debug Cores](#)).

HDL Instantiation Debug Probing Flow Overview

The HDL instantiation probing flow involves the manual customization, instantiation, and connection of various debug core components directly in the HDL design source. The debug cores that are supported in this flow in the Vivado tool are shown in table [Table 4-2](#).

Table 4-2: Debug Cores available for use in the HDL Instantiation Probing Flow

Debug Core	Version	Description	Run-time Analyzer Tool
ICON (Integrated Controller)	v1.06a	Debug core hub used to connect the ILA 1.05a and VIO 1.05a cores to the JTAG chain.	ChipScope Pro Analyzer
VIO (Virtual Input/Output)	v1.05a	Debug core that is used to monitor or control signals in a design at JTAG chain scan rates. Requires connection to an ICON core.	ChipScope Pro Analyzer
ILA (Integrated Logic Analyzer)	v1.05a	Debug core that is used to trigger on hardware events and capture data at system speeds. Requires connection to an ICON core.	ChipScope Pro Analyzer
ILA (Integrated Logic Analyzer)	v2.0	Debug core that is used to trigger on hardware events and capture data at system speeds.	Vivado logic analyzer

The ICON, VIO, and ILA v1.x cores are supported in the Vivado tool flow in order to provide compatibility with legacy designs that contain these cores. The new ILA v2.0 core has two distinct advantages over the legacy ILA v1.x core:

- Works with the integrated Vivado logic analyzer feature (refer to [Debugging the Design in Hardware, page 33](#)).
- No ICON core instance or connection is required.



IMPORTANT: *The ILA v2.0 HDL instantiation flow will result in a black box instance in your post-synthesis design netlist. This black box will be replaced with the fully populated ILA v2.0 core during the `opt_design` or `place_design` steps of the design implementation process.*

Using the HDL Instantiation Debug Probing Flow

The steps required to perform the HDL instantiation flow are:

1. Customize and generate the ILA v2.0 debug core(s) that have the right number of probe ports for the signals you want to probe.
2. Synthesize the design containing the debug core(s).
3. (Optional) Modify debug core properties.
4. Implement the design containing the debug core(s).

Customizing and Generating the Debug Cores

Use the **IP Catalog** button in the **Project Manager** to locate, select, and customize the desired debug core. The debug cores are located in the **Debug & Verification > Debug** category of the IP Catalog (see [Figure 4-6](#)). You can customize the debug core by double-clicking on the IP core or by right-click selecting the **Customize IP** menu selection. For more information on customizing the ILA v2.0 core, please refer to *LogiCORE IP ChipScope Pro Integrated Logic Analyzer (ILA) (v2) Datasheet (DS875)*. After customizing the core, click the **Generate** button in the IP customization wizard. This will generate the customized debug core and add it to the **Sources** view of your project.

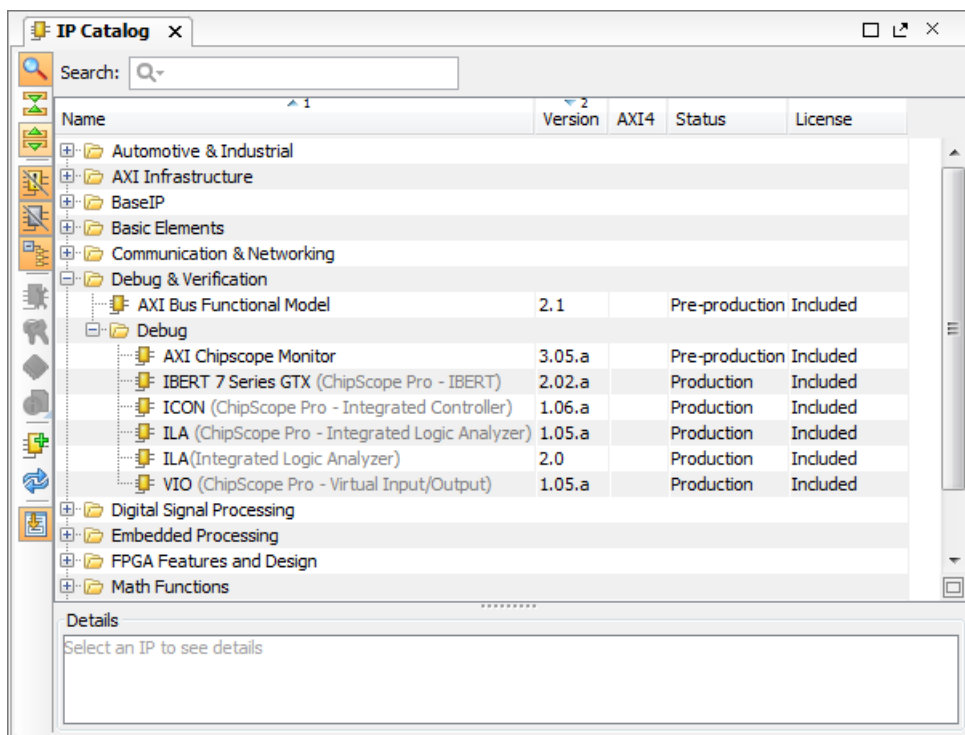


Figure 4-6: Debug Cores in the IP Catalog

Instantiating the Debug Cores

After generating the debug core, instantiate it in your HDL source code and connect it to the signals that you wish to probe for debugging purposes. Below is an example of the ILA v2.0 instance in a Verilog HDL source file:

```
ila_v2_0_0 i_ila
(
    .CLK(clk),
    .PROBE0(counterA),
    .PROBE1(counterB),
    .PROBE2(counterC),
    .PROBE3(counterD),
    .PROBE4(A_or_B),
    .PROBE5(B_or_C),
    .PROBE6(C_or_D),
    .PROBE7(D_or_A)
);
```

Note: The legacy VIO and ILA v1.x cores require a connection to an ICON v1.x core. The ILA v2.0 core instance does not require a connection to an ICON core instance. Instead, a debug_core_hub debug core will automatically be inserted into the synthesized design netlist to provide connectivity between the ILA v2.0 core and the JTAG scan chain.

Synthesizing the Design Containing the Debug Cores

In the next step, synthesize the design containing the debug core(s) by clicking **Run Synthesis** in the Vivado IDE or by running the following Tcl commands:

```
launch_runs synth_1
wait_on_run synth_1
```

You can also use the synth_design Tcl command to synthesize the design. Refer to *Vivado Design Suite User Guide: Synthesis (UG901)* [Ref 2] for more details on the various ways you can synthesize your design.



IMPORTANT: The ILA v2.0 core that you add to your design is a black box instance that will be replaced with the fully populated ILA v2.0 core during the opt_design or place_design steps of the design implementation process. You can safely ignore the following critical warning that appears during the synthesis process:

CRITICAL WARNING: [Designutils 20-1022] Could not resolve non-primitive black box cell 'ila' instantiated in module 'inst'.

Viewing the Debug Cores in the Synthesized Design

After synthesizing your design, you can open the synthesized design to view the debug cores and modify their properties. Open the synthesized design by clicking **Open Synthesized Design** in the **Flow Navigator** and select the **Debug** window layout to see the **Debug** window that shows your ILA v2.0 debug core(s) connected to the debug_core_hub (see Figure 4-7).

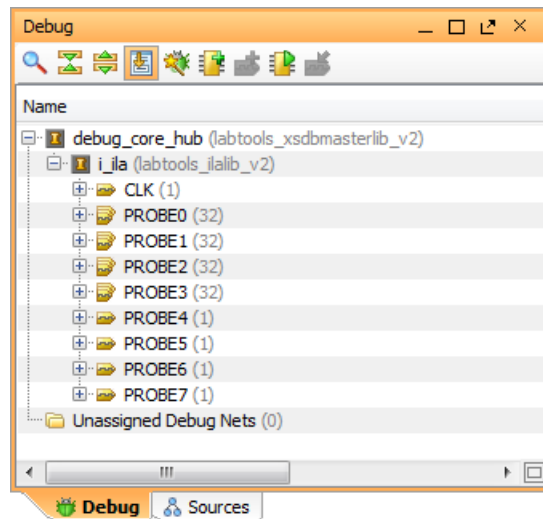


Figure 4-7: Debug Window Showing ILA v2.0 Core and Debug Core Hub

Changing the BSCAN User Scan Chain of the Debug Core Hub

You can view and change the BSCAN user scan chain index of the `debug_core_hub` by selecting the **debug_core_hub** in the **Debug** window, selecting the **Debug Core Options** view in the **Properties** window, then changing the value of the **C_USER_SCAN_CHAIN** property (see [Figure 4-8](#)).



IMPORTANT: *If you plan to mix legacy ICON, ILA, and/or VIO v1.x cores with ILA v2.0 cores, you will need to set the `C_USER_SCAN_CHAIN` property of the `debug_core_hub` to a user scan chain that does not conflict with the ICON v1.x core's Boundary Scan Chain setting. Failure to do so will result in errors later in the implementation flow.*

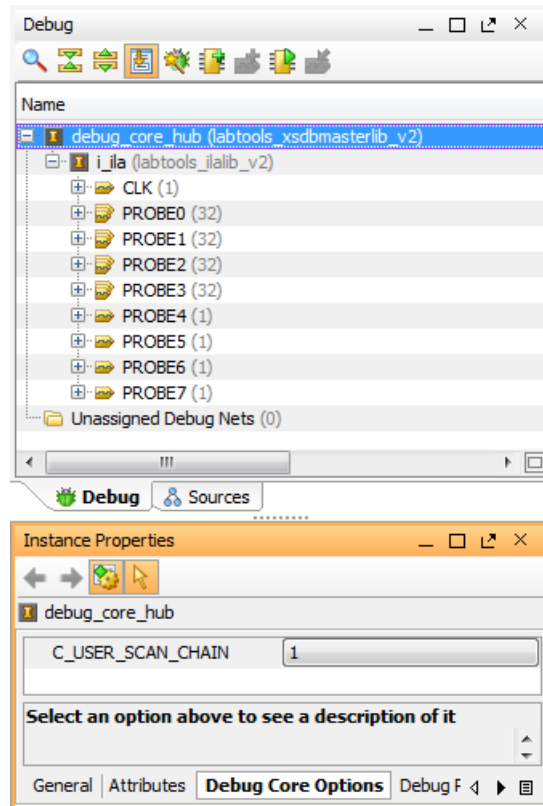


Figure 4-8: Changing the user scan chain property of the debug_core_hub

Implementing the Design Containing the Debug Cores

The Vivado software creates the debug_core_hub debug cores initially as black boxes. These cores must be implemented prior to running the placer and router.

Implementing the Debug Cores

Debug core implementation is done automatically when implementing the design; however, you can also force debug core implementation manually for floorplanning or timing analysis. To manually implement the cores: do one of the following:

- Click on the **Implement Debug Cores** icon on the toolbar menu of the **Debug** window.
- Right-click on any debug core in the **Debug** window and select the **Implement Debug Cores** option from the popup menu.

The Vivado IDE will generate and synthesize each black box debug core. This operation can take some time. A progress indicator shows that the operation is running. When the debug core implementation is complete, the debug core black boxes are resolved and you can access the generated instances.

Implementing the Design

Implement the design containing the debug core(s) by clicking **Run Implementation** in the Vivado IDE or by running the following Tcl commands:

```
launch_runs impl_1  
wait_on_run impl_1
```

You can also implement the design using the implementation commands `opt_design`, `place_design`, and `route_design`. Refer to the *Vivado Design Suite User Guide: Implementation (UG904)* [\[Ref 3\]](#) for more details on the various ways you can implement your design.

Debugging the Design in Hardware

Introduction

Once you have the debug core(s) in your design, you can use the run-time logic analyzer features to debug the design in hardware. Two different tools can be used depending on the type of debug cores in your design:

- ChipScope™ Pro Analyzer: used with ICON v1.x, ILA v1.x, VIO v1.x, and all IBERT debug cores.
- Vivado™ logic analyzer feature: used with ILA v2.0 debug cores.

If you have a mixture of ICON/ILA/VIO v1.x and ILA v2.0 debug cores in your design, you can simultaneously use both the ChipScope Pro Analyzer tool and Vivado logic analyzer feature to debug the same design running on the same hardware target board (see section called [Using Vivado Logic Analyzer to Debug the Design, page 34](#) for more details).

Launching ChipScope Pro Analyzer to Debug the Design

The ChipScope Pro Analyzer tool is used to interact with ICON v1.x, ILA v1.x, and VIO v1.x debug cores that are in your design. When the ChipScope Pro Analyzer software is installed, you can launch it directly from the Vivado IDE on any implemented design on which Generate Bitstream has been run.

To launch ChipScope Pro Analyzer, do one of the following:

- Use the Flow > Launch ChipScope Analyzer command from the main menu
- Run the `launch_chipscope_analyzer` Tcl command in the Tcl Console

The Vivado IDE passes the BIT bitstream and CDC net connection name files automatically to the ChipScope Pro Analyzer tool. For more information about ChipScope Pro Analyzer see the Xilinx website,

http://www.xilinx.com/support/documentation/dt_chipscopepro.htm.

Using Vivado Logic Analyzer to Debug the Design

The Vivado logic analyzer feature is used to interact with ILA v2.0 debug cores that are in your design. To access the Vivado logic analyzer feature, click the **Open Hardware Session** button in the **Program and Debug** section of the **Flow Navigator**.

The steps to debug your design in hardware are:

1. Connect to the hardware target and program the FPGA device with the `.bit` file
2. Set up the ILA debug core trigger and probe compare conditions.
3. Arm the ILA debug core trigger.
4. View the captured data from the ILA debug core in the **Waveform** window.

Connecting to the Hardware Target and Programming the FPGA Device

Programming an FPGA device prior to debugging are exactly the same steps described in [Using a Vivado Hardware Session to Program an FPGA Device in Chapter 2](#). After programming the device with the `.bit` file that contains the ILA v2.0 core, the **Hardware** window now shows the ILA core that was detected when scanning the device (see [Figure 5-1](#)).

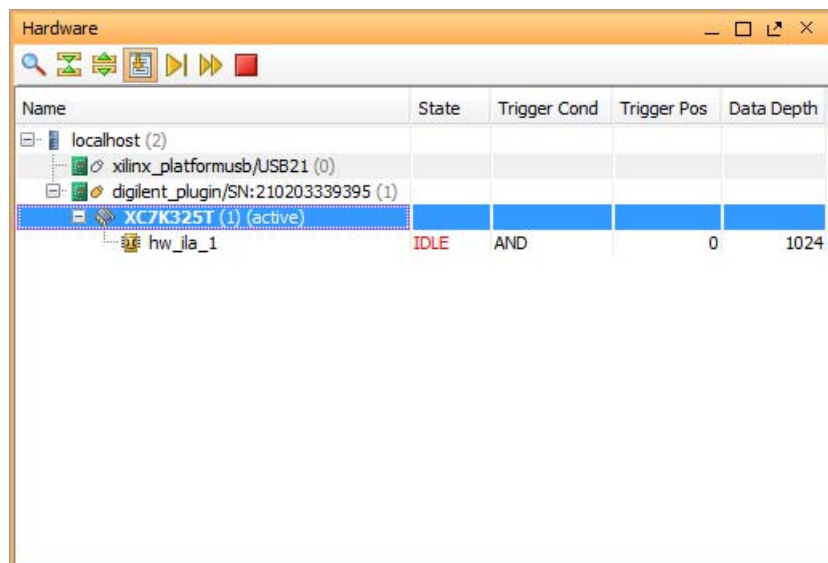


Figure 5-1: Hardware Window Showing the ILA Debug Core

Setting up the ILA Core to Take a Measurement

The ILA core(s) that you add to your design appear in the **Hardware** window under the target device. If you do not see the ILA core(s) appear, right-click on the device and select **Refresh Hardware**. This re-scans the FPGA device and refreshes the **Hardware** window.

Note: If you still do not see the ILA core after programming and/or refreshing the FPGA device, check to make sure the device was programmed with the appropriate `.bit` file and check to make sure the implemented design contains an ILA v2.0 core.

Click the ILA core (called `hw_ila_1` in [Figure 5-1](#)) to see its properties in the **ILA Core Properties** window. You can also change several ILA core settings in the **Hardware** window:

- Trigger condition
- Trigger position
- Data depth

Setting the ILA Core Trigger Condition

Use the **Trigger Cond** control in the **Hardware** window (or the **Trigger Condition** property in the **ILA Core Properties** window) to select between “AND” and “OR” settings. The “AND” setting causes a trigger event when all of the ILA probe comparisons are satisfied. The “OR” setting causes a trigger event when any of the ILA probe comparisons are satisfied. You can also use the `set_property Tcl` command to change the ILA core trigger condition:

```
set_property CONTROL.TRIGGER_CONDITION AND [get_hw_ilas hw_ila_1]
```

Setting the ILA Core Trigger Position

Use the **Trigger Pos** control in the **Hardware** window (or the **Trigger Position** property in the **ILA Core Properties** window) to set the position of the trigger mark in the captured data buffer. You can set the trigger position to any sample number in the captured data buffer. For instance, in the case of a captured data buffer that is 1024 samples deep:

- Sample number 0 corresponds to the first (left-most) sample in the captured data buffer.
- Sample number 1023 corresponds to the last (right-most) sample in the captured data buffer.
- Samples numbers 511 and 512 correspond to the two “center” samples in the captured data buffer.

You can also use the `set_property` Tcl command to change the ILA core trigger position:

```
set_property CONTROL.TRIGGER_POSITION 512 [get_hw_ilas hw_ila_1]
```

Setting the ILA Core Data Depth

Use the **Data Depth** control in the **Hardware** window (or the **Capture data depth** property in the **ILA Core Properties** window) to set the data depth of the ILA core captured data buffer. You can set the data depth to any power of two from 1 to the maximum data depth.

Note: Refer to the section called [Modifying Properties on the Debug Cores, page 25](#) for more details on how to set the maximum capture buffer data depth on ILA cores that are added to the design using the Netlist Insertion probing flow.

You can also use the `set_property` Tcl command to change the ILA core trigger position:

```
set_property CONTROL.DATA_DEPTH 512 [get_hw_ilas hw_ila_1]
```

Writing ILA Probes Information

The **ILA Probes** window contains information about the nets that you probed in your design using the ILA v2.0 core. This ILA probe information is extracted from your design and is stored in a data file that typically has an `.ltx` file extension.

Normally, the ILA probe file is automatically created during the implementation process. However, you can also use the `write_debug_probes` Tcl command to write out the debug probes information to a file:

1. Open the Synthesized or Netlist Design.
2. Run the `write_debug_probes filename.ltx` Tcl command.

Reading ILA Probes Information

The ILA probe file is automatically associated with the FPGA hardware device if the probes file is called `debug_nets.ltx` and is found in the same directory as the bitstream programming (`.bit`) file that is associated with the device.

You can also specify the location of the probes file:

1. Select the FPGA device in the **Hardware** window.
2. Set the **Probes file** location in the **Hardware Device Properties** window.
3. Click **Apply** to apply the change.

You can also set the location using the `set_property` Tcl command:

```
set_property PROBES.FILE {C:/myprobes.ltx} [lindex [get_hw_devices] 0]
```

Viewing ILA Probes

The **ILA Probes** window is used to view the probes associated with each ILA core. Once the probes file is associated with the FPGA hardware device, select the hardware device and use the right-click **Refresh Device** command to refresh the **ILA Probes** window (see [Figure 5-2](#)).

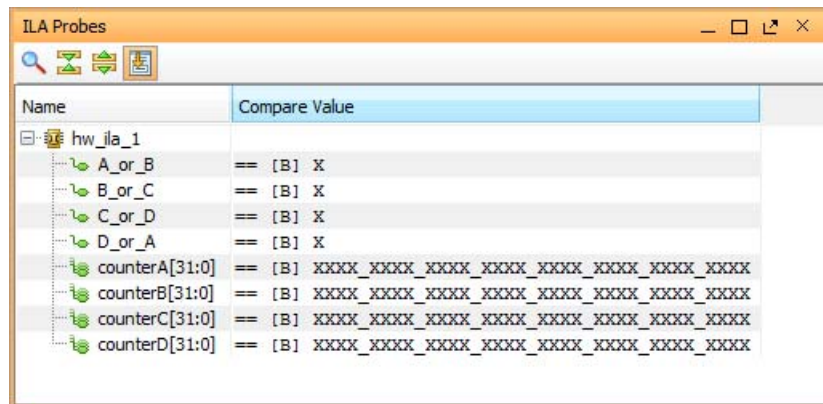


Figure 5-2: ILA Probes Window

Setting Up the ILA Probe Trigger Compare Values

The ILA probe trigger comparators are used to detect specific equality or inequality conditions on the probe inputs to the ILA v2.0 core. The trigger condition is the result of a Boolean "AND" or "OR" calculation of each of the ILA probe trigger comparator results (see section called [Setting the ILA Core Trigger Condition, page 35](#) for more details). To specify the compare values for a given ILA probe, select the **Compare Value** cell in the **ILA Probes** window to open the **Compare Value** dialog box (see [Figure 5-3](#)).

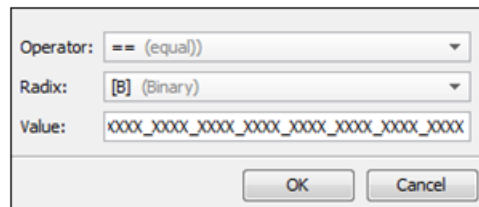


Figure 5-3: ILA Probe Compare Value Dialog Box

ILA Probe Compare Value Settings

The **Compare Value** dialog box contains three fields that you can configure:

1. **Operator:** This is the comparison operator that you can set to the following values:
 - == (equal)
 - != (not equal)
 - < (less than)
 - <= (less than or equal)
 - > (greater than)
 - >= (greater than or equal)
2. **Radix:** This is the radix or base of the Value that you can set to the following values:
 - [B] Binary
 - [H] Hexadecimal
 - [O] Octal
 - [A] ASCII
 - [U] Unsigned Decimal
 - [S] Signed Decimal
3. **Value:** This is the comparison value that will be compared (using the **Operator**) with the real-time value on the net(s) in the design that are connected to the probe input of the ILA v2.0 debug core. Depending on the Radix settings, the Value string is as follows:
 - Binary
 - 0: logical zero
 - 1: logical one
 - X: don't care
 - R: rising or low-to-high transition
 - F: falling or high-to-low transition
 - B: either low-to-high or high-to-low transitions
 - N: no transition (current sample value is the same as the previous value)
 - Hexadecimal
 - X: All bits corresponding to the value string character are "don't care" values
 - 0-9: Values 0 through 9

- A-F: Values 10 through 15
 - Octal
 - X: All bits corresponding to the value string character are "don't care" values
 - 0-7: Values 0 through 7
 - ASCII
 - Any string made up of ASCII characters
 - Unsigned Decimal
 - Any non-negative integer value
 - Signed Decimal
 - Any integer value
-

Running or Arming the ILA Core Trigger

You can run or arm the ILA core trigger in two different modes:

- **Run Trigger:** Selecting the ILA core to be armed followed by clicking the **Run Trigger** button on the **Hardware** window toolbar arms the ILA core to detect the trigger event that is defined by the ILA core trigger condition and probe compare values.
- **Run Trigger Immediate:** Selecting the ILA core to be armed followed by clicking the **Run Trigger Immediate** button on the **Hardware** window toolbar arms the ILA core to trigger immediately regardless of the settings of the ILA core trigger condition and probe compare values. This command is useful for capturing any values that present at the probe inputs of the ILA core.

You can also arm the trigger by selecting and right-clicking on the ILA core and selecting **Run Trigger** or **Run Trigger Immediate** from the popup menu (see [Figure 5-4](#)).

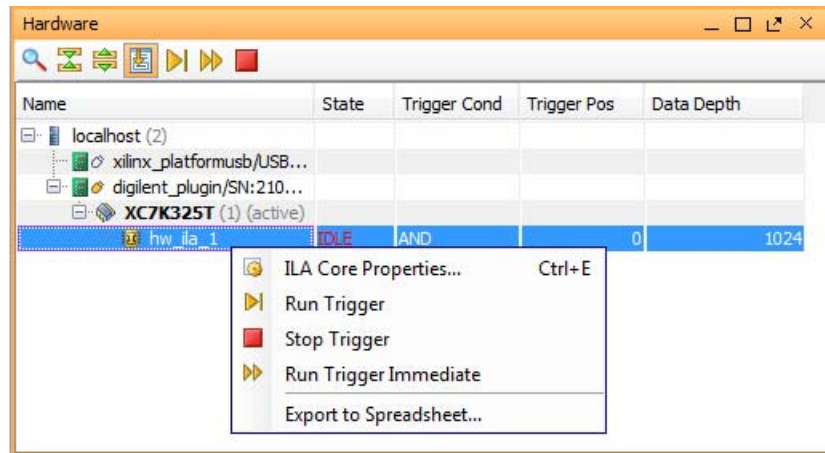


Figure 5-4: ILA Core Trigger Commands

Stopping the ILA Core Trigger

You can stop the ILA core trigger by selecting the appropriate ILA core followed by clicking on the **Stop Trigger** button on the **Hardware** window toolbar. You can also stop the trigger by selecting and right-clicking on the appropriate ILA core and selecting **Stop Trigger** from the popup menu (see [Figure 5-4](#)).

Viewing the ILA Core State

The **State** column in the **Hardware** window indicates the current state or status of each ILA core (see [Table 5-1](#)).

Table 5-1: ILA Core State Description

ILA Core State	Description
IDLE	The ILA core is idle and waiting for its trigger to be armed.
ARMED	The ILA core trigger is armed and is waiting for the trigger condition to be satisfied. This is the state when the trigger has been armed with the trigger position set to 0.
CAPTURING	The ILA core is capturing data into its data capture buffer. If the trigger position is set to a value greater than 0, this state also means that the trigger is armed.
FULL	The ILA core capture buffer is full and is being uploaded to the host for display.

Viewing Captured Data from the ILA Core in the Waveform Viewer

Once the ILA core captured data has been uploaded to the Vivado IDE, it is displayed in the **Waveform Viewer**. See [Chapter 6, Viewing ILA Probe Data Using Waveform Viewer](#) for details on using the Waveform Viewer to view captured data from the ILA core.

Saving and Restoring Captured Data from the ILA Core

In addition to displaying the captured data that is directly uploaded from the ILA core, you can also write the captured data to a file then read the data from a file and display it in the waveform viewer.

Saving Captured ILA Data to a File

Currently, the only way to upload captured data from an ILA core and save it to a file is to use the following Tcl command:

```
write_hw_ila_data my_hw_ila_data_file [upload_hw_ila_data hw_ila_1]
```

Restoring Captured ILA Data from a File

Currently, the only way to restore captured data from a file and display it in the waveform viewer is to use the following Tcl command:

```
display_hw_ila_data [read_hw_ila_data my_hw_ila_data_file]
```

Using Vivado Logic Analyzer in a Lab Environment

The Vivado logic analyzer feature is integrated into the Vivado IDE. In order to use Vivado logic analyzer feature to debug a design that is running on a target board that is in a lab environment, you need to do one of two things:

- Install and run the full Vivado IDE on your lab machine.
- Install the ISE Lab Tools 14.2 on your lab machine, and use the Vivado logic analyzer feature on your local machine to connect to a remote instance of the CSE Server.

Installing and Running the Full Vivado IDE on a Lab Machine

The requirements for installing the Vivado IDE on your lab machine are found in *Vivado Design Suite Installation and Licensing Guide (UG798)* [\[Ref 4\]](#).

The Vivado logic analyzer feature graphical user interface is only available in Project mode, but it does not require the original project used to create the design. The Vivado logic analyzer only needs two files from the original project: the bitstream programming (.bit) file and the probes (.ltx) file. Here are the steps to use the Vivado logic analyzer feature on a lab machine.

1. Install the Vivado IDE on your lab machine.
2. Copy the bitstream programming (.bit) file and the probes (.ltx) file to the lab machine.
3. Start Vivado IDE in GUI mode.
4. Create a sample project without any source, IP, or constraint files.
5. Click on the **Open Hardware Session** button in the **Flow Navigator**.
6. Follow the steps in the [Connecting to the Hardware Target and Programming the FPGA Devices](#) section to open a connection to the target board that is connected to your lab machine. Use the bitstream programming (.bit) file that you copied to the lab machine to program target FPGA device.
7. Follow the steps in the [Setting up the ILA Core to Take a Measurement](#) section and beyond to debug your design in hardware. Use the probes (.ltx) file that you copied to the lab machine when you get to the [Reading ILA Probes Information](#) section.

Connecting to a Remote CSE Server Running on a Lab Machine

If you have a network connection to your lab machine, you can also connect to the target board by connecting to a CSE server that is running on that remote lab machine. Here are the steps to using the Vivado logic analyzer feature to connect to a CSE server that is running on the lab machine:

1. Install the ISE Lab Tools 14.2 on the lab machine.
2. Start up the cse_server application on the remote lab machine. Assuming you installed the ISE Lab Tools 14.2 to the default location and your lab machine is a 64-bit Windows machine, here is the command line:

```
C:\Xilinx\14.2\LabTools\LabTools\bin\nt64\cse_server -port 50001
```

3. Start Vivado IDE in GUI mode on a different machine than your lab machine.
4. Follow the steps in the [Connecting to the Hardware Target and Programming the FPGA Device](#) section to open a connection to the target board that is connected to your lab machine. However, instead of connecting to a CSE server running on localhost, use the host name of your lab machine.

5. Follow the steps in the [Setting up the ILA Core to Take a Measurement](#) section and beyond to debug your design in hardware.

Using Vivado Logic Analyzer and ChipScope™ Pro Analyzer Simultaneously

You can use the Vivado logic analyzer feature and ChipScope™ Pro Analyzer tools to simultaneously debug the same design running on the same target board. Some examples of situations where this capability becomes important are:

- You have a design that contains an ILA v2.0 debug core and a VIO v1.x debug core that you need to interact with using the Vivado logic analyzer feature and ChipScope Pro Analyzer feature, respectively.
- You want to interact with an ILA v2.0 debug core in your design using the Vivado logic analyzer feature and you want to monitor the XADC temperature or voltage sensors using the ChipScope Pro Analyzer tool's System Monitor feature.
- You have a 7 series device and a 6 series device that you need to debug at the same time using the Vivado logic analyzer feature and the ChipScope Pro Analyzer feature, respectively.

This section covers the first case of having both an ILA v2.0 debug core and a VIO v1.x debug core in the same design. The steps that you need to follow to take advantage of this capability are:

1. Use two separate BSCAN user scan chains, one for each JTAG controller core.
2. Start two separate CSE servers, one for each run-time analyzer applications.
3. Use two different run-time analyzer applications, the Vivado logic analyzer feature and ChipScope Pro Analyzer tool.

Using Separate BSCAN User Scan Chains

Make sure that the debug_core_hub core (which is used to connect the ILA v2.0 core to a BSCANE2 primitive) and the ICON v1.x core (which is used to connect the VIO v1.x core to a BSCANE2 primitive) are configured to use different user scan chains (see section [Changing the BSCAN User Scan Chain of the Debug Core Hub in Chapter 4](#) for more details).

Setting Up Separate CSE Servers

- Make sure two separate CSE server instances are running on the machine that is connected to the target board. For instance, on a Windows platform, do the following in two separate `cmd` windows:

- In the first cmd window, run `cse_server -port 50001`.
- In the second cmd window, run `cse_server -port 50002`.

The CSE server running on ports 50001 and 50002 will be used by the Vivado logic analyzer feature and ChipScope Pro Analyzer tool, respectively.

Running Vivado Logic Analyzer Feature and ChipScope Pro Analyzer Tool

Now that you have set up the design and the CSE servers, you can use the Vivado logic analyzer feature and ChipScope Pro Analyze tool to debug the design by following these steps:

1. Start the Vivado IDE in GUI mode.
2. Click **Open Hardware Session** in the **Flow Navigator**.
3. Connect to the CSE server that is running on localhost:50001, and program the target device (see [Using a Vivado Hardware Session to Program an FPGA Device in Chapter 2](#) for more details)
4. Launch the ChipScope Pro Analyzer using the **Flow>Launch ChipScope Analyzer** menu command.
5. In the ChipScope Pro Analyzer tool, select **JTAG Chain>Server Host Setting**. Change the server to **localhost:50002** (see [Figure 5-5](#))

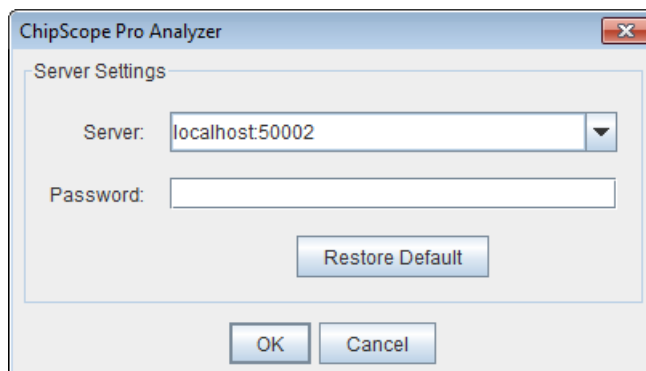


Figure 5-5: Changing the ChipScope Pro Analyzer Server Host Settings

6. In the ChipScope Pro Analyzer, connect to the target board using the appropriate cable in the JTAG Chain menu.

- Use each of the run-time analyzer tools to interact with their respective debug cores (see Figure 5-6)

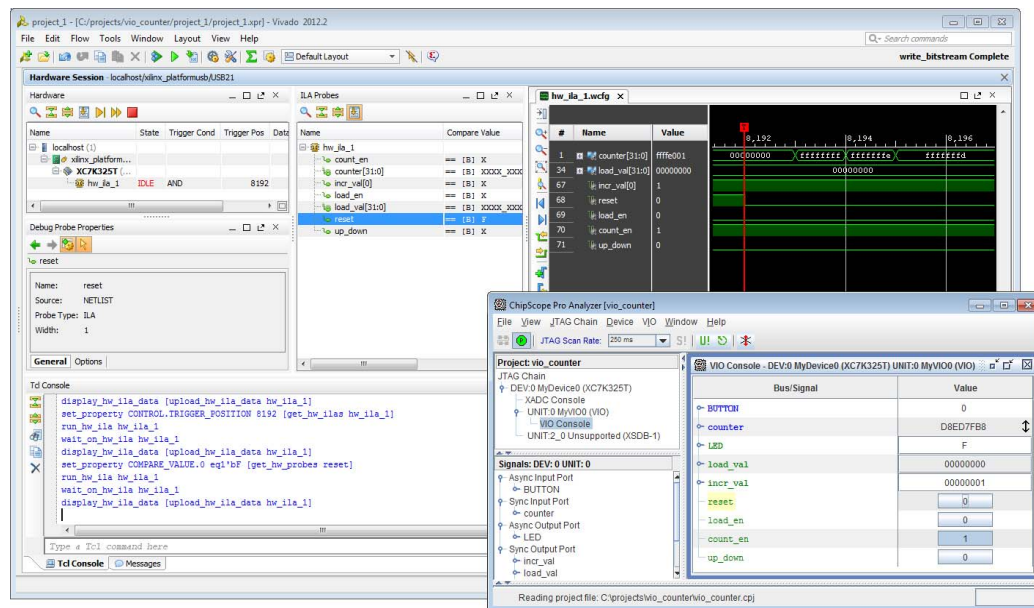


Figure 5-6: Using Vivado Logic Analyzer Feature and ChipScope Pro Analyzer Tool to Debug the Design

Description of Hardware Session Tcl Objects and Commands

You can use Tcl commands to interact with your hardware under test. The hardware is organized in a set of hierarchical first class Tcl objects (see Table 5-2).

Table 5-2: Hardware Session Tcl Objects

Tcl Object	Description
hw_server	Object referring to CSE server. Each hw_server can have one or more hw_target objects associated with it.
hw_target	Object referring to JTAG cable or board. Each hw_target can have one or more hw_device objects associated with it.
hw_device	Object referring to a device in the JTAG chain, including Xilinx FPGA devices. Each hw_device can have one or more hw_ila objects associated with it.
hw_ila	Object referring to an ILA core in the Xilinx FPGA device. Each hw_ila object can have only one hw_ila_data object associated with it. Each hw_ila object can have one or more hw_probe objects associated with it.

Table 5-2: Hardware Session Tcl Objects (Cont'd)

Tcl Object	Description
hw_ila_data	Object referring to data uploaded from an ILA debug core.
hw_probe	Object referring to the probe input of an ILA debug core.

For more information about the hardware session commands, run the `help -category hardware` Tcl command in the Tcl Console.

Description of hw_server Tcl Commands

Table 5-3 contains descriptions of all Tcl commands used to interact with hardware servers.

Table 5-3: Descriptions of hw_server Tcl Commands

Tcl Command	Description
connect_hw_server	Open a connection to a hardware server.
current_hw_server	Get or set the current hardware server.
disconnect_hw_server	Close a connection to a hardware server.
get_hw_servers	Get list of hardware server names for the CSE servers.
refresh_hw_server	Refresh a connection to a hardware server.

Description of hw_target Tcl Commands

Table 5-4 contains descriptions of all Tcl commands used to interact with hardware targets.

Table 5-4: Descriptions of hw_target Tcl Commands

Tcl Command	Description
close_hw_target	Close a hardware target.
current_hw_target	Get or set the current hardware target.
get_hw_targets	Get list of hardware targets for the hardware servers.
open_hw_target	Open a connection to a hardware target on the hardware server.
refresh_hw_target	Refresh a connection to a hardware target.

Description of hw_device Tcl Commands

Table 5-5 Descriptions of hw_device Tcl Commands contains descriptions of all Tcl commands used to interact with hardware devices.

Table 5-5: Descriptions of hw_device Tcl Commands

Tcl Command	Description
current_hw_device	Get or set the current hardware device.
get_hw_device	Get list of hardware devices for the target.
program_hw_device	Program Xilinx FPGA devices.
refresh_hw_device	Refresh a hardware device.

Description of hw_ila Tcl Commands

Table 5-6 Descriptions of hw_ila Tcl Commands contains descriptions of all Tcl commands used to interact with ILA v2.0 debug cores.

Table 5-6: Descriptions of hw_ila Tcl Commands

Tcl Command	Description
current_hw_ila	Get or set the current hardware ILA.
get_hw_ilas	Get list of hardware ILAs for the target.
reset_hw_ila	Reset hw_ila control properties to default values.
run_hw_ila	Arm hw_ila triggers.
wait_on_hw_ila	Wait until all data has been captured.

Description of hw_ila_data Tcl Commands

Table 5-7 Descriptions of hw_ila_data Tcl Commands contains descriptions of all Tcl commands used to interact with captured ILA data.

Table 5-7: Descriptions of hw_ila_data Tcl Commands

Tcl Command	Description
current_hw_ila_data	Get or set the current hardware ILA data
display_hw_ila_data	Display hw_ila_data in waveform viewer
get_hw_ila_data	Get list of hw_ila_data objects
read_hw_ila_data	Read hw_ila_data from a file
upload_hw_ila_data	Stop the ILA core from capturing data and upload any captured data.
write_hw_ila_data	Write hw_ila_data to a file.

Description of hw_probe Tcl Commands

Table 5-8 contains descriptions of all Tcl commands used to interact with captured ILA data.

Table 5-8: Descriptions of hw_probe Tcl Commands

Tcl Command	Description
get_hw_probes	Get list of hardware probes.

Using Hardware Session Tcl Commands

Below is an example Tcl command script that interacts with the following example system:

- One KC705 board's Digilent JTAG-SMT1 cable (serial number 12345) accessible via a CSE server running on localhost:50001.
- Single ILA core in a design running in the XC7K325T device on the KC705 board.
- ILA core has a probe called counter[3:0].

Example Tcl Command Script

```
# Connect to the Digilent Cable on localhost:50001
connect_hw_server -host localhost -port 50001
current_hw_target [get_hw_targets */digilent_plugin/SN:12345]
open_hw_target

# Program and Refresh the XC7K325T Device
current_hw_device [lindex [get_hw_devices] 0]
refresh_hw_device -update_hw_probes false [lindex [get_hw_devices] 0]
set_property PROGRAM.FILE {C:/design.bit} [lindex [get_hw_devices] 0]
set_property PROBES.FILE {C:/design.ltx} [lindex [get_hw_devices] 0]
program_hw_devices [lindex [get_hw_devices] 0]
refresh_hw_device [lindex [get_hw_devices] 0]

# Set Up ILA Core Trigger Position and Probe Compare Values
set_property CONTROL.TRIGGER_POSITION 512 [get_hw_ilas hw_ila_1]
set_property COMPARE_VALUE.0 eq4'b0000 [get_hw_probes counter]

# Arm the ILA trigger and wait for it to finish capturing data
run_hw_ila hw_ila_1
wait_on_hw_ila hw_ila_1

# Upload the captured ILA data, display it, and write it to a file
current_hw_ila_data [upload_hw_ila_data hw_ila_1]
display_hw_ila_data [current_hw_ila_data]
write_hw_ila_data my_hw_ila_data [current_hw_ila_data]
```


Viewing ILA Probe Data Using Waveform Viewer

Introduction

With the Vivado™ Integrated Design Environment (IDE) simulator open, you can begin working with the waveform to analyze your design and debug your code. The Vivado logic analyzer populates design data in other areas, such as the hardware and the ILA probes windows.

Waveform Viewer Features and Limitations

Waveform Feature Summary

- Analog waveform support for ILA probe vectors with a minimum width of 2.
- Real radix can apply to digital or analog waveforms.

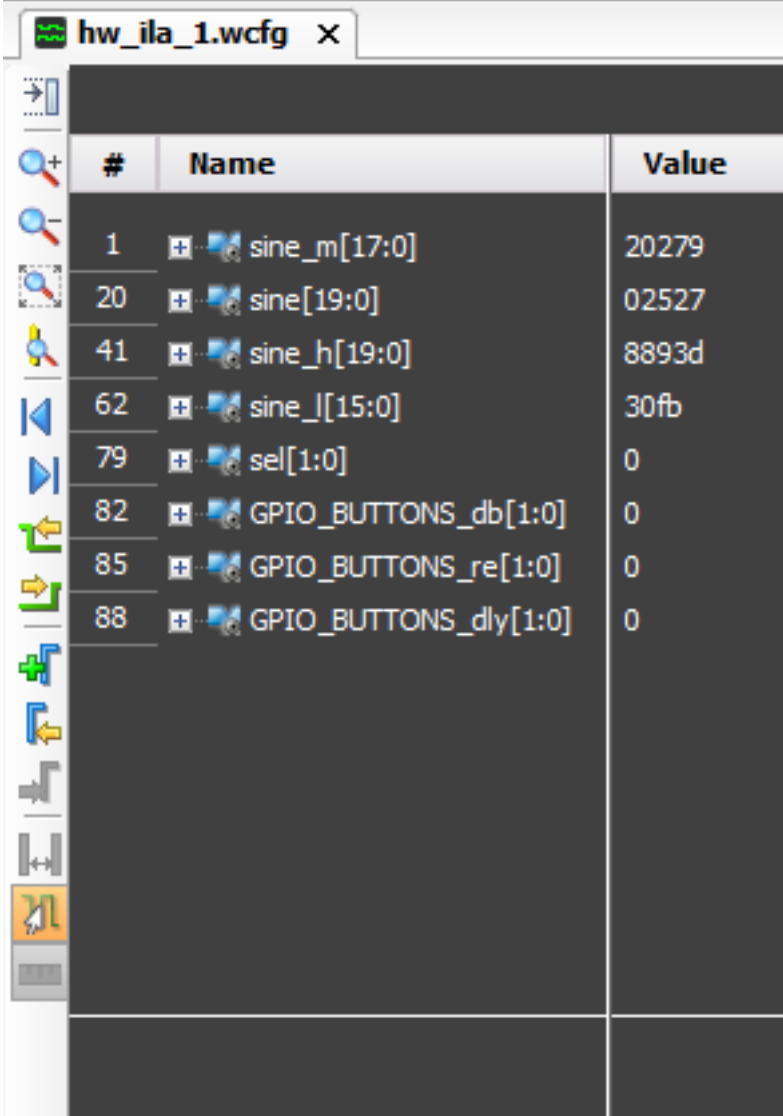
Waveform Limitations

- A maximum bus width of 64 bits on real numbers. Incorrect values are possible for buses wider than 64 bits.
- Floating point supports only 32- and 64-bit arrays.
- Switching between analog and digital waveform styles resets the row height to defaults (100 pixels for analog, 20 for digital).
- Wide arrays close to 64 bits with a radix other than real could contain rounding problems in Least Significant Bits (LSB).

Waveform Viewer Configuration Signals and Buses

The scalar and vector ILA probes in the **Waveform** window are the design objects in the waveform.

The ILA probes display with a corresponding identifying button. You can hover the mouse over the button for a description. [Figure 6-1](#) is an example of ILA probes in the **Waveform Configuration** window.

The screenshot shows a window titled 'hw_ila_1.wcfg'. On the left is a vertical toolbar with various icons for navigation and configuration. The main area is a table with three columns: '#', 'Name', and 'Value'. The table lists several ILA probes with their IDs, names, and current values.

#	Name	Value
1	sine_m[17:0]	20279
20	sine[19:0]	02527
41	sine_h[19:0]	8893d
62	sine_l[15:0]	30fb
79	sel[1:0]	0
82	GPIO_BUTTONS_db[1:0]	0
85	GPIO_BUTTONS_re[1:0]	0
88	GPIO_BUTTONS_dly[1:0]	0

Figure 6-1: Waveform ILA Probes

The ILA probes display with an **ID Number**, **Name**, and **Value**. The toolbar buttons on the left give you access to navigation features that are described in the following sections.


Using the Zoom Features

You have zoom functions as sidebar buttons to zoom in and out of a wave configuration as needed.

You can also use the mouse wheel with the **CTRL** key in combination after clicking within the waveform to zoom in and out to emulate the operation of the dials on an oscilloscope.



Waveform Options Dialog Box

When you select the **Waveforms Options** button  the **Waveform Options** dialog box, shown in [Figure 6-2](#), opens.

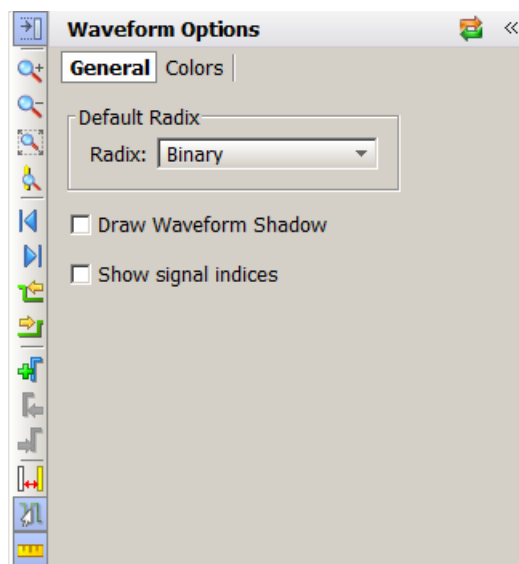


Figure 6-2: Waveform Options Dialog Box

The options are as follows:

- **General:** Set the default radix.
- **Show signal indices:** Checkbox inserts a small definition line between signal numbers.
- **Colors:** Lets you set colors for the objects within the waveform.

ILA Probes in Waveform Configuration

You can add ILA probe scalars and vectors, also called signals and buses, to the waveform configuration file, then save that configuration to a WDB file. You can populate the **Wave** window with the probes from your ILA core using menu commands, or using Tcl commands in the **Tcl** Console.

To add ILA probes to the waveform configuration:

1. In the **ILA Probes** window, expand the desired ILA core, and select a probe.
2. Right-click, and select **Add to Wave** window from the popup menu.

You can add copies of the same signal or bus in a wave configuration for comparing waveforms. You can place copies of the same signal or bus anywhere in the wave configuration, such as in groups or virtual buses.

To add a copy of a signal or bus, do the following:

1. Select a signal or bus in the wave configuration in the **Wave** window.
2. Select **Edit > Copy** or type **Ctrl+C**.

The signal name is copied to the clipboard.

3. Select **Paste** or type **Ctrl+V**.

The signal or bus is now copied to the wave configuration. You can move the signal or bus using drag and drop as needed.

Customizing the Wave Configuration

You can customize the Wave configuration using the features that are listed and briefly described in [Table 6-1](#); the feature name links to the subsection that fully describes the feature.

Table 6-1: Customization Features in the Wave Configuration

Feature	Description
Cursors	The main cursor and secondary cursor in the Wave window let you display and measure time, and they form the focal point for various navigation activities.
Markers	You can add markers to navigate through the waveform, and to display the waveform value at a particular time.
Dividers	You can add a divider to create a visual separator of signals.
Using Groups	You can add a group, that is a collection to which signals and buses can be added in the wave configuration as a means of organizing a set of related signals.
Using Virtual Buses	You can add a virtual bus to your wave configuration, to which you can add logic scalars and arrays.
Renaming Objects	You can rename objects, signals, buses, and groups.
Displaying Names	You can display the full hierarchical name (long name), the simple signal or bus name (short name), or a custom name for each signal.

Table 6-1: Customization Features in the Wave Configuration (Cont'd)

Feature	Description
Radixes	The default radix controls the bus radix that displays in the wave configuration, Objects panel, and the Console panel.
Bus Bit Order	You can change the Bus bit order from Most Significant Bit (MSB) to Least Significant Bit (LSB) and vice versa.

Cursors

Cursors are used primarily for temporary indicators of sample position and are expected to be moved frequently, as in the case when you are measuring the distance (in samples) between two waveform edges.



TIP: For more permanent indicators, used in situations such as establishing a time-base for multiple measurements, add markers to the Wave window instead. See [Markers, page 53](#) for more information.

You can place the main cursor with a single click in the **Wave** window.

To place a secondary cursor, **Ctrl+Click** and hold the waveform, and drag either left or right. You can see a flag that labels the location at the top of the cursor.

Alternatively, you can hold the **SHIFT** key and click a point in the waveform. The main cursor remains the original position, and the other cursor is at the point in the waveform that you clicked.

Note: To preserve the location of the secondary cursor while positioning the main cursor, hold the **Shift** key while clicking. When placing the secondary cursor by dragging, you must drag a minimum distance before the secondary cursor appears.

To move a cursor, hover over the cursor until you see the grab symbol, and click and drag the cursor to the new location.


As you drag the cursor in the Wave window, you see a hollow or filled-in circle if the **Snap to Transition** button is selected, which is the default behavior.

- A hollow circle ○ indicates that you are between transitions in the waveform of the selected signal.
- A filled-in circle ● indicates that you are hovering over the waveform transition of the selected signal. A secondary cursor can be hidden by clicking anywhere in the **Wave** window where there is no cursor, marker, or floating ruler.




Markers

Use a marker when you want to mark a significant event within your waveform in a permanent fashion. Markers allow you to measure distance (in samples) relevant to that marked event.

You can add, move, and delete markers as follows:

- You add markers to the wave configuration at the location of the main cursor.
 - a. Place the main cursor at the sample number where you want to add the marker by clicking in the **Wave** window at the sample number or on the transition.
 - b. Select **Edit > Markers > Add Marker**, or click the **Add Marker** button. 

A marker is placed at the cursor, or slightly offset if a marker already exists at the location of the cursor. The sample number of the marker displays at the top of the line.

- You can move the marker to another location in the waveform using the drag and drop method. Click the marker label (at the top of the marker) and drag it to the location.
 - The drag symbol  indicates that the marker can be moved. As you drag the marker in the **Wave** window, you see a hollow or filled-in circle if the **Snap to Transition** button is selected, which is the default behavior.
 - A filled-in circle  indicates that you are hovering over a transition of the waveform for the selected signal or over another marker.
 - For markers, the filled-in circle is white.
 - A hollow circle  indicates that you are between transitions in the waveform of the selected signal.
 - Release the mouse key to drop the marker to the new location.
- You can delete one or all markers with one command. Right-click over a marker, and do one of the following:
 - Select **Delete Marker** from the popup menu to delete a single marker.
 - Select **Delete All Markers** from the popup menu to delete all markers.

Note: You can also use the **Delete** key to delete a selected marker.

 - Use **Edit > Undo** to reverse a marker deletion.

Trigger Marker

The red trigger marker (whose label is a red letter 'T') is a special marker that indicates the occurrence of the trigger event in the capture buffer. The position of the trigger marker in the buffer directly corresponds to the Trigger Position setting (see section [Setting the ILA Core Trigger Position in Chapter 5](#)).

Note: The trigger marker is not movable using the same technique as regular markers. Its position is set using the ILA core's Trigger Position property setting.

Dividers

Dividers create a visual separator between signals. You can add a divider to your wave configuration to create a visual separator of signals, as follows:

1. In a **Name** column of the **Wave** window, click a signal to add a divider below that signal.
2. From the popup menu, select **Edit > New Divider**, or right-click and select **New Divider**.

The change is visual and nothing is added to the HDL code. The new divider is saved with the wave configuration file when you save the file.

You can move or delete Dividers as follows:

- Move a Divider to another location in the waveform by dragging and dropping the divider name.
- To delete a Divider, highlight the divider, and click the **Delete** key, or right-click and select **Delete** from the popup menu.

Dividers can be renamed also; see [Renaming Objects, page 56](#).

Using Groups

A Group is a collection of expandable and collapsible categories, to which you can add signals and buses in the wave configuration to organize related sets of signals. The group itself displays no waveform data but can be expanded to show its contents or collapsed to hide them. You can add, change, and remove groups.

To add a Group:

1. In a wave configuration, select one or more signals or buses to add to a group.
Note: A group can include dividers, virtual buses, and other groups.
2. Select **Edit > New Group**, or right-click and select **New Group** from the popup menu.

A Group that contains the selected signal or bus is added to the wave configuration.

A Group is represented with the **Group** button. 

The change is visual and nothing is added to the ILA core.

You can move other signals or buses to the group by dragging and dropping the signal or bus name.

You can move or remove Groups as follows:

- Move Groups to another location in the **Name** column by dragging and dropping the group name.

- Remove a group, by highlighting it and selecting **Edit > Wave Objects > Ungroup**, or right-click and select **Ungroup** from the popup menu. Signals or buses formerly in the group are placed at the top-level hierarchy in the wave configuration.

Groups can be renamed also; see [Renaming Objects, page 56](#).



CAUTION! The **Delete** key removes the group and its nested signals and buses from the wave configuration.

Using Virtual Buses

You can add a virtual bus to your wave configuration, which is a grouping to which you can add logic scalars and arrays. The virtual bus displays a bus waveform, which shows the signal waveforms in the vertical order that they appear under the virtual bus, flattened to a one-dimensional array. You can then change or remove virtual buses after adding them.

To add a virtual bus:

- In a wave configuration, select one or more signals or buses you want to add to a virtual bus.
- Select **Edit > New Virtual Bus**, or right-click and select **New Virtual Bus** from the popup menu.

The virtual bus is represented with the **Virtual Bus** button .

The change is visual and nothing is added to the HDL code.

You can move other signals or buses to the virtual bus by dragging and dropping the signal or bus name. The new virtual bus and its nested signals or buses are saved when you save the wave configuration file. You can also move it to another location in the waveform by dragging and dropping the virtual bus name.

You can rename a virtual bus; see [Renaming Objects](#).

To remove a virtual bus, and ungroup its contents, highlight the virtual bus, and select **Edit > Wave Objects > Ungroup**, or right-click and select **Ungroup** from the popup menu.



CAUTION! The **Delete** key removes the virtual bus and its nested signals and buses from the wave configuration.

Renaming Objects

You can rename any object in the **Wave** window, such as signals, dividers, groups, and virtual buses.

1. Select the object name in the **Name** column.
2. Select **Rename** from the popup menu.
3. Replace the name with a new one.
4. Press **Enter** or click outside the name to make the name change take effect.

You can also double-click the object name and then type a new name. The change is effective immediately. Object name changes in the wave configuration do not affect the names of the nets attached to the ILA core probe inputs.

Displaying Names

You can display the full hierarchical name (long name), the simple signal or bus name (short name), or a custom name for each signal. The signal or bus name displays in the **Name** column of the wave configuration. If the name is hidden:

- Expand the **Name** column until you see the entire signal name.
- Use the scroll bar in the **Name** column to view the name.

To change the display name:

1. Select one or more signal or bus names. Use **Shift+** click or **Ctrl+** click to select many signal names.
2. Right-click, and select **Name >**:
 - **Long** to display the full hierarchical name.
 - **Short** to display the name of the signal or bus only.
 - **Custom** to display the custom name given to the signal when renamed.

The name changes immediately according to your selection.

Radixes

Understanding the type of data on your bus is important. You need to recognize the relationship between the radix setting and the data type to use the waveform options of Digital and Analog effectively. See [About Radixes and Analog Waveforms, page 59](#) for more information about the radix setting and its effect on Analog waveform analysis.

You can change the radix of an individual signal (ILA probe) in the **Wave** window as follows:

1. Right-click a bus in the **Wave** window.
2. Select **Radix** and the format you want from the drop-down menu:
 - **Binary**
 - **Hexadecimal**

- **Unsigned Decimal**
- **Signed Decimal**
- **Octal**
- **ASCII**



IMPORTANT: Changes to the radix of an item in the Objects window do not apply to values in the **Wave** window or the **Tcl** Console. To change the radix of an individual signal (ILA probe) in the **Wave** window, use the **Wave** window popup menu.

- Maximum bus width of 64 bits on real. Incorrect values are possible for buses wider than 64 bits.
- Floating point supports only 32- and 64-bit arrays.

Using the Floating Ruler

The floating ruler assists with time measurements using a sample number base other than the absolute sample numbers shown on the standard ruler at the top of the **Wave** window.

You can display (or hide) a floating ruler and move it to a location in the **Wave** window. The sample base (sample 0) of the floating ruler is the secondary cursor, or, if there is no secondary cursor, the selected marker.

The floating ruler button and the floating ruler itself are visible only when the secondary cursor (or selected marker) is present.

1. Do either of the following to display or hide a floating ruler:
 - Place the secondary cursor.
 - Select a marker.
2. Select **View > Floating Ruler**, or click the **Floating Ruler** button.



You only need to follow this procedure the first time. The floating ruler displays each time the secondary cursor is placed or a marker is selected.

Select the command again to hide the floating ruler.

Bus Bit Order

You can reverse the bus bit order in the wave configuration to switch between MSB-first and LSB-first signal representation.

To reverse the bit order:

1. Select a bus.
2. Right-click and select **Reverse Bit Order**.

The bus bit order is reversed. The **Reverse Bit Order** command is marked to show that this is the current behavior.

About Radixes and Analog Waveforms

Bus values are interpreted as numeric values, which are determined by the radix setting on the bus wave object, as follows:

- Binary, octal, hexadecimal, ASCII, and unsigned decimal radixes cause the bus values to be interpreted as unsigned integers. The format of data on the bus must match the radix setting.
- Any non-0 or -1 bits cause the entire value to be interpreted as 0.
- The signed decimal radix causes the bus values to be interpreted as signed integers.
- Real radixes cause bus values to be interpreted as fixed point or floating point real numbers, as determined by the settings of the **Real Settings** dialog box, shown in [Figure 6-3, page 59](#).

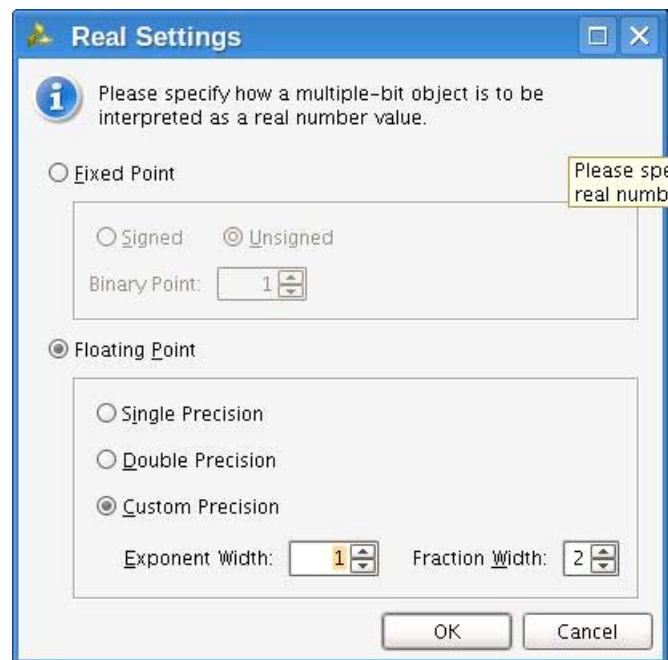


Figure 6-3: Real Settings Dialog Box

The options are as follows:

- **Fixed Point:** Specifies that the bits of the selected bus wave object(s) is interpreted as a fixed point, signed, or unsigned real number.

- **Binary Point:** Specifies how many bits to interpret as being to the right of the binary point. If **Binary Point** is larger than the bit width of the wave object, wave object values cannot be interpreted as fixed point, and when the wave object is shown in Digital waveform style, all values show as <Bad Radix>. When shown as analog, all values are interpreted as 0.
- **Floating Point:** Specifies that the bits of the selected bus wave object(s) should be interpreted as an IEEE floating point real number.

Note: Only single precision and double precision (and custom precision with values set to those of single and double precision) are supported.

Other values result in <Bad Radix> values as in **Fixed Point**.

Exponent Width and **Fraction Width** must add up to the bit width of the wave object, or else <Bad Radix> values result.

Viewing Analog Waveforms

To convert a digital waveform to analog, do the following:

1. In the **Name** area of a **Wave** window, right-click on the bus for the popup menu.
2. Select **Waveform Style** and then **Analog Settings** to choose an appropriate drawing setting.

The digital drawing of the bus converts to an analog format.

You can adjust the height of either an analog waveform or a digital waveform by selecting and then dragging the rows.

Figure 6-4 shows the **Analog Settings** dialog box with the settings for analog waveform drawing.

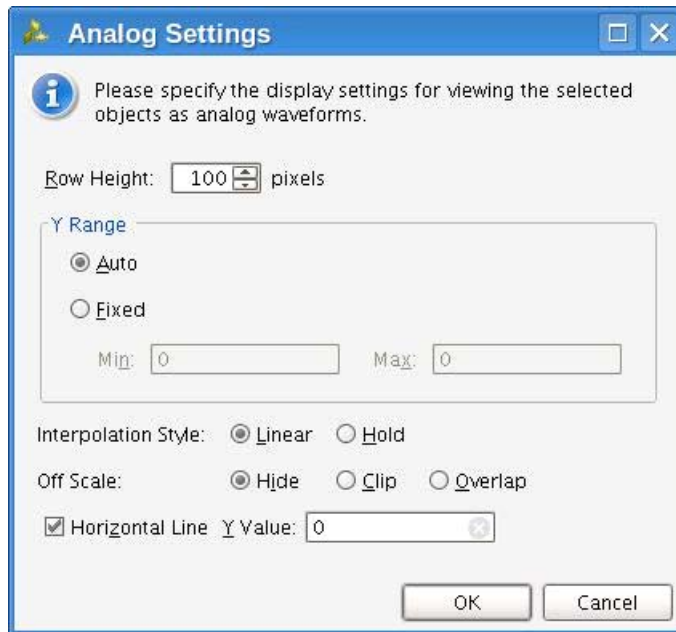


Figure 6-4: Analog Settings Dialog Box

The **Analog Settings** dialog box options are as follows:

- **Row Height:** Specifies how tall to make the select wave object(s), in pixels. Changing the row height does not change how much of a waveform is exposed or hidden vertically, but rather stretches or contracts the height of the waveform.

When switching between Analog and Digital waveform styles, the row height is set to an appropriate default for the style (20 for digital, 100 for analog).

- **Y Range:** Specifies the range of numeric values to be shown in the waveform area.
 - **Auto:** Specifies that the range should continually expand whenever values in the visible time range of the window are discovered to lie outside the current range.
 - **Fixed:** Specifies that the time range is to remain at a constant interval.
 - **Min:** Specifies the value displays at the bottom of the waveform area.
 - **Max:** Specifies the value displays at the top.

Both values can be specified as floating point; however, if radix of the wave object radix is integral, the values are truncated to integers.

- **Interpolation Style:** Specifies how the line connecting data points is to be drawn.
 - **Linear:** Specifies a straight line between two data points.

- **Hold:** Specifies that of two data points, a horizontal line is drawn from the left point to the X-coordinate of the right point, then another line is drawn connecting that line to the right data point, in an L shape.
- **Off Scale:** Specifies how to draw waveform values that lie outside the Y range of the waveform area.
- **Hide:** Specifies that outlying values are not shown, such that a waveform that reaches the upper or lower bound of the waveform area disappears until values are again within the range.
- **Clip:** Specifies that outlying values be altered so that they are at the top or bottom of the waveform area, such that a waveform that reaches the upper- or lower-bound of the waveform area follows the bound as a horizontal line until values are once again within the range.
- **Overlap:** Specifies that the waveform be drawn wherever its values are, even if they lie outside the bounds of the waveform area and overlap other waveforms, up to the limits of the wave window itself.
- **Horizontal Line:** Specifies whether to draw a horizontal rule at the given value. If the check-box is on, a horizontal grid line is drawn at the vertical position of the specified Y value, if that value is within the Y range of the waveform.

As with Min and Max, the Y value accepts a floating point number but truncates it to an integer if the radix of the selected wave objects is integral.



IMPORTANT: *Analog settings are saved in a wave configuration; however, because control of zooming in the Y dimension is highly interactive, unlike other wave object properties such as radix, they do not affect the modification state of the wave configuration. Consequently, zoom settings are not saved with the wave configuration.*

Zoom Gestures

In addition to the zoom gestures supported for zooming in the X dimension, when over an analog waveform, additional zoom gestures are available, as shown in Figure 6-5.

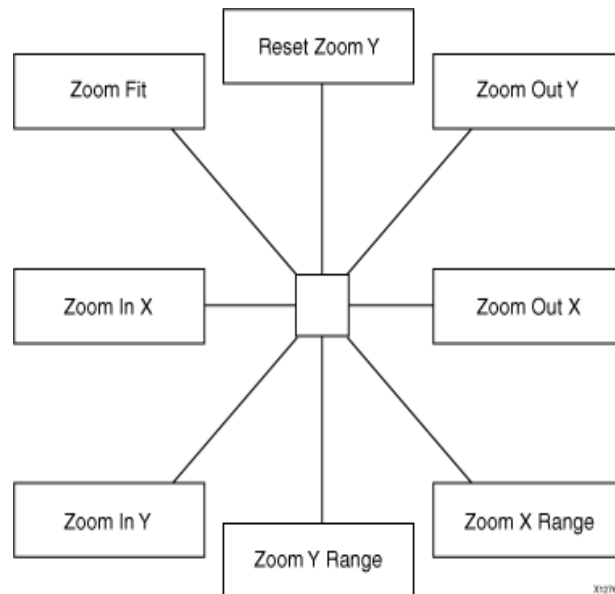


Figure 6-5: Analog Zoom Options

To invoke a zoom gesture, hold down the left mouse button and drag in the direction indicated in the diagram, where the starting mouse position is the center of the diagram.

The additional Zoom gestures are as follows:

- **Zoom Out Y:** Zooms out in the Y dimension by a power of 2 determined by how far away the mouse button is released from the starting point. The zoom is performed such that the Y value of the starting mouse position remains stationary.
- **Zoom Y Range:** Draws a vertical curtain which specifies the Y range to display when the mouse is released.
- **Zoom In Y:** Zooms in toward the Y dimension by a power of 2 determined by how far away the mouse button is released from the starting point.

The zoom is performed such that the Y value of the starting mouse position remains stationary.

- **Reset Zoom Y:** Resets the Y range to that of the values currently displayed in the wave window and sets the Y Range mode to **Auto**.

All zoom gestures in the Y dimension set the Y Range analog settings. **Reset Zoom Y** sets the Y Range to **Auto**, whereas the other gestures set Y Range to **Fixed**.

Device Configuration Bitstream Settings

Scope

This table describes all of the device configuration settings available for use with the Vivado™ logic analyzer.

Table A-1: User Table Formats

Settings	Default Value	Possible Values	Description
BITSTREAM.COFIG.BPI_1ST_READ_CYCLE	1	1, 2, 3, 4	Helps synchronize BPI configuration with the timing of page mode operations in Flash devices. It allows you to set the cycle number for a valid read of the first page. The BPI_page_size must be set to 4 or 8 for this option to be available.
BITSTREAM.COFIG.BPI_PAGE_SIZE	1	1, 4, 8	For BPI configuration, this sub-option lets you specify the page size which corresponds to the number of reads required per page of Flash memory.
BITSTREAM.CONFIG.BPI_SYNC_MODE	Disable	Disable, Type1, Type2	Sets the BPI synchronous configuration mode for different types of BPI flash devices. <ul style="list-style-type: none"> • Disable (the default) disables the synchronous configuration mode. • Type1 enables the synchronous configuration mode and settings to support the Micron G18(F) family. • Type2 enables the synchronous configuration mode and settings to support the Micron (Numonyx) P30 family.
BITSTREAM.CONFIG.CCLKPIN	Pullup	Pullup, Pullnone	Adds an internal pull-up to the Cclk pin. The Pullnone setting disables the pullup.
BITSTREAM.CONFIG.CONFIGFALLBACK	Disable	Disable, Enable	Enables or disables the loading of a default bitstream when a configuration attempt fails.
BITSTREAM.CONFIG.CONFIGRATE	3	3, 6, 9, 12, 16, 22, 26, 33, 40, 50, 66	Bitstream generation uses an internal oscillator to generate the configuration clock, Cclk, when configuring is in a master mode. Use this sub-option to select the rate for Cclk.

Table A-1: User Table Formats (Cont'd)

Settings	Default Value	Possible Values	Description
BITSTREAM.CONFIG.DCIUPDATEMODE	AsRequired	AsRequired, Continuous, Quiet	Controls how often the Digitally Controlled Impedance circuit attempts to update the impedance match for DCI IOSTANDARDS.
BITSTREAM.CONFIG.DONEPIN	Pullup	Pullup Pullnone	Adds an internal pull-up to the DONE pin. The Pullnone setting disables the pullup. Use DonePin only if you intend to connect an external pull-up resistor to this pin. The internal pull-up resistor is automatically connected if you do not use DonePin
BITSTREAM.CONFIG.EXTMASTERCCLK_EN	Disable	Disable, div-8, div-4, div-2, div-1	Allows an external clock to be used as the configuration clock for all master modes. The external clock must be connected to the dual-purpose USERCCLK pin.
BITSTREAM.CONFIG.INITPIN	Pullup	Pullup, Pullnone	Specifies whether you want to add a Pullup resistor to the INIT pin, or leave the INIT pin floating.
BITSTREAM.CONFIG.INITSIGNALSERROR	Enable	Enable, Disable	When Enabled the init_b pin will not be set to 0 when CFG error is detected
BITSTREAM.CONFIG.M0PIN	Pullup	Pullup, Pulldown, Pullnone	Adds an internal pull-up, pull-down, or neither to the M0 pin. Select Pullnone to disable both the pull-up resistor and the pull-down resistor on the M0 pin.
BITSTREAM.CONFIG.M1PIN	Pullup	Pullup, Pulldown, Pullnone	Adds an internal pull-up, pull-down, or neither to the M1 pin. Select Pullnone to disable both the pull-up resistor and the pull-down resistor on the M0 pin.
BITSTREAM.CONFIG.M2PIN	Pullup	Pullup, Pulldown, Pullnone	Adds an internal pull-up, pull-down, or neither to the M2 pin. Select Pullnone to disable both the pull-up resistor and the pull-down resistor on the M0 pin.
BITSTREAM.CONFIG.NEXT_CONFIG_ADDR	none	<string>	Sets the starting address for the next configuration in a MultiBoot setup, which is stored in the General1 and General2 registers.
BITSTREAM.CONFIG.NEXT_CONFIG_REBOOT	Enable	Enable, Disable	When set to Disable the IPROG command is removed from the bitfile
BITSTREAM.CONFIG.OVERTEMPPOWERDOWN	Disable	Disable, Enable	Enables the device to shut down when the system monitor detects a temperature beyond the acceptable operational maximum. An external circuitry setup for the System Monitor is required in order to use this option.
BITSTREAM.CONFIG.PERSIST	No	No, Yes, CTLReg, X1, X8, X16, X32, SPI1, SPI2, SPI4, BPI8, BPI	Prohibits use of the SelectMAP mode pins for use as user I/O. Refer to the data sheet for a description of SelectMAP mode and the associated pins. Persist is needed for Readback and Partial Reconfiguration using the SelectMAP configuration pins, and should be used when either SelectMAP or Serial modes are used. Only the SelectMAP pins are affected, but this option should be used for access to config pins (other than JTAG) after configuration.

Table A-1: User Table Formats (Cont'd)

Settings	Default Value	Possible Values	Description
BITSTREAM.CONFIG. PROGPIN	Pullup	Pullup, Pullnone	Adds an internal pull-up to the ProgPin pin. The Pullnone setting disables the pullup. The pullup affects the pin after configuration.
BITSTREAM.CONFIG. REVISIONSELECT	00	00, 01, 10, 11	Specifies the internal value of the RS[1:0] settings in the Warm Boot Start Address (WBSTAR) register for the next warm boot.
BITSTREAM.CONFIG. REVISIONSELECT_ TRISTATE	Disable	Disable, Enable	Specifies the whether the RS[1:0] tristate is enabled by setting the option in the Warm Boot Start Address (WBSTAR).
BITSTREAM.CONFIG. SELECTMAPABORT	Enable	Enable, Disable	Enables or disables the SelectMAP mode Abort sequence. If disabled, an Abort sequence on the device pins is ignored.
BITSTREAM.CONFIG. SPI_32BIT_ADDR	No	No, Yes	Enables SPI 32-bit address style, which is required for SPI devices with storage of 256 Mb and larger.
BITSTREAM.CONFIG. SPI_BUSWIDTH	1	1, 2, 4	Sets the SPI bus to Dual (x2) or Quad (x4) mode for Master SPI configuration from third party SPI Flash devices.
BITSTREAM.CONFIG. SPI_FALL_EDGE	No	No, Yes	Sets the FPGA to use a falling edge clock for SPI data capture. This improves timing margins and may allow faster clock rates for configuration.
BITSTREAM.CONFIG. TCKPIN	Pullup	Pullup, Pulldown, Pullnone	Adds a pull-up, a pull-down, or neither to the TCK pin, the JTAG test clock. The Pullnone setting shows that there is no connection to either the pull-up or the pull-down.
BITSTREAM.CONFIG. TDIPIN	Pullup	Pullup, Pulldown, Pullnone	Adds a pull-up, a pull-down, or neither to the TDI pin, the serial data input to all JTAG instructions and JTAG registers. The Pullnone setting shows that there is no connection to either the pull-up or the pull-down.
BITSTREAM.CONFIG. TDOPIN	Pullup	Pullup, Pulldown, Pullnone	Adds a pull-up, a pull-down, or neither to the TdoPin pin, the serial data output for all JTAG instruction and data registers. The Pullnone setting shows that there is no connection to either the pull-up or the pull-down.
BITSTREAM.CONFIG. TIMER_CFG	none	<8-digit hex string>	Sets the value of the Watchdog Timer in Configuration mode. This option cannot be used at the same time as TIMER_USR.
BITSTREAM.CONFIG. TIMER_USR	0x00000000	<8-digit hex string>	Sets the value of the Watchdog Timer in User mode. This option cannot be used at the same time as TIMER_CFG.
BITSTREAM.CONFIG. TMSPIN	Pullup	Pullup, Pulldown, Pullnone	Adds a pull-up, pull-down, or neither to the TMS pin, the mode input signal to the TAP controller. The TAP controller provides the control logic for JTAG. The Pullnone setting shows that there is no connection to either the pull-up or the pull-down.

Table A-1: User Table Formats (Cont'd)

Settings	Default Value	Possible Values	Description
BITSTREAM.CONFIG. UNUSEDPIN	Pulldown	Pulldown, Pullup, Pullnone	Adds a pull-up, a pull-down, or neither to unused SelectIO pins (IOBs). It has no effect on dedicated configuration pins. The list of dedicated configuration pins varies depending upon the architecture. The Pullnone setting shows that there is no connection to either the pull-up or the pull-down.
BITSTREAM.CONFIG. USERID	0xFFFFFFFF	<8-digit hex string>	Used to identify implementation revisions. You can enter up to an 8-digit hexadecimal string in the User ID register.
BITSTREAM.CONFIG. USR_ACCESS	None	none, <8-digit hex string>, TIMESTAMP	Writes an 8-digit hexadecimal string, or a timestamp into the AXSS configuration register. The format of the timestamp value is dddddd MMMM yyyy hh mm ss : day, month, year (year 2000 = 00000), hour, minute, seconds. The contents of this register may be directly accessed by the FPGA fabric via the USR_ACCESS primitive
BITSTREAM.ENCRYPTION. ENCRYPT	No	No Yes	Encrypts the bitstream.
BITSTREAM.ENCRYPTION. ENCRYPTKEYSELECT	bbram	bbram, efuse	Determines the location of the AES encryption key to be used, either from the battery-backed RAM (BBRAM) or the eFUSE register. Note: This property is only available when the Encrypt option is set to True.
BITSTREAM.ENCRYPTION. HKEY	Pick	Pick, <hex string>	HKey sets the HMAC authentication key for bitstream encryption. 7 series devices have an on-chip bitstream-keyed Hash Message Authentication Code (HMAC) algorithm implemented in hardware to provide additional security beyond AES decryption alone. These devices require both AES and HMAC keys to load, modify, intercept, or clone the bitstream. The pick setting tells the bitstream generator to select a random number for the value. To use this option, you must first set Encrypt to Yes
BITSTREAM.ENCRYPTION. KEY0	Pick	Pick, <hex string>	Key0 sets the AES encryption key for bitstream encryption. The pick setting tells the bitstream generator to select a random number for the value. To use this option, you must first set Encrypt to Yes.
BITSTREAM.ENCRYPTION. KEYFILE	none	<string>	Specifies the name of the input encryption file (with a .nky file extension). To use this option, you must first set Encrypt to Yes.
BITSTREAM.ENCRYPTION. STARTCBC	Pick	Pick, <32-bit hex string>	Sets the starting cipher block chaining (CBC) value. The pick setting enables selection of a random number for the value.

Table A-1: User Table Formats (Cont'd)

Settings	Default Value	Possible Values	Description
BITSTREAM.GENERAL.COMPRESS	False	True, False	Uses the multiple frame write feature in the bitstream to reduce the size of the bitstream, not just the Bitstream (.bit) file. Using Compress does not guarantee that the size of the bitstream will shrink.
BITSTREAM.GENERAL.CRC	Enable	Enable, Disable	Controls the generation of a Cyclic Redundancy Check (CRC) value in the bitstream. When enabled, a unique CRC value is calculated based on bitstream contents. If the calculated CRC value does not match the CRC value in the bitstream, the device will fail to configure. When CRC is disabled a constant value is inserted in the bitstream in place of the CRC, and the device does not calculate a CRC.
BITSTREAM.GENERAL.DEBUGBITSTREAM	No	No, Yes	Lets you create a debug bitstream. A debug bitstream is significantly larger than a standard bitstream. DebugBitstream can be used only for master and slave serial configurations. DebugBitstream is not valid for Boundary Scan or Slave Parallel/SelectMAP. In addition to a standard bitstream, a debug bitstream offers the following features: <ul style="list-style-type: none"> Writes 32 0s to the LOUT register after the synchronization word. Loads each frame individually. Performs a Cyclic Redundancy Check (CRC) after each frame. Writes the frame address to the LOUT register after each frame.
BITSTREAM.GENERAL.DISABLE_JTAG	No	No, Yes	Disables communication to the Boundary Scan (BSCAN) block via JTAG after configuration.
BITSTREAM.GENERAL.JTAG_XADC	Enable	Enable, Disable, StatusOnly	Enables or disables the JTAG connection to the XADC.
BITSTREAM.GENERAL.XADCENHANCEDLINEARITY	Off	Off, On	Disables some built-in digital calibration features that make INL look worse than the actual analog performance.
BITSTREAM.READBACK.ACTIVERECONFIG	No	No, Yes	Prevents the assertions of GHIGH and GSR during configuration. This is required for the active partial reconfiguration enhancement features.
BITSTREAM.READBACK.ICAP_SELECT	Auto	Auto, Top, Bottom	Selects between the top and bottom ICAP ports.
BITSTREAM.READBACK.READBACK	False	True, False	Lets you perform the Readback function by creating the necessary readback files.
BITSTREAM.READBACK.SECURITY	None	None, Level1, Level2	Specifies whether to disable Readback and Reconfiguration. Note: Specifying Security Level1 disables Readback. Specifying Security Level2 disables Readback and Reconfiguration.

Table A-1: User Table Formats (Cont'd)

Settings	Default Value	Possible Values	Description
BITSTREAM.READBACK. XADCPARTIALRECONFIG	Disable	Disable, Enable	When Disabled XADC can work continuously during Partial Reconfiguration. When Enabled XADC will work in Safe mode during partial reconfiguration.
BITSTREAM.STARTUP. DONEPIPE	Yes	Yes, No	Tells the FPGA device to wait on the CFG_DONE (DONE) pin to go High and then wait or the first clock edge before moving to the Done state.
BITSTREAM.STARTUP. DONE_CYCLE	4	4, 1, 2, 3, 5, 6, Keep	Selects the Startup phase that activates the FPGA Done signal. Done is delayed when DonePipe=Yes.
BITSTREAM.STARTUP. GTS_CYCLE	5	5, 1, 2, 3, 4, 6, Done, Keep	Selects the Startup phase that releases the internal tristate control to the I/O buffers.
BITSTREAM.STARTUP. GWE_CYCLE	6	6, 1, 2, 3, 4, 5, Done, Keep	Selects the Startup phase that asserts the internal write enable to flip-flops, LUT RAMs, and shift registers. GWE_cycle also enables the BRAMS. Before the Startup phase, both BRAM writing and reading are disabled.
BITSTREAM.STARTUP. LCK_CYCLE	NoWait	NoWait, 0, 1, 2, 3, 4, 5, 6	Selects the Startup phase to wait until DLLs/DCMs/PLLs lock. If you select NoWait, the Startup sequence does not wait for DLLs/DCMs/PLLs to lock.
BITSTREAM.STARTUP. MATCH_CYCLE	Auto	Auto, NoWait, 0, 1, 2, 3, 4, 5, 6	Specifies a stall in the Startup cycle until digitally controlled impedance (DCI) match signals are asserted. DCI matching does not begin on the Match_cycle that was set in BitGen. The Startup sequence simply waits in this cycle until DCI has matched. Given that there are a number of variables in determining how long it will take DCI to match, the number of CCLK cycles required to complete the Startup sequence may vary in any given system. Ideally, the configuration solution should continue driving CCLK until DONE goes high.
BITSTREAM.STARTUP. STARTUPCLK	Cclk	Cclk, UserClk, JtagClk	The StartupClk sequence following the configuration of a device can be synchronized to either Cclk, a User Clock, or the JTAG Clock. The default is Cclk. <ul style="list-style-type: none"> Cclk lets you synchronize to an internal clock provided in the FPGA device. UserClk lets you synchronize to a user-defined signal connected to the CLK pin of the STARTUP symbol. JtagClk lets you synchronize to the clock provided by JTAG. This clock sequences the TAP controller which provides the control logic for JTAG.

Additional Resources

Xilinx Resources

For support resources such as Answers, Documentation, Downloads, and Forums, see the Xilinx Support website at:

www.xilinx.com/support.

For a glossary of technical terms used in Xilinx documentation, see:

www.xilinx.com/company/terms.htm.

Solution Centers

See the [Xilinx Solution Centers](#) for support on devices, software tools, and intellectual property at all stages of the design cycle. Topics include design assistance, advisories, and troubleshooting tips.

References

These documents provide supplemental material useful with this guide:

Vivado Design Suite 2012.2 Documentation

(<http://www.xilinx.com/cgi-bin/docs/rdoc?v=2012.2;t=vivado+docs>)

1. *Vivado Design Suite User Guide: Logic Simulation (UG937)*
2. *Vivado Design Suite User Guide: Synthesis (UG901)*
3. *Vivado Design Suite User Guide: Implementation (UG904)*
4. *Vivado Design Suite Installation and Licensing Guide (UG798)*

ISE Design Suite 14.2 Documentation

5. iMPACT Help

(http://www.xilinx.com/cgi-bin/docs/rdoc?l=en;v=14.2;d=isehelp_start.htm;a=pim_c_overview)

LogiCORE IP ChipScope Pro Integrated Logic Analyzer (ILA) (v2) Datasheet (DS875)

(<http://www.xilinx.com/cgi-bin/docs/ndoc?l=en;t=data+sheet;d=ds875.pdf>) .