HDL Design with Precision RTL Synthesis: CPLD Flow Tutorial

Lattice Semiconductor Corporation
5555 NE Moore Court
Hillsboro, OR 97124
(503) 268-8000

December 2005

Copyright

Trademarks

Disclaimers

Table of Contents

1

HDL Design with Precision RTL Synthesis: CPLD Flow Tutorial

This tutorial shows you how to use Mentor Graphics® Precision® RTL Synthesis from within ispLEVER® to synthesize a Verilog design and generate an EDIF file for a Lattice CPLD device.

Note

If you want to learn how to use Precision RTL Synthesis in standalone mode or understand more about its advanced features, see the *Precision RTL Synthesis Users Manual*, *Precision RTL Synthesis Style Guide*, and *Precision Synthesis Reference Manual* in the online help.

Learning Objectives

When you have completed this tutorial, you should be able to do the following:

- Create a new EDIF project in ispLEVER and target a device.
- Start Precision RTL Synthesis from within ispLEVER, synthesize your Verilog HDL design, and generate an EDIF netlist file.
- Import the EDIF file into the ispLEVER system, fit the design, generate a JEDEC file, and view the Fitter report.
- Perform static timing analysis using the Performance Analyst and view the results.

Time to Complete This Tutorial

The time to complete this tutorial is about 20 minutes.

System Requirements

One of the following software configurations is required to complete the tutorial:

- IspLEVER Starter
- ispLEVER

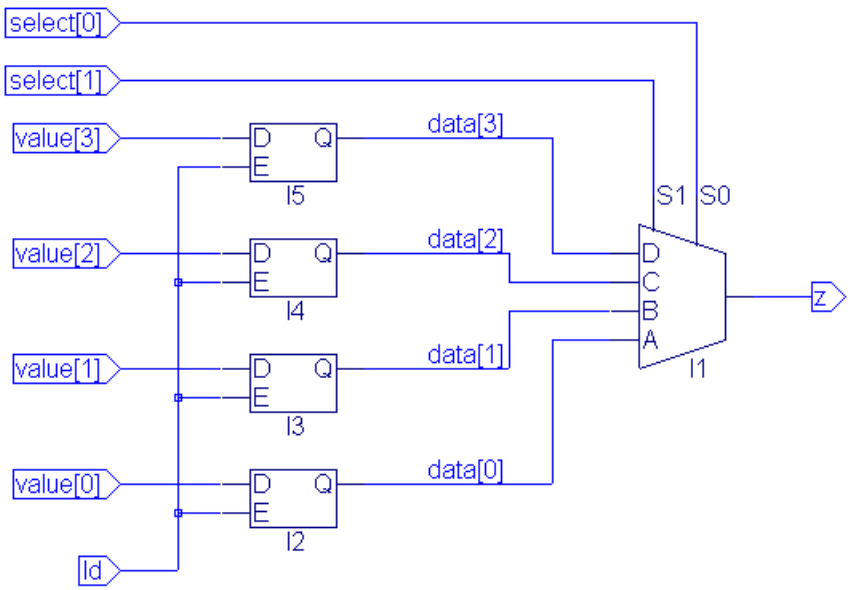
Accessing Online Help

You can find online help information on any tool included in the tutorial at any time by pressing the F1 key.

About the Tutorial Design

The tutorial design consists of a simple 4 to 1 multiplexer, as shown in Figure 1:

Figure 1: Tutorial Design



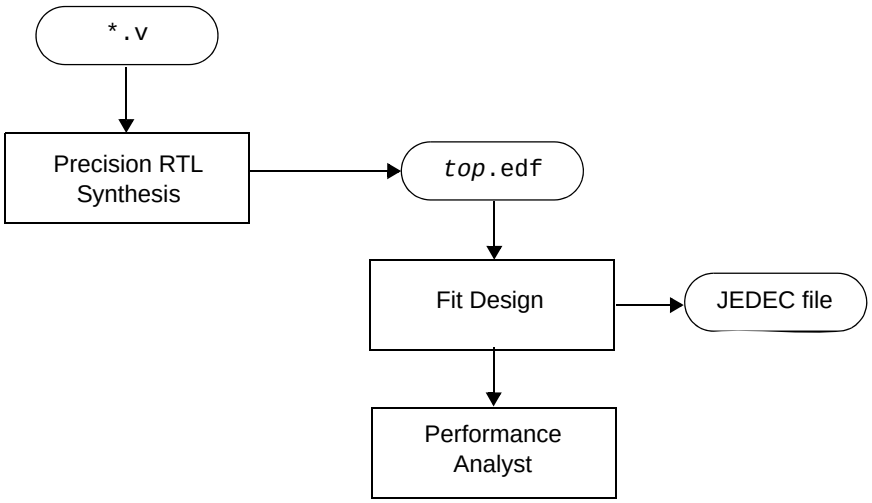
This tutorial first directs you to create an EDIF project in the Project Navigator, then select the target device in which the design will be implemented. It assumes that functional simulation has already been performed. Next, you

start Precision RTL Synthesis and open a new Precision project. After you import the Verilog source files and set the implementation options, the tool synthesizes the design into the target device and generates an EDIF netlist. You then import the EDIF netlist into the Project Navigator project and fit the design. Finally, you perform a static timing analysis and examine the results.

About the Tutorial Data Flow

Figure 2 illustrates the design flow that the tutorial takes.You may find it helpful to refer to this diagram as you move through the tutorial tasks.

Figure 2: Tutorial Design Flow



Task 1: Create a New Project

To begin a new project in the Project Navigator, you must create a project directory. Then give the project file a name (`.syn`) and declare the project type (EDIF).

The ispLEVER software saves an initial design file with the `.syn` file extension in the directory that you specify. All project files are copied to or created in this directory. The project type specifies that all design sources will be of this type.

To create a new project:

1. Start the ispLEVER system, if it is not already running.
2. In the Project Navigator, choose **File > New Project** to open the Project Wizard dialog box.

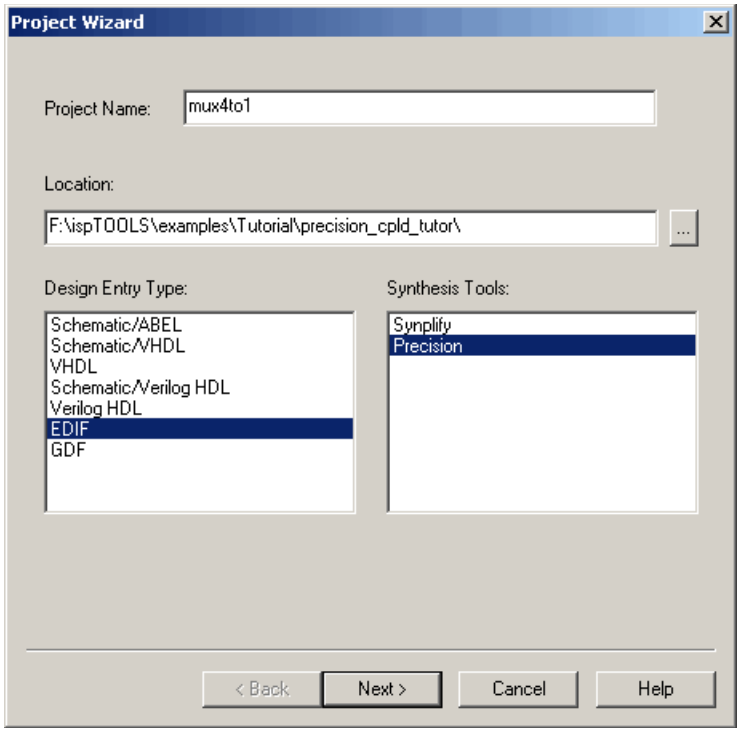
3. In the dialog box, do the following:
   a. In the Project Name box, type **`mux4to1`**.
   b. In the Location box, change to the following directory: `<install_path>\examples\tutorial\` **`precision_cpld_tutor`**.

Note

If you want to preserve the original tutorial design files, save the `precision_cpld_tutor` directory to another location on your computer before proceeding.

   c. In the Design Entry Type box, select **EDIF**.
   d. In the Synthesis Tools box, select **Precision**, shown in Figure 3.
   e. Click **Next** to open the Project Wizard – Select Device dialog box.

Figure 3: Project Wizard Dialog Box



Task 2: Target a Device

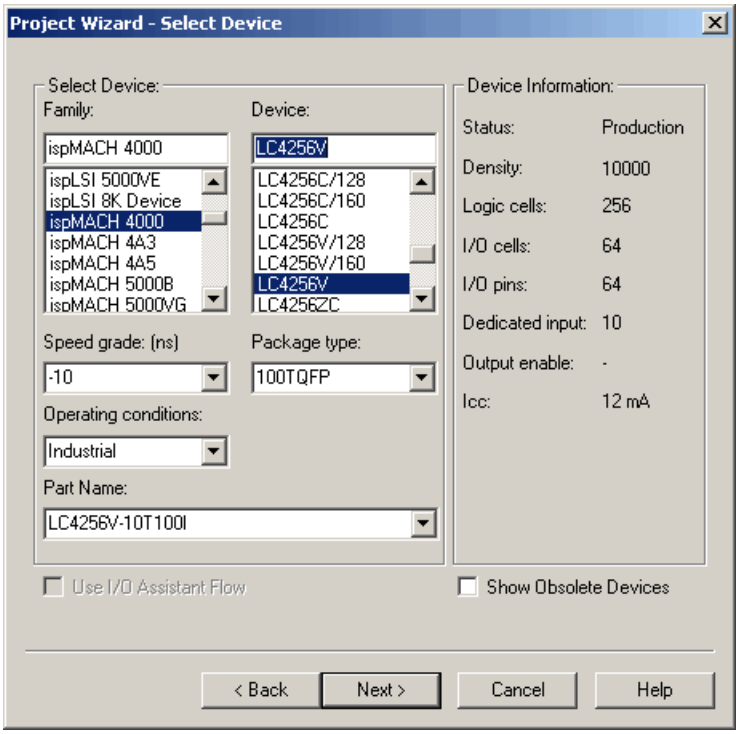
The Project Wizard enables you to target a design to a specific Lattice device.

In the Project Navigator Sources in Project window, the device icon appears next to the target device for the project. You can double-click the icon to change the target device at any time during the design process.

To view the list of available devices and select the target device:

1. In the Project Wizard – Select Device dialog box, do the following:
   a. In the Family box, select **ispMACH 4000**.
   b. In the Device box, select **LC4256V**.
   c. Accept the default settings for the rest of the boxes, as shown in Figure 4.

Figure 4: Project Wizard - Select Device Dialog Box



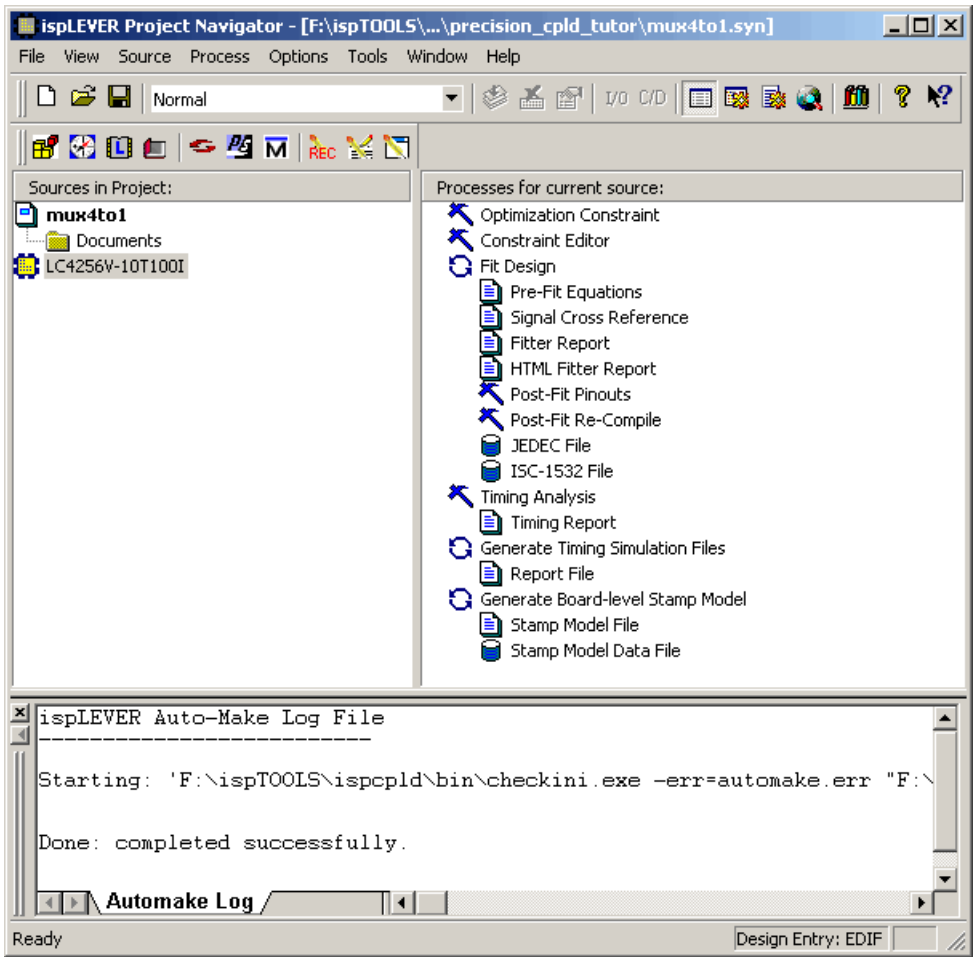
   d. Click **Next** to open the Project Wizard – Add Source dialog box.
2. In the Project Wizard – Add Source dialog box, click **Next**, then click **Finish**.

Your Project Navigator should look like Figure 5.

Note

Click on the part name to see the contents of the Processes for Current Source window.

Figure 5: Project Navigator Window Showing New Project



Task 3: Create a Precision RTL Synthesis Project

For HDL designs, the ispLEVER software provides two synthesis tools that are integrated into the Project Navigator environment: Synplicity's Synplify and Mentor Graphics' Precision RTL Synthesis. You can synthesize your Verilog or VHDL design as a standalone process by choosing the synthesis tool from the Project Navigator or Lattice Semiconductor program group in your Start menu, or you can synthesize automatically and seamlessly within the Project Navigator.

Precision RTL Synthesis is a logic synthesis tool that starts with a high-level design written in the Verilog or VHDL hardware description language (HDLs).

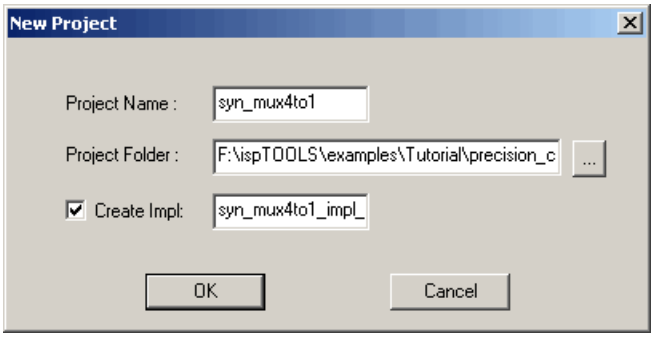
Then it converts the HDL description into small, high-performance design netlists that are optimized for Lattice devices.

To start Precision RTL Synthesis:

1. In the Project Navigator, choose **Tools > Precision Synthesis** to open the Precision RTL Synthesis tool.
2. Click **OK** to close the Tip of the Day.
3. Choose **File > New Project**, or click on the **New Project** icon in the Project pane to open the New Project dialog box, shown in Figure 6.
4. In the dialog box, do the following:
   a. In the Project Name box, type **`syn_mux4to1`**.
   b. In the Project Folder box, make sure the following directory is shown:

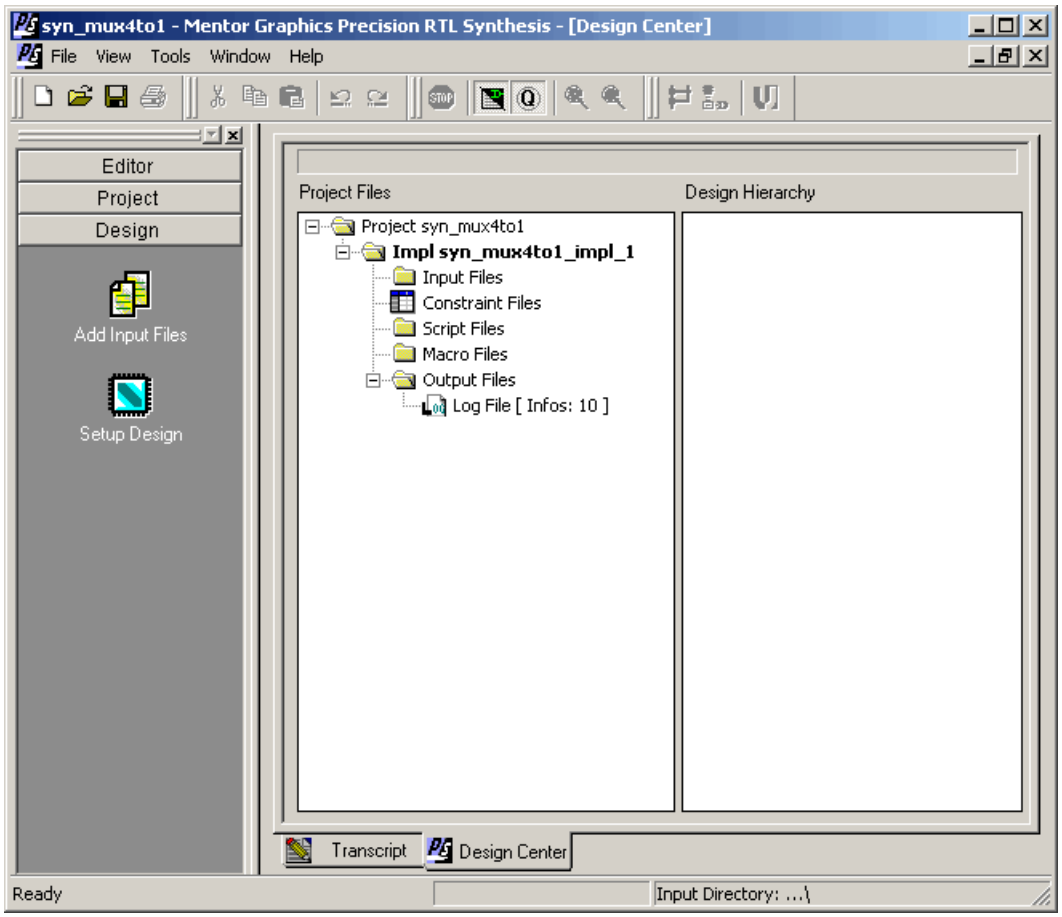
      `<install_path>\examples\Tutorial\`
      `precision_cpld_tutor`
   c. Click **OK**.

Figure 6: New Project Dialog Box



The Precision RTL Synthesis window looks like that shown in Figure 7.

Figure 7: Precision RTL Synthesis Window Showing syn_mux4to1 Project



Task 4: Add the Verilog HDL Source File

Next, you will add a Verilog HDL source file to the project.

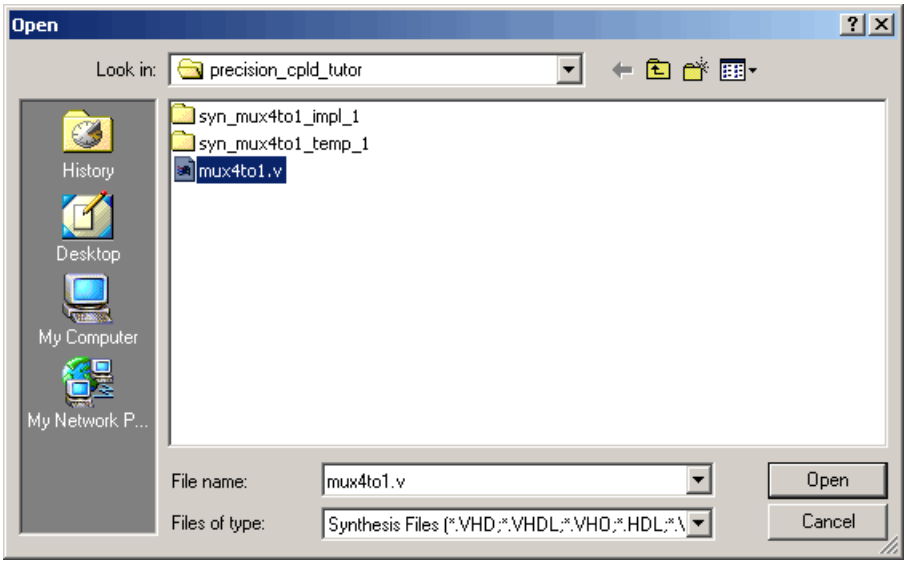
To add a source file to the project:

1. Click the **Add Input Files** icon in the Design pane.

   You should see one Verilog HDL (`.v`) file called `mux4to1.v` listed in the Open dialog box, as shown in Figure 8.

2. Select the `mux4to1.v` file, then click **Open**.

Figure 8: Open Dialog Box



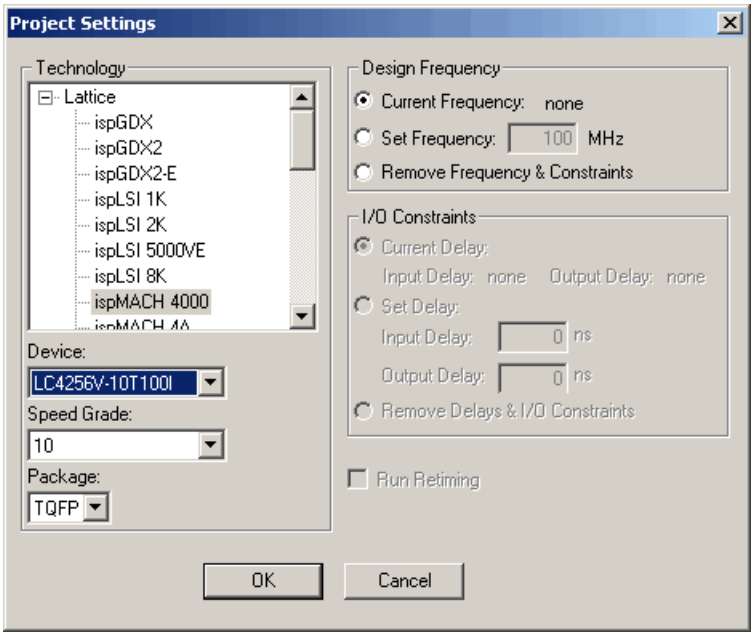
Task 5: Set Implementation Options

Now you will set the target device and implementation options for logic synthesis.

To set the implementation options:

1. In the Design pane, click the **Setup Design** icon.
2. In the Project Settings dialog box, shown in Figure 9, click the plus sign (+) next to Lattice, and select **ispMACH 4000**.
3. In the Device box, select **LC4256V-10T100I**.
4. Click **OK**.

Figure 9: Project Settings Dialog Box



Task 6: Compile the Design

In this task, you will compile the Verilog HDL design.

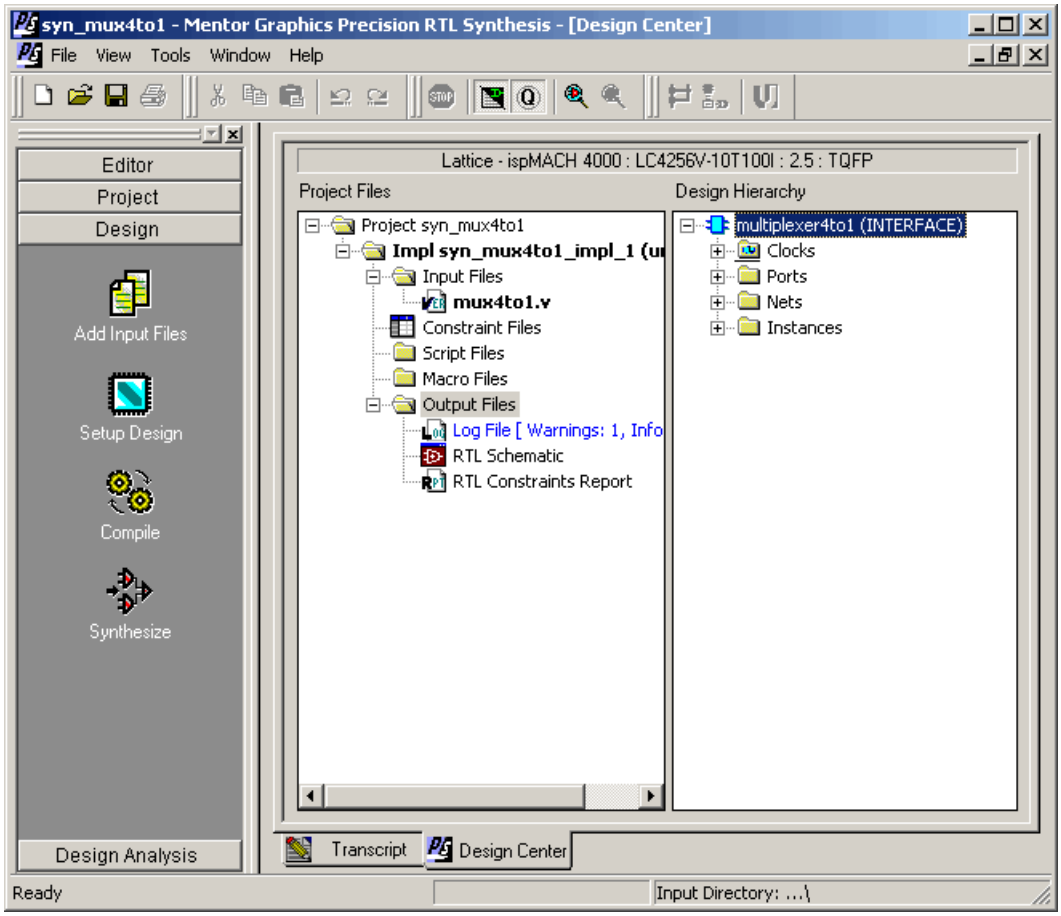
To compile the design:

1. In the Design pane, click **Compile**.

   The Precision RTL Synthesis software reads the input file and compiles the design using the source file, targeted device, and other settings.

   The Precision RTL Synthesis window now lists the clocks, ports, nets, and instances in the design, as shown in Figure 10.

Figure 10: Precision RTL Synthesis Window After Compilation



Select the Design Analysis pane to examine key reports and graphical views of the design, including the RTL Schematic (View RTL Schematic).

Task 7: Synthesize the Design

Now you will synthesize the Verilog HDL design.

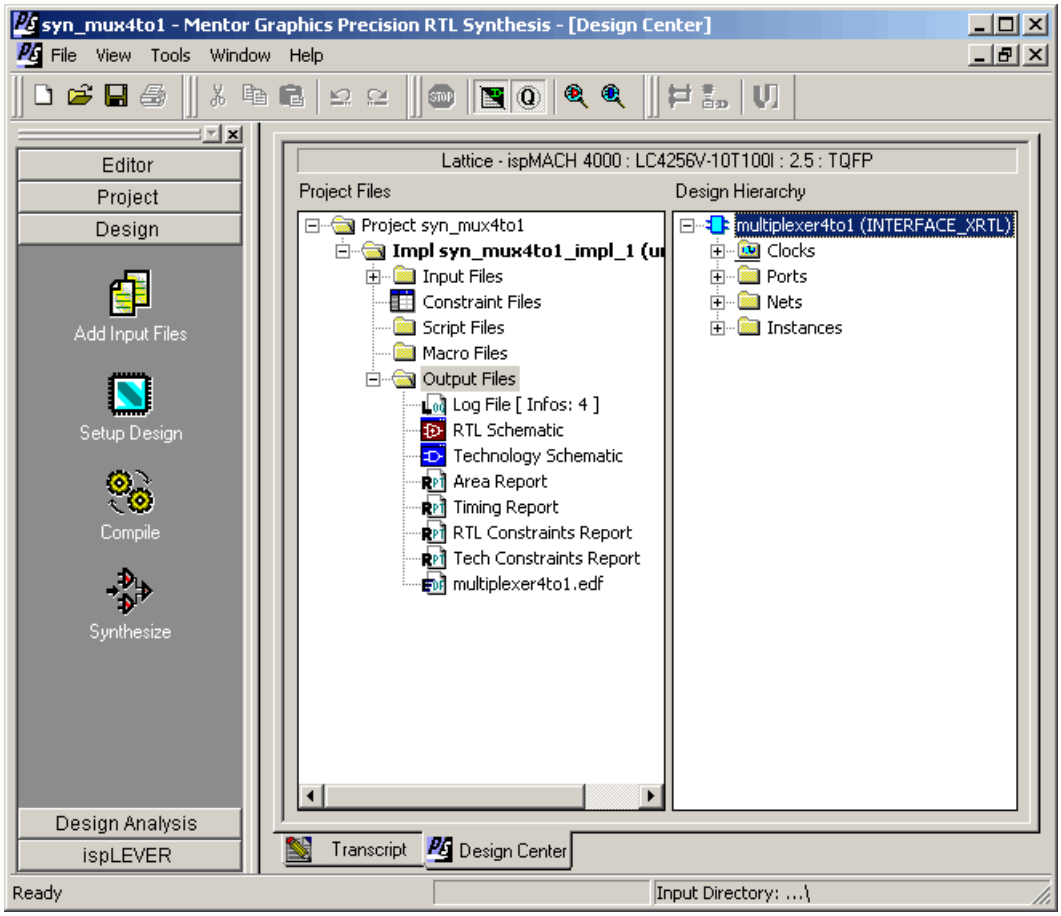
To synthesize the design:

1. Click the **Synthesize** icon.

   Precision RTL Synthesis synthesizes the Verilog HDL design and creates an EDIF file, as well as several other files, and displays them in the window, as shown in Figure 11. If you like, you can double-click the different files and view them. When you finish, close the files.

   Select the Design Analysis pane to examine key reports for area (Report Area) and timing (Report Timing) and graphical views of the design, including the post-synthesis schematic (View Schematic) and critical path (View Critical Path).

Figure 11: Precision RTL Synthesis Window After Synthesis



2. Select **File > Save Project**.
3. Choose **File > Exit** to close Precision RTL Synthesis. At the `Are you sure you want to exit?` prompt, click **Yes**.

Task 8: Import the EDIF File into Your Project

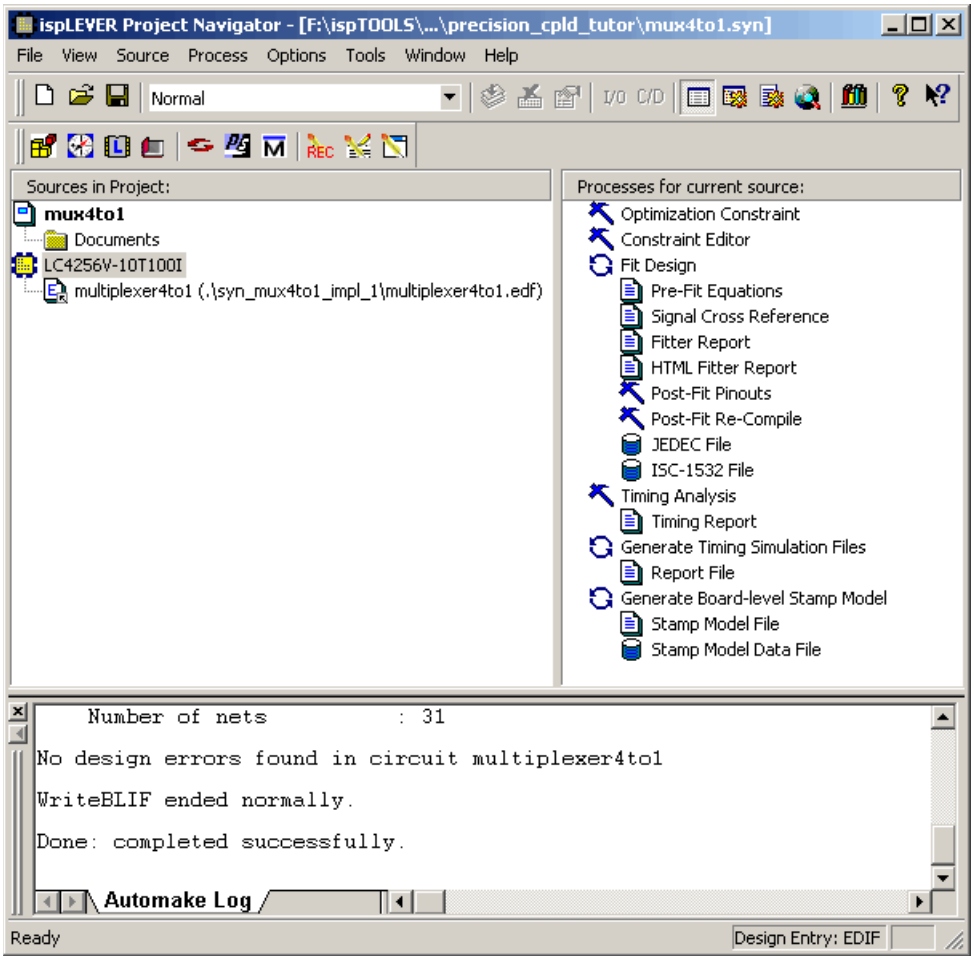
Now you are ready to import the EDIF file into ispLEVER project. You can import EDIF 2 0 0 netlists from third-party synthesis tools, such as Synplify or Precision RTL Synthesis, into ispLEVER.

To import an EDIF netlist into your project:

1. In the ispLEVER Project Navigator, choose **Source > Import**.
2. Go to **`syn_mux4to1_impl_1`** folder within your project, select **`multiplexer4to1.edf`**, and click **Open**.

   The software adds the selected EDIF file (`multiplexer4to1.edf`) to the project sources, as shown in Figure 12.

Figure 12: Project Navigator Showing Added EDIF File



Note

After you import an EDIF file into the ispLEVER project, it is always linked to the Project Navigator. Therefore, if you make changes and recompile your HDL file to create a new EDIF file, your project is automatically updated as well.

Task 9: Fit the Design and View the Report

The ispLEVER software has a single user interface with all options preset to deliver the highest possible push-button performance for most devices. When you double-click a process, all the processes prior to that process run automatically. Therefore, all you have to do is double-click the final process. However, here you will run one process at a time and view the results as you go.

At the end of a successful fitter run, the ispLEVER software generates a JEDEC file, as well as a fitter report so that you can see how the ispLEVER software has utilized and routed the part.

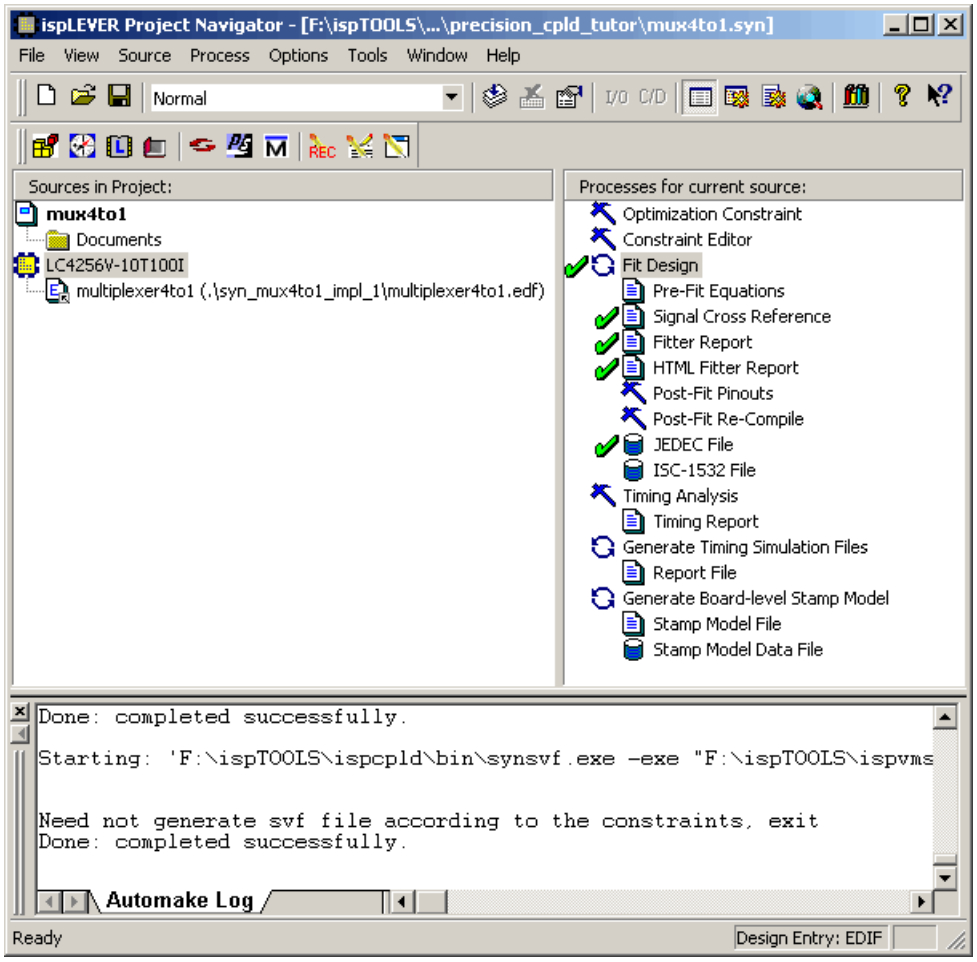
To fit the design and view the report:

1. With the target device selected in the Sources in Project window, double-click **Fit Design** in the Processes for Current Source window to run the Fit Design process.

   The ispLEVER software successfully fits the design in the specified device and generates a JEDEC file. The results are shown in Figure 13.

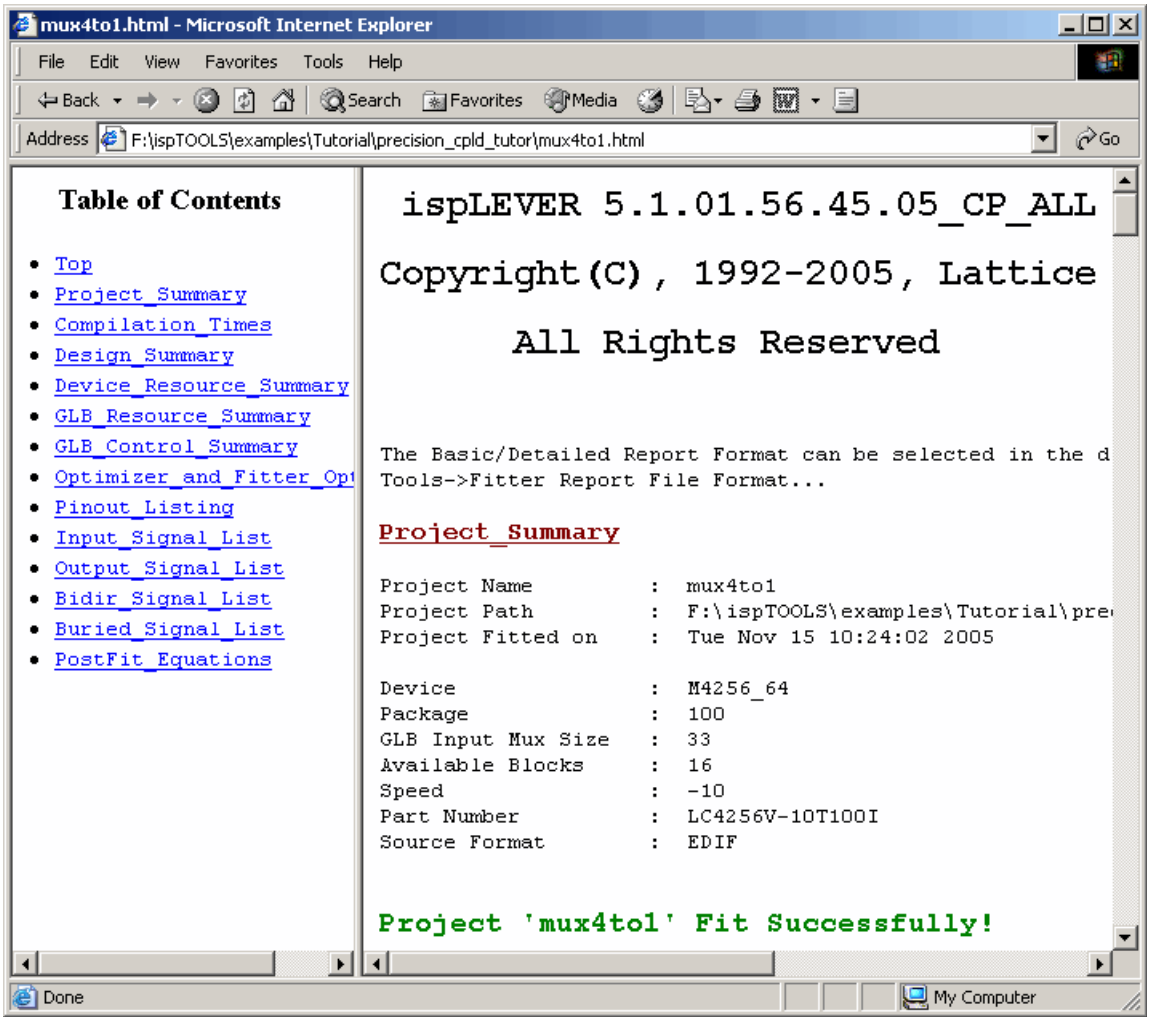
   Optional: If you like, you can right-click on the JEDEC File process and select **View** to create and look at the contents of the JEDEC file. Close the file when you are through.

Figure 13: Project Navigator Showing Fitted Design



2. Double-click on the **HTML Fitter Report** process to open the report in your browser, as shown in Figure 14. View the contents and then close the report.

Figure 14: HTML Fitter Report



Task 10: Perform Static Timing Analysis

Static timing analysis is the process of verifying circuit timing by totaling the propagation delays along paths between clocked or combinational elements in a circuit. The analysis can determine and report timing data such as the critical path, setup and hold-time requirements, and the maximum frequency.

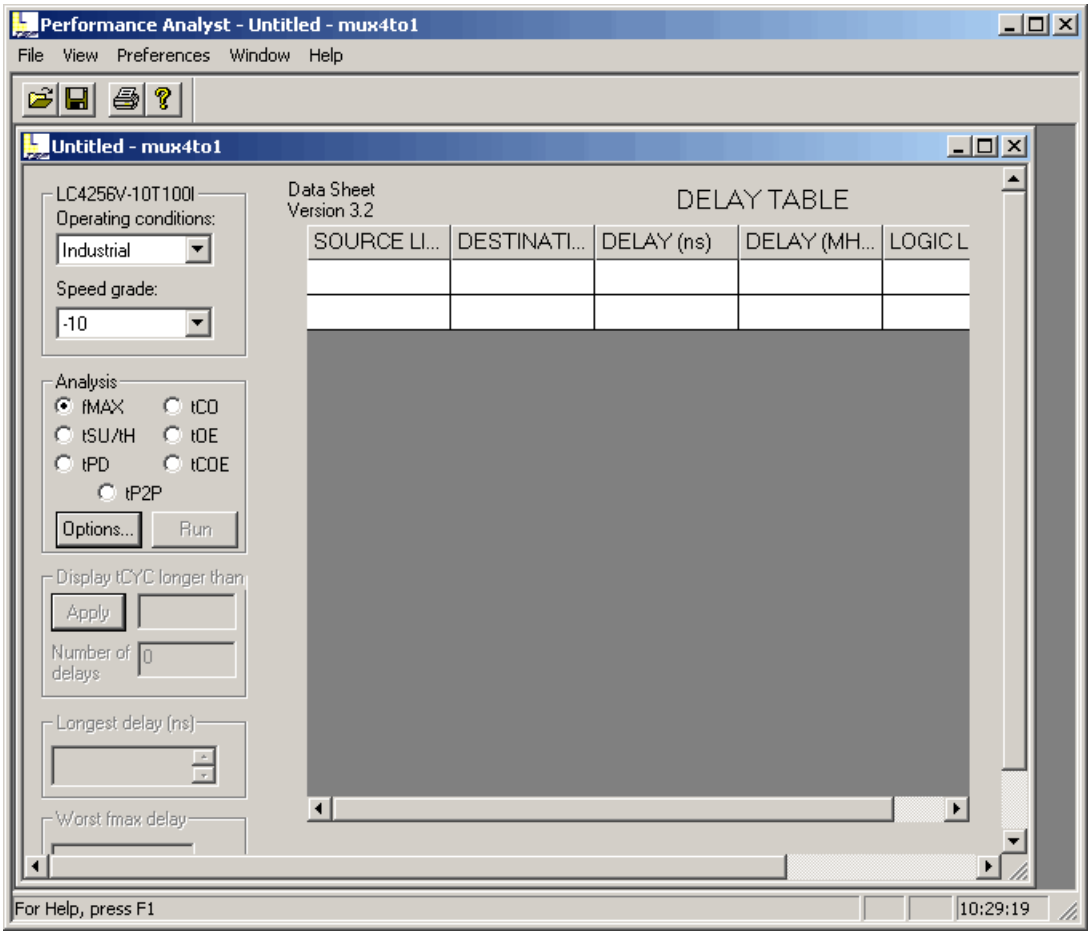
The Performance Analyst traces each logical path in the design and calculates the path delays using the device's timing model and worst-case AC specifications supplied in the device data sheet.

The timing analysis results are displayed in a graphical spreadsheet with source signals displayed on the vertical axis and destination signals displayed on the horizontal axis. The worst-case delay value is displayed in a spreadsheet cell if there is at least one delay path between the source and destination. To more easily identify performance bottlenecks, you can double-click a cell to view the path delay details.

To perform timing analysis:

1. In the Project Navigator Sources in Project window, select the target device.
2. In the Processes for Current Source window, double-click the **Timing Analysis** process to run the timing analysis and open the Performance Analyst, shown in Figure 15.

Figure 15: Performance Analyst Window



The Performance Analyst performs seven distinct analysis types: fMAX, tSU/tH, tPD, tCO, tOE, tCOE, and tP2P. The first type, fMAX, is an internal register-to-register delay analysis. fMAX measures the maximum clock operating frequency, limited by worst-case register-to-register delay. The tP2P type is the path between any two user-specified pins. The remaining five types are external pin-to-pin delay analysis. Timing threshold filters, source and destination filters, and path filters can be used to independently fine-tune each analysis.

3. Under Analysis, select **tCO** and then click **Run**.

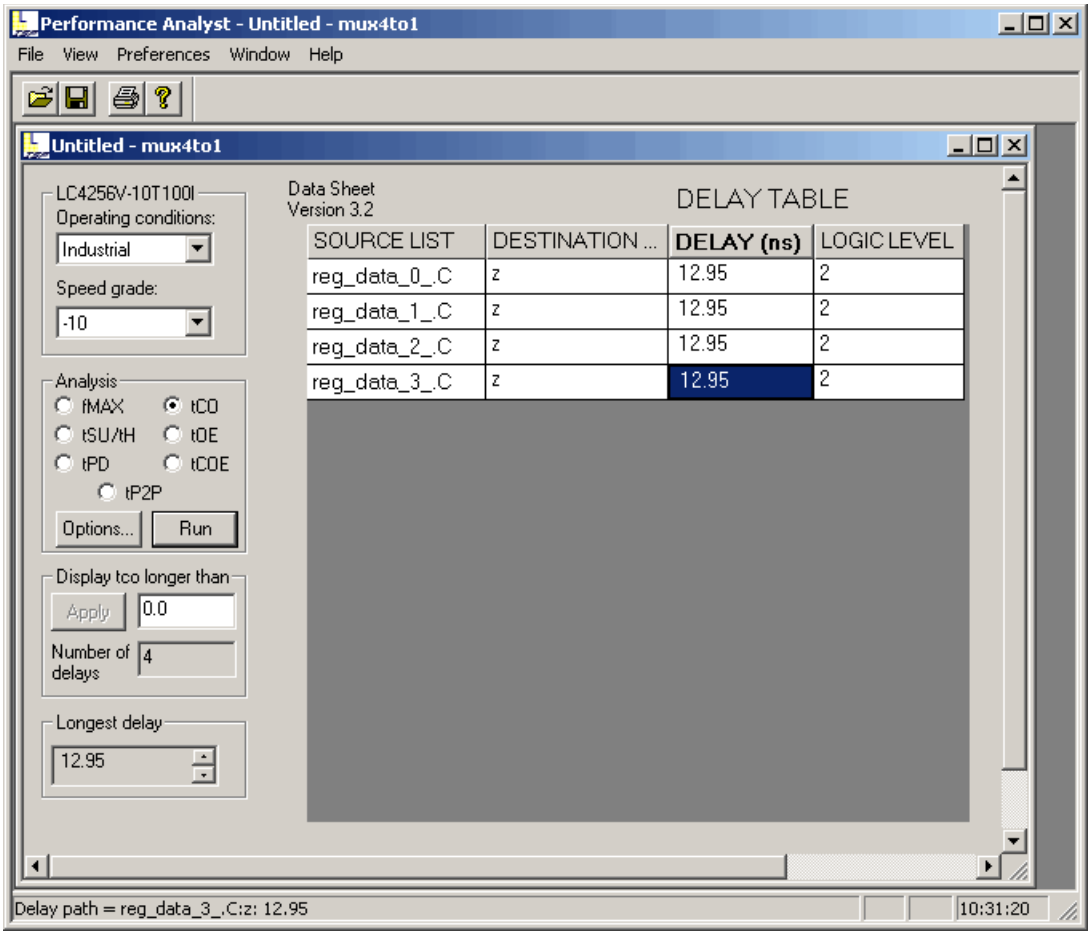
   The tCO path trace analysis reports clock-to-out delay starting from the primary input, going through the clock of flip-flops or gate of latches, and

ending at the primary output. In this case it is 12.95 ns. Figure 16 shows the results of the analysis.

Note

Your timing numbers may differ slightly.

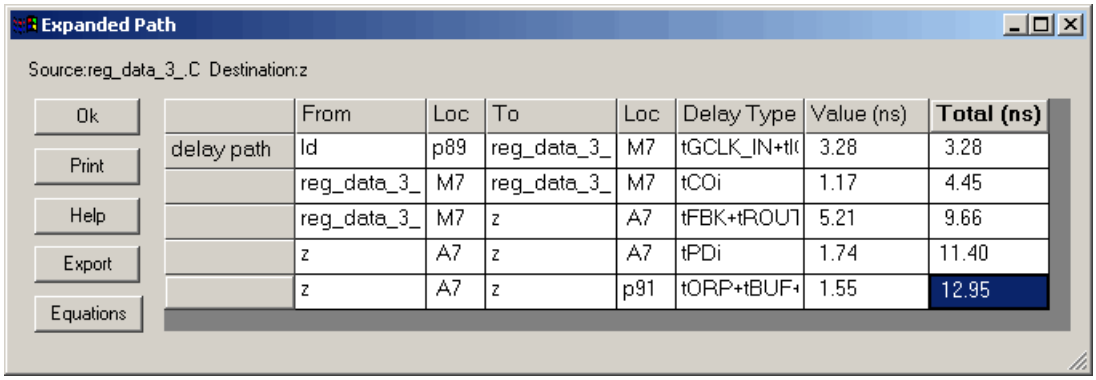
Figure 16: Performance Analyst Window Showing Clock-to-Out Delay



4. Click the highlighted cell (**12.95**) in the spreadsheet window to open the Expanded Path dialog box, shown in Figure 17.

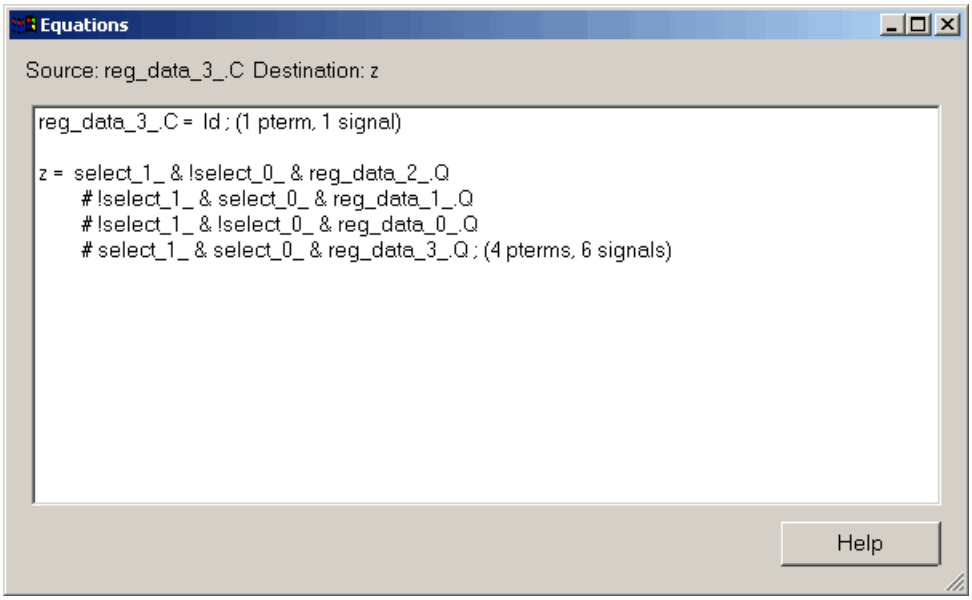
   This dialog box enables you to analyze the individual timing components used to calculate the timing path. It shows a source pin (From) and a destination pin (To). It also shows the delay type, the delay of that path (value ns), and the cumulative delay of all the signals.

Figure 17: Expanded Path Dialog Box.



5. Click **Equations** to open the Equations dialog box, shown in Figure 18, which shows the functional relationship between the selected source and the destination.

Figure 18: Equations Dialog Box



6. Close the Performance Analyst without saving.
7. Close ispLEVER without saving.

Summary

You have completed the HDL Design with Precision RTL Synthesis: CPLD Flow tutorial. In this tutorial you have learned how to do the following:

- Create a new EDIF project in the ispLEVER system and target a device.
- Start Precision RTL Synthesis from within ispLEVER, synthesize your Verilog HDL design, and generate an EDIF netlist file.
- Import the EDIF file into ispLEVER.

- Fit the design, generate a JEDEC file, and view the Fitter report.
- Run static timing analysis using the Performance Analyst and view the results.

Glossary

Following are the terms and concepts that you should understand to use this tutorial effectively.

EDIF. EDIF (Electronic Design Interchange Format) is a format used to exchange design data between different electronic computer-aided design systems. It is designed to be written and read by computer programs that are constituent parts of EDA systems or tools. Its syntax has been designed for easy machine parsing and is similar to LISP. The ispLEVER software supports EDIF Version 2 0 0.

HDL. An HDL is a hardware description language, which describes the structure and function of integrated circuits.

static timing analysis. Static timing analysis is the process of verifying circuit timing by totaling the propagation delays along paths between clocked or combinational elements in a circuit. The analysis can determine and report timing data such as the critical path, setup and hold-time requirements, and the maximum frequency.

synthesis. Synthesis is the process of translating a high-level design (RTL) description consisting of state machines, truth tables, and/or Boolean equations into a process-specific gate-level logic implementation.

Verilog. Verilog is a language for describing the structure and function of integrated circuits.

VHDL. VHDL (or VHSIC (Very High-Speed Integrated Circuits) Hardware Description Language) is a language for describing the structure and function of integrated circuits.

Recommended Reference Materials

You can find additional information on the subjects covered by this tutorial from the following recommended sources:

- Precision RTL Synthesis Users Manual
- Precision RTL Synthesis Style Guide
- Precision Synthesis Reference Manual
- Lattice Semiconductor ispLEVER online help
- Data sheets, technical notes, and other information on CPLDs on the Lattice Web site at http://www.latticesemi.com