

Identify[®] Microsemi Edition Quick Tutorial

November 2014

SYNOPSYS[®]

Copyright and Trademarks

Copyright Notice and Proprietary Information

Copyright © 2014 Synopsys, Inc. All rights reserved. This software and documentation contain confidential and proprietary information that is the property of Synopsys, Inc. The software and documentation are furnished under a license agreement and may be used or copied only in accordance with the terms of the license agreement. No part of the software and documentation may be reproduced, transmitted, or translated, in any form or by any means, electronic, mechanical, manual, optical, or otherwise, without prior written permission of Synopsys, Inc., or as expressly provided by the license agreement.

Right to Copy Documentation

The license agreement with Synopsys permits licensee to make copies of the documentation for its internal use only.

Each copy shall include all copyrights, trademarks, service marks, and proprietary rights notices, if any. Licensee must assign sequential numbers to all copies. These copies shall contain the following legend on the cover page:

“This document is duplicated with the permission of Synopsys, Inc., for the exclusive use of _____ and its employees. This is copy number _____.”

Destination Control Statement

All technical data contained in this publication is subject to the export control laws of the United States of America. Disclosure to nationals of other countries contrary to United States law is prohibited. It is the reader's responsibility to determine the applicable regulations and to comply with them.

Disclaimer

SYNOPSYS, INC., AND ITS LICENSORS MAKE NO WARRANTY OF ANY KIND, EXPRESS OR IMPLIED, WITH REGARD TO THIS MATERIAL, INCLUDING, BUT NOT LIMITED TO, THE IMPLIED WARRANTIES OF MERCHANTABILITY AND FITNESS FOR A PARTICULAR PURPOSE.

Registered Trademarks (®)

Synopsys, AEON, AMPS, Astro, Behavior Extracting Synthesis Technology, Cadabra, CATS, Certify, CHIPit, CoMET, CODE V, Design Compiler, DesignWare, EMBED-IT!, Formality, Galaxy Custom Designer, Global Synthesis, HAPS, HapsTrak, HDL Analyst, HSIM, HSPICE, Identify, Leda, LightTools, MAST, METeor, ModelTools, NanoSim, NOVeA, OpenVera, ORA, PathMill, Physical Compiler, PrimeTime, SCOPE, Simply Better Results, SiVL, SNUG, SolvNet, Sonic Focus, STAR Memory System, Syndicated, Synplicity, the Synplicity logo, Synplify, Synplify Pro, Synthesis Constraints Optimization Environment, TetraMAX, UMRBus, VCS, Vera, and YIELDirector are registered trademarks of Synopsys, Inc.

Trademarks (™)

AFGen, Apollo, ARC, ASAP, Astro-Rail, Astro-Xtalk, Aurora, AvanWaves, BEST, Columbia, Columbia-CE, Cosmos, CosmosLE, CosmosScope, CRITIC, CustomExplorer, CustomSim, DC Expert, DC Professional, DC Ultra, Design Analyzer, Design Vision, DesignerHDL, DesignPower, DFTMAX, Direct Silicon Access, Discovery, Eclipse, Encore, EPIC, Galaxy, HANEX, HDL Compiler, Hercules, Hierarchical Optimization Technology, High-performance ASIC Prototyping System, HSIMplus, i-Virtual Stepper, IICE, in-Sync, iN-Tandem, Intelli, Jupiter, Jupiter-DP, JupiterXT, JupiterXT-ASIC, Liberty, Libra-Passport, Library Compiler, Macro-PLUS, Magellan, Mars, Mars-Rail, Mars-Xtalk, Milkyway, ModelSource, Module Compiler, MultiPoint, ORAengineering, Physical Analyst, Planet, Planet-PL, Polaris, Power Compiler, Raphael, RippledMixer, Saturn, Scirocco, Scirocco-i, SiWare, Star-RCXT, Star-SimXT, StarRC, System Compiler, System Designer, Taurus, Total-Recall, TSUPREM-4, VCSi, VHDL Compiler, VMC, and Worksheet Buffer are trademarks of Synopsys, Inc.

Service Marks (sm)

MAP-in, SVP Café, and TAP-in are service marks of Synopsys, Inc.

SystemC is a trademark of the Open SystemC Initiative and is used under license.

ARM and AMBA are registered trademarks of ARM Limited.

Saber is a registered trademark of SabreMark Limited Partnership and is used under license.

All other product or company names may be trademarks of their respective owners.

Printed in the U.S.A
November 2014

Introduction

This simple tutorial teaches you how to instrument and debug a small HDL design. The design is a simple 4-bit counter with a clock and reset. Two versions of the counter are provided: one in VHDL and one in Verilog.

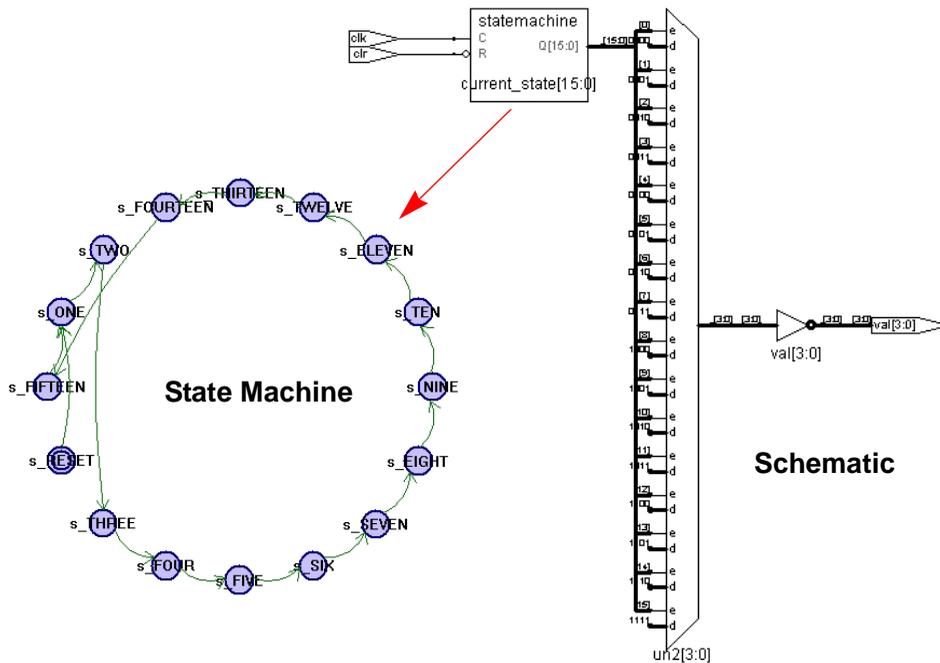
Note: This tutorial simulates hardware debug data by applying randomly generated data to all instrumented nodes. This data does not reflect the actual operation of the design and only serves to show the format of the debug data.

This tutorial introduces the reader to basic Identify operations. The tutorial includes the following major topics which are intended to be performed in the order listed below.

- [Instrumenting Your Design](#), on page 7
- [Setting up the IICE](#), on page 10
- [Writing the Instrumented Design](#), on page 14
- [Debugging Your Design](#), on page 15
- [Selecting the Cable Type](#), on page 16
- [Triggering on a Breakpoint](#), on page 17
- [Triggering on a Watchpoint](#), on page 19
- [Using the Complex Counter](#), on page 20
- [Generating Waveforms](#), on page 21

Design Schematic

The following figure shows the simple state machine configured as a 4-bit counter. The state diagram is shown to the left of the schematic.



Design Description

The tutorial design is implemented in Verilog as a single module with two always block statements and in VHDL as a single entity with two processes. The first always block (Verilog) or process (VHDL) implements a state machine; the second always block or process computes the output values based on the current state.

Instrumenting Your Design



You use the Identify instrumentor to select both breakpoints and watchpoints and to set the sampling and triggering modes. The Identify instrumentor is launched from the Synplify Pro synthesis tool and is run prior to synthesis.

The HDL design and project files for this tutorial are included in one of two “counter” subdirectories under the `share/demo_design` directory in the Identify software installation. Before you begin the tutorial, copy the appropriate (VHDL or Verilog) “counter” subdirectory to a local directory and make sure that you have read and write permission for the directory and files.

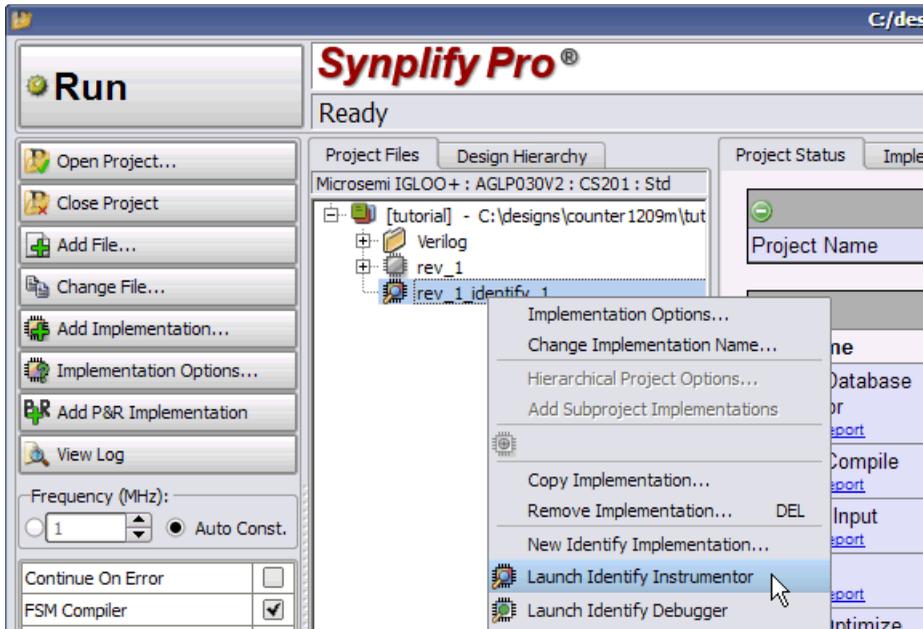
Note: While performing the tutorial, the active project (`.prj`) file will be updated; copying the files to a local directory preserves the original files installed in the share directory.

To begin the instrumentation:

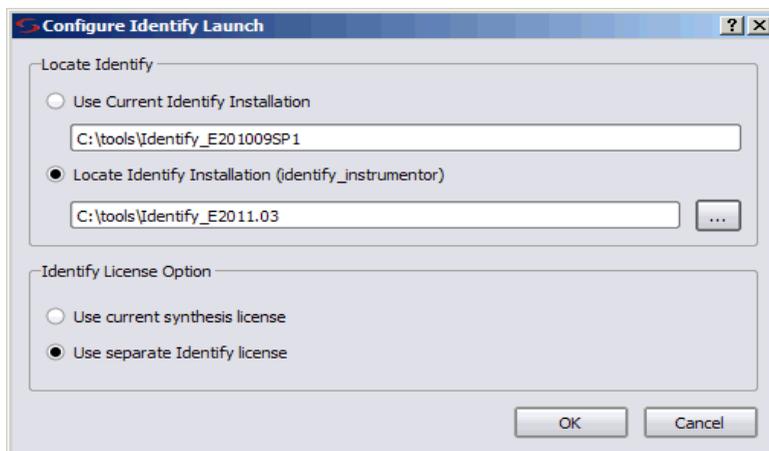
1. Start the Synplify Pro synthesis tool.
2. In the project view, click the Open Project button to display the Open Project dialog box and click the Existing Project button.
3. Navigate to the location where you copied the `counter_verilog` or `counter_vhdl` subdirectory. Each subdirectory includes an HDL design file (`counter.v` or `counter.vhd`) and a Microsemi-specific project file (`counter_verilog.prj` or `counter_vhdl.prj`).
4. Select (open) the corresponding Microsemi-specific project file.
5. Select File->Save As from the menu and rename the selected project file to `tutorial.prj`.

Note: The remainder of this document uses the term *tutorial* to reference the selected Verilog or VHDL project.

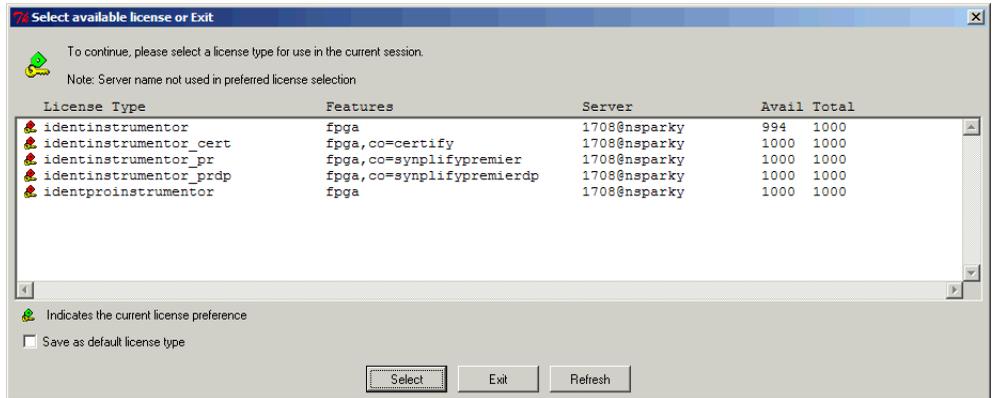
- Right click on the Identify implementation and select Launch Identify Instrumentor from the popup menu.



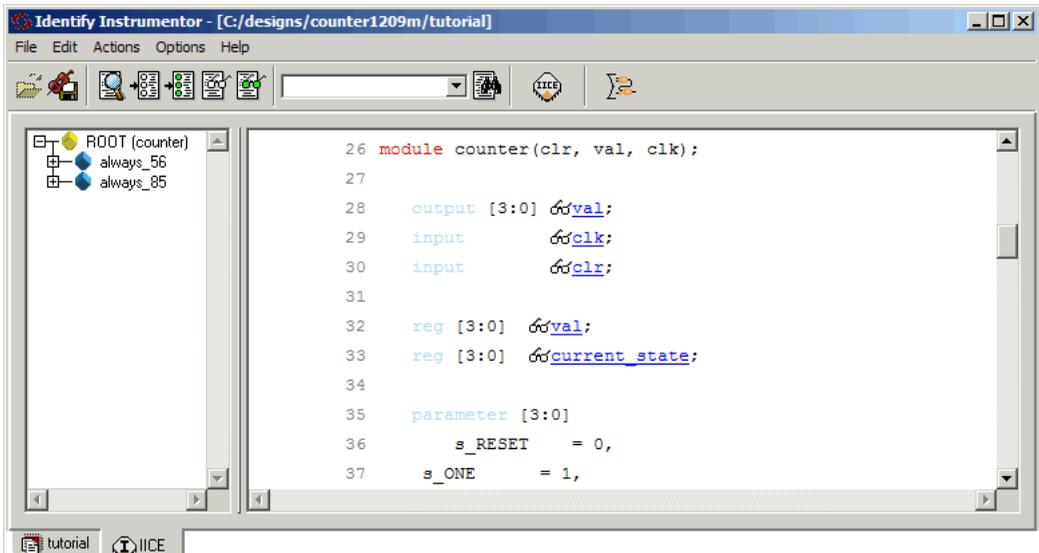
- If prompted, enter the location of the Identify installation in the Configure Identify Launch dialog box, click the Locate Identify Installation radio button, and click OK to launch the Identify instrumentor.



8. If prompted for a license, select a license from the list of available licenses displayed and click Select.



The figure below shows the initial Identify instrumentor window as launched from the Synplify Pro synthesis tool. The window shows the design hierarchy on the left and the HDL file content with all the potential instrumentation marked and available for selection on the right.



Setting up the IICE



Click on the Edit IICE settings icon on the toolbar to bring up the IICE Sampler tab shown in the following figure. The IICE Sampler tab defines the sample depth, sampling modes, and the sample clock.

On the IICE Sampler tab:

1. Leave Buffer type set to `internal_memory`
2. Select 128 for the sample buffer depth.
3. Leave the Allow qualified sampling check box unchecked.
4. Leave the Allow always-armed sampling check box unchecked.
5. Leave the Allow data compression check box unchecked.
6. Enter `/clk` for the sample clock and select positive polarity for the clock edge.
7. After you have set and/or verified the above IICE Sampler tab settings, click the IICE Controller tab.

The IICE Controller tab selects the type of triggering.

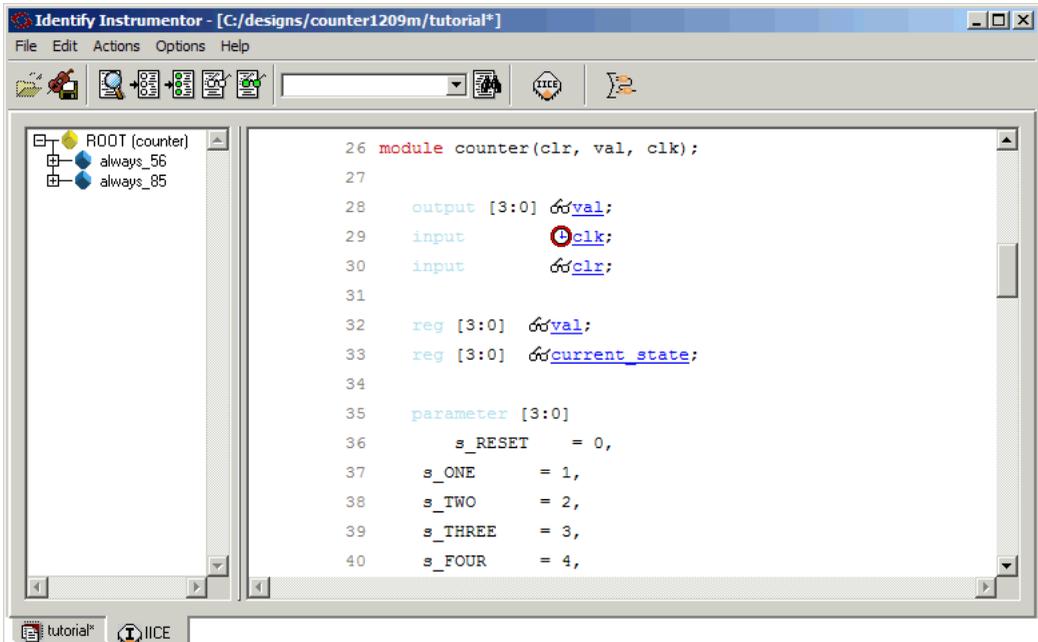
The screenshot shows the 'IICE Controller' dialog box. At the top, there is a tab labeled 'IICE Controller'. Below the tab, there are two fields: 'Current IICE:' with a dropdown menu showing 'IICE' and 'IICE type:' with a text field containing 'regular'. The main area is divided into two sections. The first section, 'IICE Controller', contains three radio buttons: 'Simple triggering' (unselected), 'Complex counter triggering' (selected), and 'State Machine triggering' (unselected). Below these are three spinners: 'Width:' set to 16, 'Trigger states:' set to 4, and 'Trigger conditions:' set to 4. The second section, 'IICE Options', contains a dropdown for 'Import external trigger signals:' set to 0, and two checkboxes: 'Export IICE trigger signal' (unchecked) and 'Allow cross-triggering in IICE' (unchecked).

On the IICE Controller tab:

1. Make sure that the Complex counter triggering radio button is selected and that the Width is set to 16.
2. Leave the Import external trigger signals set to 0.
3. Leave the Export IICE trigger signal check box unchecked; the Allow cross-triggering in IICE check box cannot be selected until a second IICE unit is created.
4. Click the OK button at the bottom of the dialog box.

Selecting the Instrumentation

After setting up the IICE, the HDL code for the tutorial design is displayed in the Identify instrumentor window as shown in the following figure. Use the hierarchy browser on the left to navigate through your design. Clicking on a hierarchical node displays the corresponding section of the source code.



Selecting Watch Points



In the source code display, scroll down and select the signal `current_state` on line 53 (VHDL) or line 33 (Verilog) for instrumentation by clicking on the watch-point (glasses) icon displayed next to its name. When you click on the icon (or on the signal name), a popup menu is displayed as shown in the following figure to allow you to select how the watch-point signal is to be instrumented.

```

53  signal current_state: state:
54  begin
55
56  process(clk, clr)
57  begin
58      if clr = '0' then
59          current_state <= s_RESET;
60      elsif clk'event and clk = '1' then
61          case current_state is
        62              when s_RESET    => current_state <= s_ONE;
        63              when s_ONE     => current_state <= s_TWO;
        64              when s_TWO    => current_state <= s_THREE;
        65              when s_THREE   => current_state <= s_FOUR;
        66              when s_FOUR    => current_state <= s_FIVE;
    
```

Select Sample and trigger for the current_state signal. The icons preceding each occurrence of the signal in the HDL code will be green and an accumulation of the total number of bits for each instrumentation type will be displayed in the console window.

Note that when you select an instrumentation type, the icon changes color according to the following table.

Icon Color	Watch-Point Selection
Green	Sample and trigger
Blue	Sample only
Pink	Trigger only
Clear (unfilled)	Not instrumented

Selecting Breakpoints



The circular icons to the left of the line numbers beginning on line 62 select the corresponding breakpoint for instrumentation. When selected, the color of the icon changes to green. Click on the icons on lines 63, 69, and 70 to select their corresponding breakpoints.

```

57  begin
58    if  $\overline{clk}$  = '0' then
59       $\overline{current\_state}$  <= s_RESET;
60    elsif  $\overline{clk}$ 'event and  $\overline{clk}$  = '1' then
61      case  $\overline{current\_state}$  is
62        when s_RESET    =>  $\overline{current\_state}$  <= s_ONE;
63        when s_ONE      =>  $\overline{current\_state}$  <= s_TWO;
64        when s_TWO      =>  $\overline{current\_state}$  <= s_THREE;
65        when s_THREE    =>  $\overline{current\_state}$  <= s_FOUR;
66        when s_FOUR     =>  $\overline{current\_state}$  <= s_FIVE;
67        when s_FIVE     =>  $\overline{current\_state}$  <= s_SIX;
68        when s_SIX      =>  $\overline{current\_state}$  <= s_SEVEN;
69        when s_SEVEN    =>  $\overline{current\_state}$  <= s_EIGHT;
70        when s_EIGHT    =>  $\overline{current\_state}$  <= s_NINE;

```

Writing the Instrumented Design



To write the instrumented design, select File->Save project instrumentation from the menu or click on the Save project's activated instrumentation icon on the toolbar. Saving the project automatically adds a set of files to the Identify implementation directory which are then used by the synthesis tool to incorporate the instrumented logic into the design.

At this point, you would:

- synthesize the design in the Synplify Pro synthesis tool to generate the output netlist
- place and route the synthesized output netlist in the Libero place and route tool
- program the resultant bit file into the FPGA
- cable the board containing the programmed FPGA to your host for analysis by the Identify debugger

Debugging Your Design

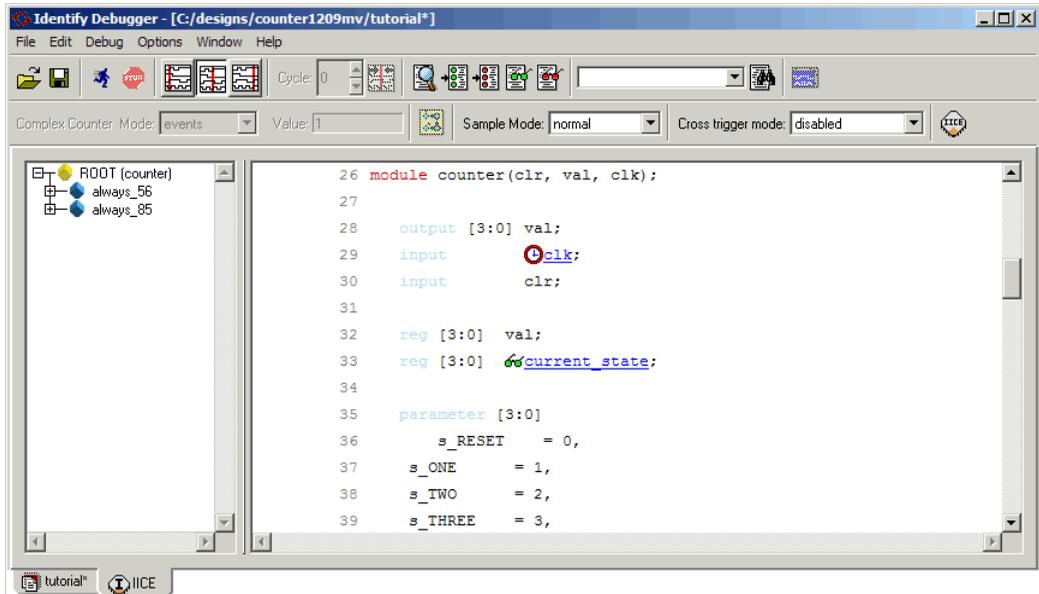
Debugging your design is done from the Identify debugger. To launch the debugger from the Synplify Pro synthesis tool:

1. Open the tutorial project in the synthesis tool and highlight the Identify implementation in the Project view.
2. With the right mouse button, select Launch Identify Debugger from the popup menu or click the Launch Identify Debugger icon in the top menu bar.

If you are prompted for a license, select the appropriate license from the list of available licenses displayed.

Note: To avoid being prompted for a license each time you start the Identify debugger, check the Save as default license type box before selecting your license.

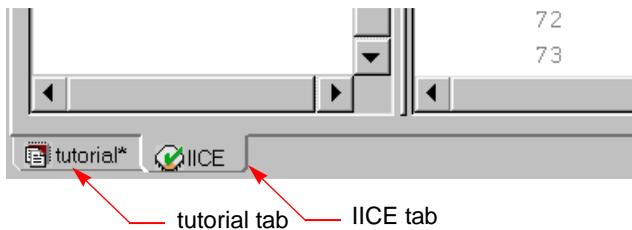
The Identify debugger opens your project in the instrumentation window with the hierarchy browser displayed on the left and the HDL source code displayed on the right as shown in the following figure. Note that the only instrumentation visible in the source code display are the breakpoints and watchpoints that you selected during the instrumentation phase with the Identify instrumentor.



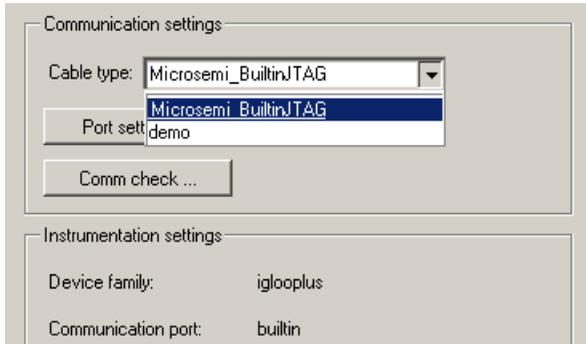
Selecting the Cable Type

To run the tutorial, select the “demo” cable type by:

1. Clicking on the “tutorial” tab at the lower left corner of the window to display the project window.



2. Selecting demo from the Cable type drop-down menu.



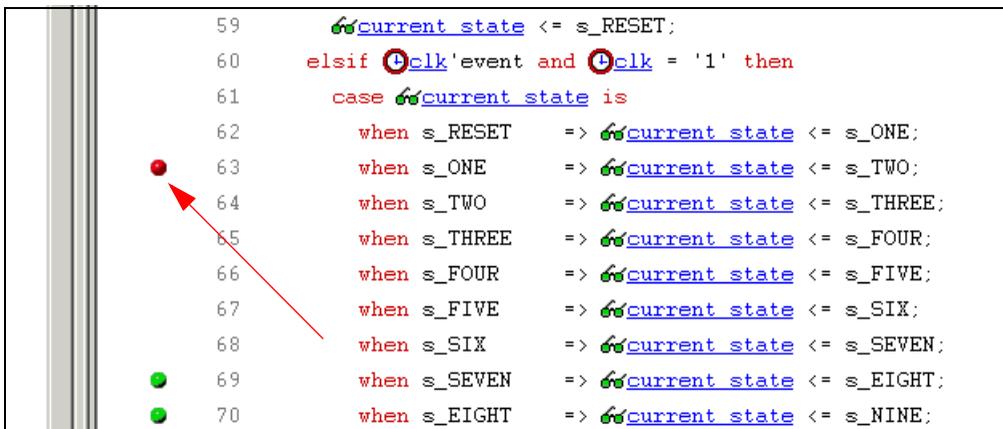
If a demo selection is not available, enter the following Tcl command at the console window prompt:

```
com cabletype demo
```

3. Clicking on the IIICE tab at the lower left corner of the window to redisplay the instrumentation window.

Triggering on a Breakpoint

In the source code display, use the scroll bar to scroll down until the first breakpoint on line 63 is visible on the left side of the source code and then click on the breakpoint to activate it.



Notice that the breakpoint icon changes from green to red indicating that the breakpoint is active. The breakpoint at line 63 triggers on the positive edge of the sample clock when the `current_state` signal has the value `s_ONE`.



Now that you have an active trigger condition, arm the IICE trigger circuits on the FPGA device by clicking the Run icon in the menu bar.

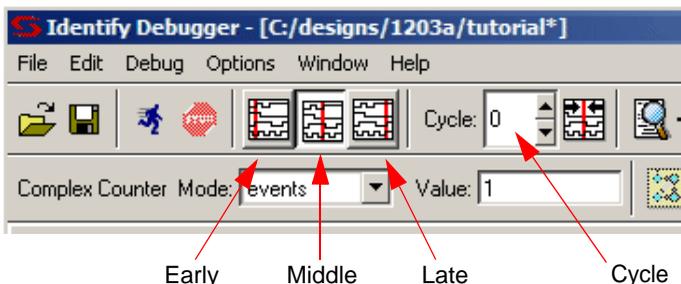
Clicking on the Run icon downloads the trigger information to the IICE. When the trigger occurs, the sampled data is transferred back to the debugger (when using Verilog, the value of the sampled data returned is highlighted). The small arrow to the left of the breakpoint icon indicates which breakpoint triggered (identifying which breakpoint triggered is important when multiple breakpoints are active).

```

59     current_state4'b0110 = s_RESET; /* 4'b0000 */
60     else begin
61         case (current_state4'b0110)
62     s_RESET: current_state4'b0110 = s_ONE;
63     s_ONE: current_state4'b0110 = s_TWO;
64     s_TWO: current_state4'b0110 = s_THREE;
65     s_THREE: current_state4'b0110 = s_FOUR;
66     s_FOUR: current_state4'b0110 = s_FIVE;
67     s_FIVE: current_state4'b0110 = s_SIX;
68     s_SIX: current_state4'b0110 = s_SEVEN;
69     s_SEVEN: current_state4'b0110 = s_EIGHT;
70     s_EIGHT: current_state4'b0110 = s_NINE;

```

The Cycle display in the middle of the menu bar shows the value zero where the trigger occurred. By clicking on the up-down arrows on the right, you can increase or decrease the cycle count to show values immediately before or after the trigger point.



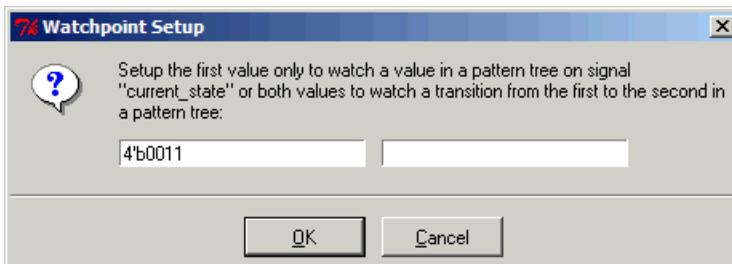
You can change where the trigger point is in the buffer by selecting one of the Early, Middle, or Late icons to the left of the cycle counter and again clicking the Run icon. The trigger location changes the next time the IICE triggers.

Triggering on a Watchpoint

You can also trigger on a watchpoint that is specified on any sampled signal. The Watchpoint Setup dialog box accepts any legal VHDL or Verilog expression that evaluates to a constant.

To set a simple watchpoint:

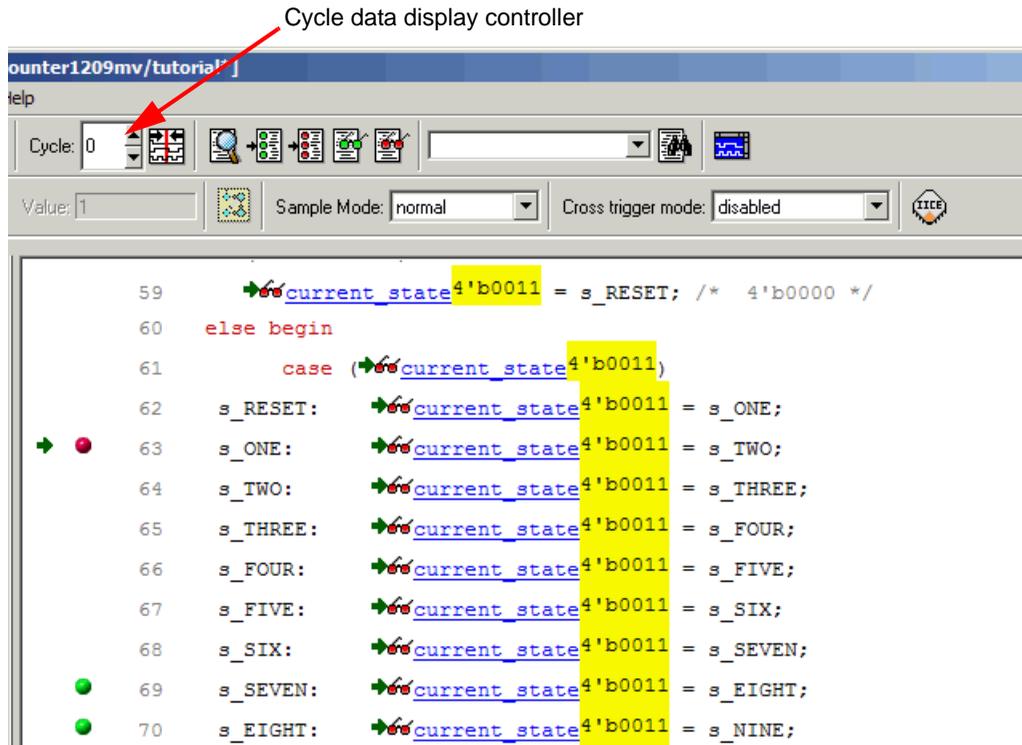
1. Click on the `current_state` signal
2. Select Set trigger expressions from the popup menu
3. In the first (left) field, enter `s_THREE` (VHDL) or `4'b0011` (Verilog)
4. Click OK



Click the Run icon. When signal `current_state` reaches the value `s_THREE` (VHDL) or `4'b0011` (Verilog), the IICE triggers.

Note: Because randomly generated data is applied, the trigger watchpoint (`s_THREE` or `4'b0011`) may not reach its intended value. Click the adjacent STOP icon if triggering does not occur within a few seconds.

Using the Cycle data display controller, you can now browse back and forth through the debugger data buffer to view the design activity.



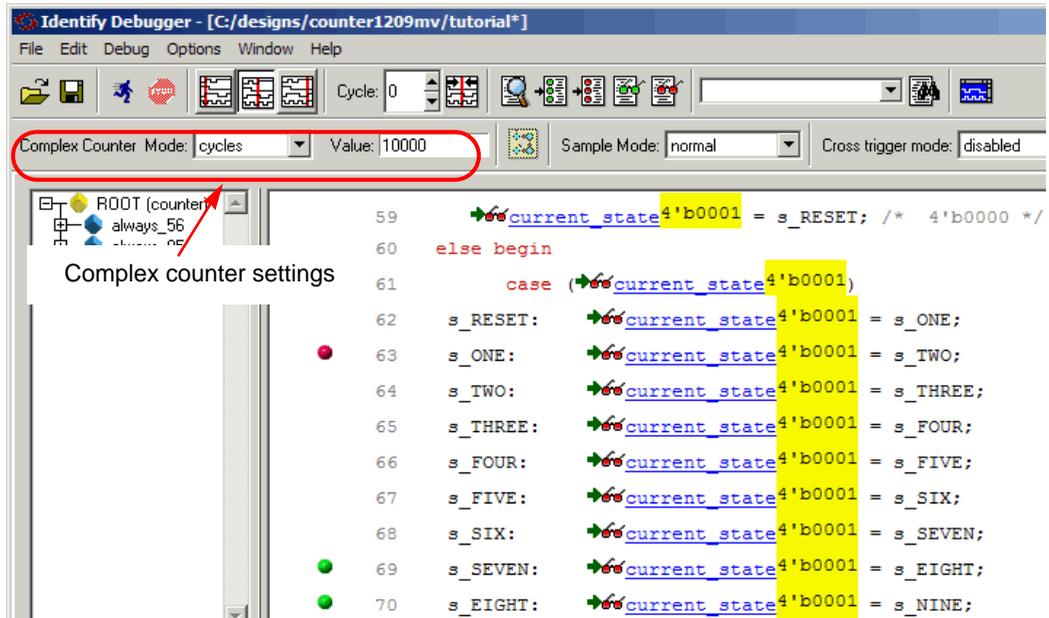
Using the Complex Counter

The default settings for the complex counter mode (events with a value of 1) effectively disable the counter. To use the complex counter to wait for a breakpoint and/or watchpoint trigger event and then to count a specified number of cycles before triggering the sample buffer:

1. Set the counter mode to cycles and the counter value to a value greater than 1 (note that you must have previously enabled Complex counter triggering on the IICE Controller tab in the Identify instrumentor).
2. Change the watchpoint of signal `current_state` to `s_ONE` (VHDL) or `4'b0001` (Verilog).
3. Click the Run icon and wait for the data to download.

The value at time zero will be updated with the sample data after the specified number of cycles has occurred as shown in the following figure.

Note: Because randomly generated data is applied to all instrumented nodes, the results displayed do not reflect actual design operation.



Generating Waveforms



Display the debug data by clicking the Open Waveform Display icon in the menu bar.

All sampled signals are included in the waveform display with two additional signals automatically added at the top of the display. The first signal, `identify_cycle`, shows the trigger location in the sample buffer. The second signal, `identify_sampleclock`, shows every clock edge. The following figure shows a typical waveform view with the `identify_cycle` and `identify_sampleclock` signals highlighted.

