

LatticeMico8 Tutorial



June 2012

Copyright

Copyright © 2012 Lattice Semiconductor Corporation.

This document may not, in whole or part, be copied, photocopied, reproduced, translated, or reduced to any electronic medium or machine-readable form without prior written consent from Lattice Semiconductor Corporation.

Trademarks

Lattice Semiconductor Corporation, L Lattice Semiconductor Corporation (logo), L (stylized), L (design), Lattice (design), LSC, CleanClock, Custom Mobile Device, DiePlus, E2CMOS, Extreme Performance, FlashBAK, FlexiClock, flexiFLASH, flexiMAC, flexiPCS, FreedomChip, GAL, GDX, Generic Array Logic, HDL Explorer, iCE Dice, iCE40, iCE65, iCEcable, iCEchip, iCEcube, iCEcube2, iCEman, iCEprog, iCEsab, iCEsocket, IPexpress, ISP, ispATE, ispClock, ispDOWNLOAD, ispGAL, ispGDS, ispGDX, ispGDX2, ispGDXV, ispGENERATOR, ispJTAG, ispLEVER, ispLeverCORE, ispLSI, ispMACH, ispPAC, ispTRACY, ispTURBO, ispVIRTUAL MACHINE, ispVM, ispXP, ispXPGA, ispXPLD, Lattice Diamond, LatticeCORE, LatticeEC, LatticeECP, LatticeECP-DSP, LatticeECP2, LatticeECP2M, LatticeECP3, LatticeECP4, LatticeMico, LatticeMico8, LatticeMico32, LatticeSC, LatticeSCM, LatticeXP, LatticeXP2, MACH, MachXO, MachXO2, MACO, mobileFPGA, ORCA, PAC, PAC-Designer, PAL, Performance Analyst, Platform Manager, ProcessorPM, PURESPEED, Reveal, SiliconBlue, Silicon Forest, Speedlocked, Speed Locking, SuperBIG, SuperCOOL, SuperFAST, SuperWIDE, sysCLOCK, sysCONFIG, sysDSP, sysHSI, sysI/O, sysMEM, The Simple Machine for Complex Design, TraceID, TransFR, UltraMOS, and specific product designations are either registered trademarks or trademarks of Lattice Semiconductor Corporation or its subsidiaries in the United States and/or other countries. ISP, Bringing the Best Together, and More of the Best are service marks of Lattice Semiconductor Corporation.

Other product names used in this publication are for identification purposes only and may be trademarks of their respective companies.

Disclaimers

NO WARRANTIES: THE INFORMATION PROVIDED IN THIS DOCUMENT IS “AS IS” WITHOUT ANY EXPRESS OR IMPLIED WARRANTY OF ANY KIND INCLUDING WARRANTIES OF ACCURACY, COMPLETENESS, MERCHANTABILITY, NONINFRINGEMENT OF INTELLECTUAL PROPERTY, OR FITNESS FOR ANY PARTICULAR PURPOSE. IN NO EVENT WILL LATTICE SEMICONDUCTOR CORPORATION (LSC) OR ITS SUPPLIERS BE LIABLE FOR ANY DAMAGES WHATSOEVER (WHETHER DIRECT, INDIRECT, SPECIAL, INCIDENTAL, OR CONSEQUENTIAL, INCLUDING, WITHOUT LIMITATION, DAMAGES FOR LOSS OF PROFITS, BUSINESS INTERRUPTION, OR LOSS OF INFORMATION) ARISING OUT OF THE USE OF OR INABILITY TO USE THE INFORMATION PROVIDED IN THIS DOCUMENT, EVEN IF LSC HAS BEEN ADVISED OF THE POSSIBILITY OF SUCH DAMAGES. BECAUSE SOME JURISDICTIONS PROHIBIT THE EXCLUSION OR LIMITATION OF CERTAIN LIABILITY, SOME OF THE ABOVE LIMITATIONS MAY NOT APPLY TO YOU.

LSC may make changes to these materials, specifications, or information, or to the products described herein, at any time without notice. LSC makes no commitment to update this documentation. LSC reserves the right to discontinue any product or service without notice and assumes no obligation to correct any errors contained herein or to advise any user of this document of any correction if such be made. LSC recommends its customers obtain the latest version of the relevant information to establish, before ordering, that the information being relied upon is current.

Type Conventions Used in This Document

Convention	Meaning or Use
Bold	Items in the user interface that you select or click. Text that you type into the user interface.
<i><Italic></i>	Variables in commands, code syntax, and path names.
Ctrl+L	Press the two keys at the same time.
<code>Courier</code>	Code examples. Messages, reports, and prompts from the software.
<code>...</code>	Omitted material in a line of code.
<code>.</code> <code>.</code> <code>.</code>	Omitted lines in code and report examples.
[]	Optional items in syntax descriptions. In bus specifications, the brackets are required.
()	Grouped items in syntax descriptions.
{ }	Repeatable items in syntax descriptions.
	A choice between items in syntax descriptions.

Contents

LatticeMico8 Tutorial	1
Introduction	1
Learning Objectives	2
Time to Complete This Tutorial	2
System Requirements	2
Accessing Online Help	3
About the Tutorial Design	3
Tutorial Data Flow	4
Task 1: Create a New Lattice Diamond Project	7
Task 2: Create LatticeMico8 Platform	11
Create a New MSB Platform	11
Add the Microcontroller Core	16
Add the Peripheral Components	19
Specify the Connections Between Master and Slave Ports	21
Assign Component Addresses	24
Assign Interrupt Request Priorities	25
Perform a Design Rule Check	26
Generate the LatticeMico8 Platform	26
Create a User Top-Level Module	28
Task 3: Create Software Application Code	29
Create a New C/C++ SPE Project	31
Build the Project	33
Task 4: Deploy Software Application Code to LatticeMico8	34
Task 5: Generate Platform Bitstream	38
Import the MSB Output File	38
Connect the Microcontroller to the FPGA Pins	39
Perform Functional Simulation	40
Perform Timing Simulation	41
Generate the Bitstream	41
Task 6: Download Platform Bitstream to FPGA	41
Summary	44

LatticeMico8 Tutorial

Introduction

This tutorial steps you through the basic process involved in using the LatticeMico System software to implement an instance of the LatticeMico8 microcontroller and other attached components in a Lattice Semiconductor device for the MachXO2 Control Board. LatticeMico System encompasses three tools: the Mico System Builder (MSB), the C/C++ Software Project Environment (C/C++ SPE), and the Debugger. The MSB and the C/C++ SPE together enable you to build an embedded microcontroller system on a single FPGA device and to write and debug the software that drives it. Such a microcontroller lowers cost by saving board space and increases performance by reducing the number of external wires.

The LatticeMico System interface is based on the Eclipse environment, which is an open-source development and application framework for building software.

Although you can install LatticeMico System as a stand-alone tool, this tutorial assumes that you have installed Lattice Diamond before installing LatticeMico System. After you have created a project in Lattice Diamond, the tutorial shows you how to use MSB to choose a Lattice Semiconductor 8-bit microcontroller, attach components to it, and generate a top-level design, including the microcontroller and the chosen components. Next you will use Lattice Diamond to synthesize, map, place, and route the design and generate a bitstream for it. You will then download this bitstream to the FPGA on the board. The tutorial then changes to the Lattice Software Project Environment (C/C++ SPE) and shows how to use C/C++ SPE to write and compile the software application code that exercises the microcontroller and components. Finally, it shows how to deploy the code to the microcontroller.

This tutorial is intended for a new or infrequent user of the LatticeMico System software and covers only the basic aspects of it. For more detailed information

on the LatticeMico System software, see the sources listed in “Recommended References”.

Learning Objectives

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

- ▶ Use MSB to configure a Lattice Semiconductor 8-bit microcontroller for your design, select the desired components, and connect the selected components to the microcontroller with a shared-bus arbitration scheme, which is the default.
- ▶ Use The Lattice Software Project Environment to create the C/C++ software application code that drives the microcontroller and components.
- ▶ Compile and deploy the software application code to the microcontroller.
- ▶ Import the Verilog or Verilog/VHDL files generated by MSB into the Lattice Diamond development tools.
- ▶ Import an .lpf file containing the pinout.
- ▶ Synthesize, map, place, and route the design.
- ▶ Generate a bitstream of the microcontroller and download it to an FPGA on the board.

Time to Complete This Tutorial

The time to complete this tutorial is about two hours.

System Requirements

You can run this tutorial on Windows or Linux.

Windows

If you will be running this tutorial on Windows on a PC, your system must meet the following minimum system requirements:

- ▶ Pentium IV PC running at 2 GHz or faster
- ▶ Microsoft Windows XP® Professional, or Windows Vista®, or Windows 7.
- ▶ Available USB 2.0 port

The following software is required to complete the tutorial:

- ▶ Lattice Diamond version 1.3 or later

- ▶ LatticeMico System version 1.3 or later

See the Lattice Diamond Installation Notice for Windows for the current release for information on installing software on the Windows platform.

Linux

If you will be running this tutorial on Linux on a PC, your system must meet the following minimum system requirements:

- ▶ Red Hat Enterprise Linux operating system Version 4.0 or 5.0
- ▶ Lattice Diamond version 1.3 or later
- ▶ Linux system with USB port

See the Lattice Diamond Installation Notice for Linux for the current release for information on installing software on the Linux platform.

Hardware

This tutorial requires the following hardware:

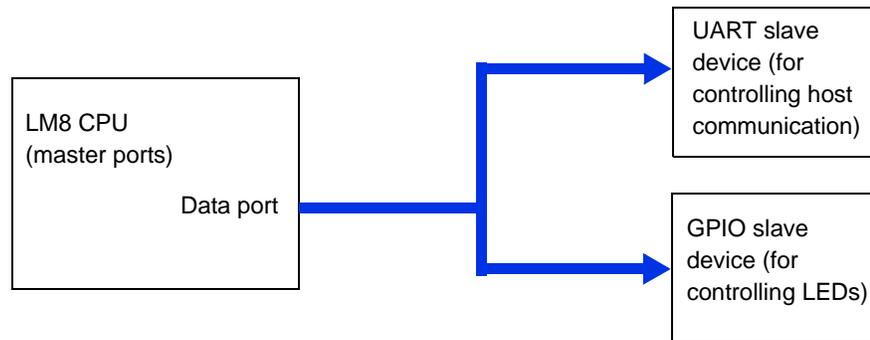
- ▶ A MachXO2 Control Board for Lattice MachXO2 FPGA.
- ▶ USB cable
- ▶ AC adapter cord

Accessing Online Help

You can access the online Help for MSB, C/C++ SPE, the Debugger, or Eclipse Workbench by choosing Help > Help Contents in the LatticeMico System graphical user interface.

About the Tutorial Design

This tutorial uses a MachXO2 device, and all references are based on the MachXO2 device. The tutorial design consists of the LatticeMico8 embedded microcontroller, a GPIO, and a UART. After you add these components, you will specify the connections between the master and slave ports on these components, as shown in Figure 1.

Figure 1: Desired Connections Between Master and Slave Ports

In this design, the instruction port and the data port of the LatticeMico8 microcontroller are the master ports. All other ports are slave ports. The LatticeMico8 data port accesses the LatticeMico GPIO and the LatticeMico UART.

Tutorial Data Flow

You will perform the following major steps to create an embedded microcontroller system:

1. Create a new project in Lattice Diamond.
2. Create a microcontroller platform for the LatticeMico8 microcontroller in MSB with a shared-bus arbitration scheme, which is the default.
3. Write the software application code for the microcontroller platform in C/C++ SPE.
4. Deploy the software application code to the microcontroller's internal PROM and Scratchpad.
5. Generate a bitstream of the microcontroller platform in Diamond.
6. Download the hardware bitstream to the FPGA using Diamond Programmer.

This tutorial supports both Verilog and mixed Verilog/VHDL design flows in Diamond for Windows and Linux users. The Windows Verilog design flow for using LatticeMico System to create an embedded microcontroller and the software code for it is shown in Figure 2. The Windows mixed Verilog/VHDL design flow is shown in Figure 3. The difference between the two methods is that mixed verilog/VHDL designs have a VHDL wrapper as an output from MSB. The VHDL wrapper is an input to Synthesis and Functional Simulation in the Diamond flow.

Functional and timing simulation are performed in third-party and simulation tools outside of Diamond.

Figure 2: Design Flow for Windows Verilog Users

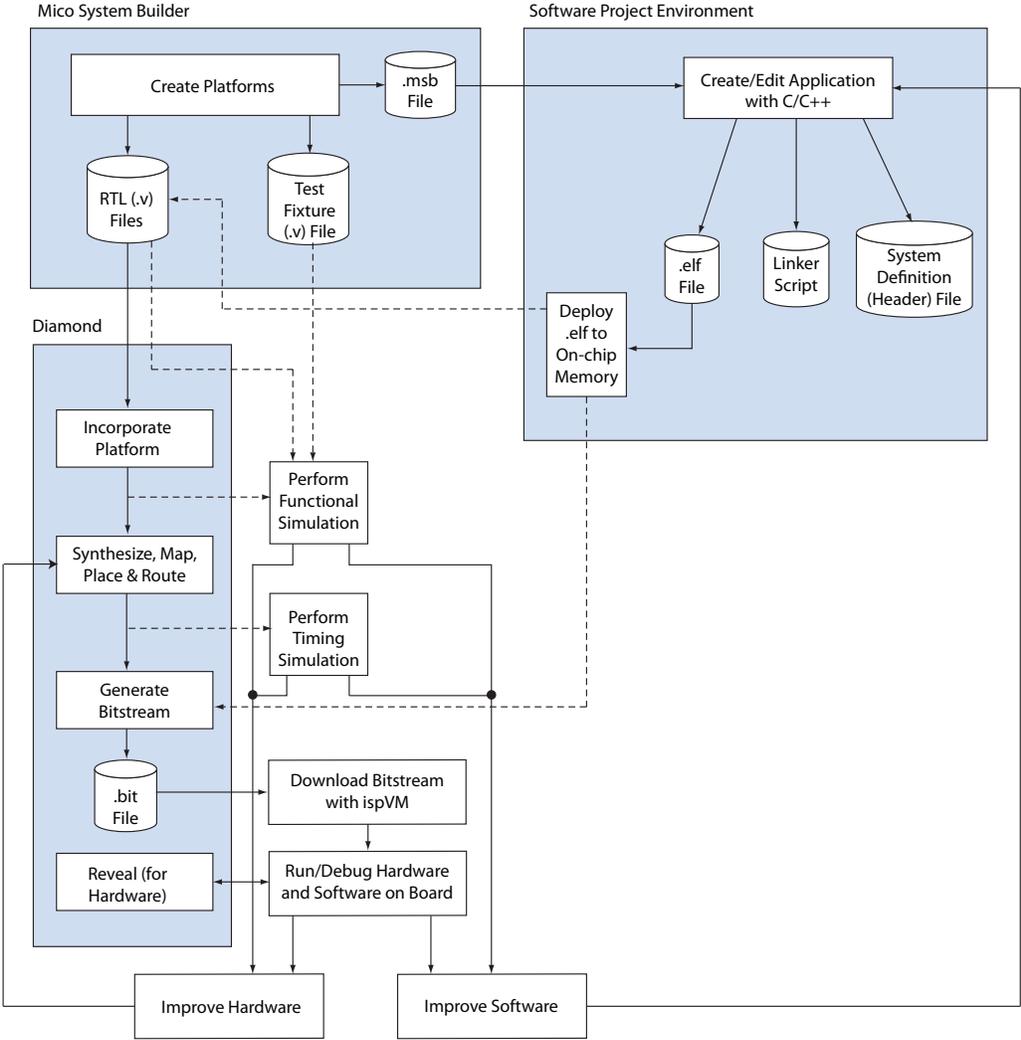
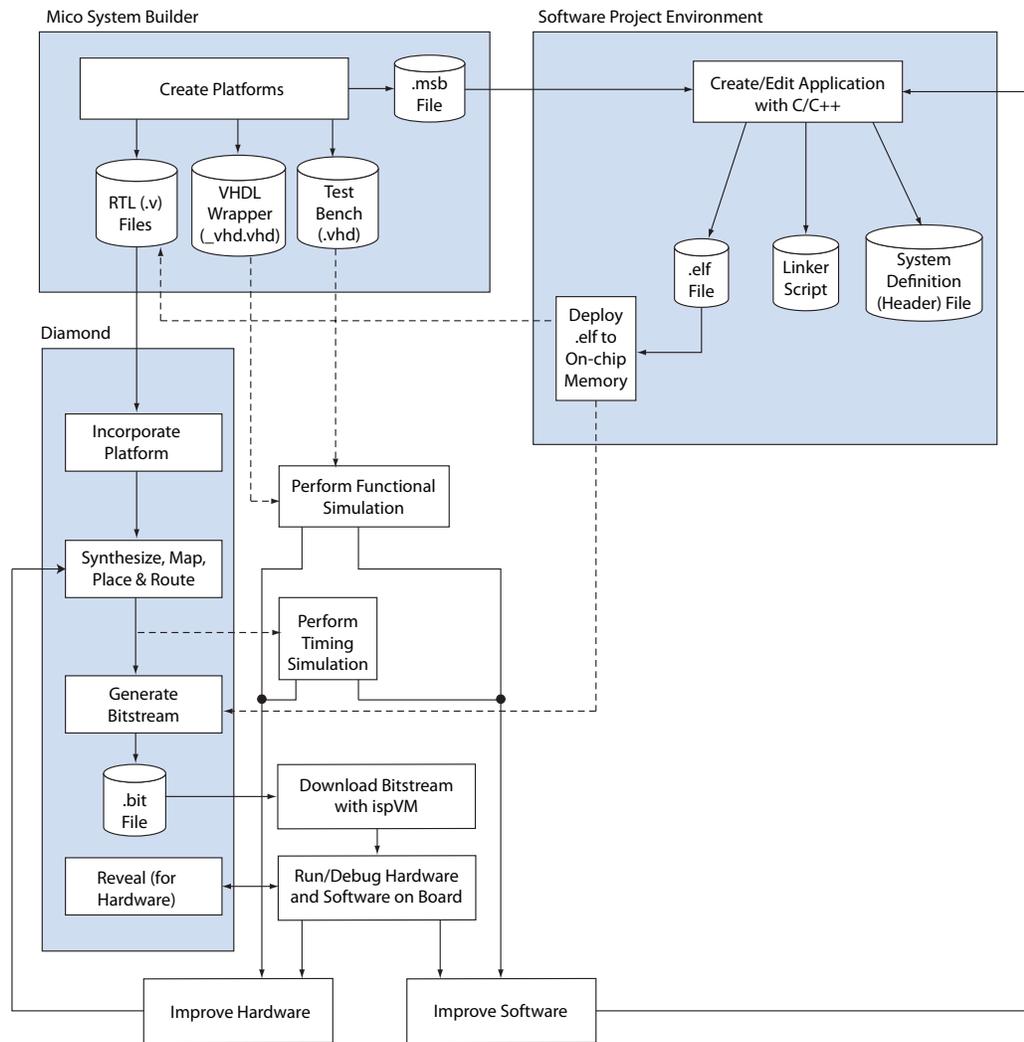


Figure 3: Design Flow for Windows VHDL Users, Using Mixed Verilog/VHDL Design Entry



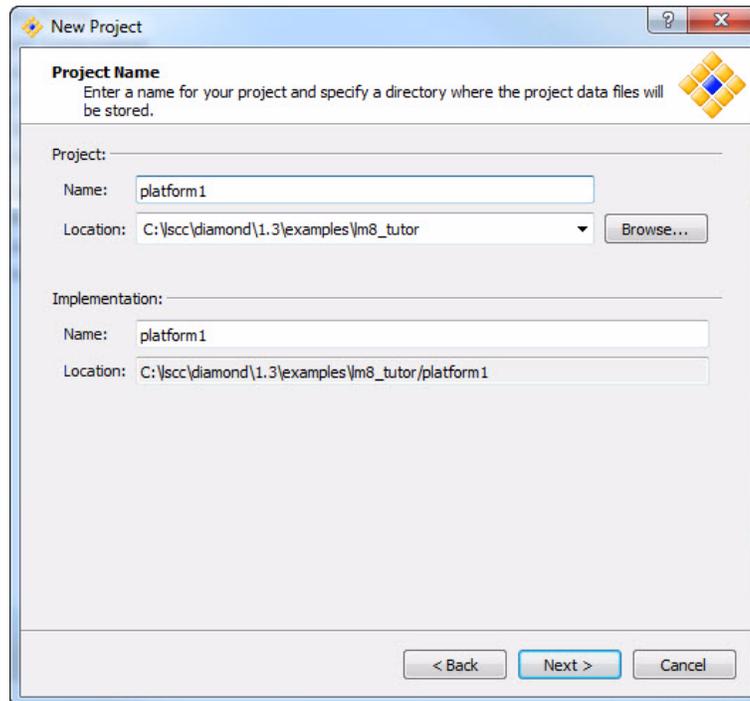
Task 1: Create a New Lattice Diamond Project

As a first step, you will create a new project in Diamond.

To create a new Lattice Diamond project:

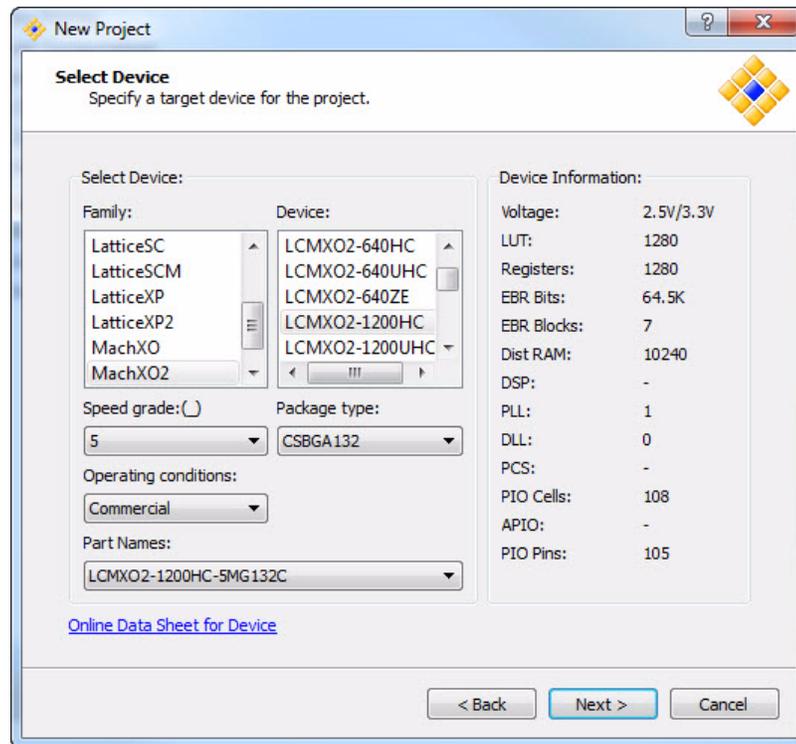
1. Create a folder called **lm8_tutor** in the following directory:
 - ▶ For Windows, `<Diamond_install_path>\examples`
 - ▶ For Linux, `~/LatticeMico`
2. Start Lattice Diamond:
 - ▶ On the Windows desktop, choose **Start > Programs > Lattice Diamond > Lattice Diamond**.
 - ▶ On the Linux command line, run the following script:
`<Diamond_install_path>/bin/linux/diamond.`
3. Choose **File > New > Project**, and then click **Next** in the New Project wizard.
4. In the New Project wizard dialog box, shown in Figure 4, select or specify the following:
 - a. In the Project Name box, enter **platform1**.
 - b. In the Location box, enter the path for the lm8_tutor directory:
 - ▶ For Windows, `<Diamond_install_path>\examples\lm8_tutor`
 - ▶ For Linux, `~/LatticeMico/lm8_tutor`

By default, Diamond uses the Project name and location for the implementation and fills in this information. Although you can change to a different name and directory for the first implementation, you will use the default settings for this tutorial.

Figure 4: New Project Wizard

5. Click **Next** to proceed to the Add Source dialog box, and then click Next. You will add the source later.
6. In Select Device dialog box, shown in Figure 5, make the following selections:
 - a. In the Family box, select MachXO2.
 - b. In the Device box, select LCMXO2-1200HC.
 - c. In the Speed grade box, select 5.
 - d. In the Package Type box, select CSBGA132.
 - e. In the Operating Conditions box, select Commercial.

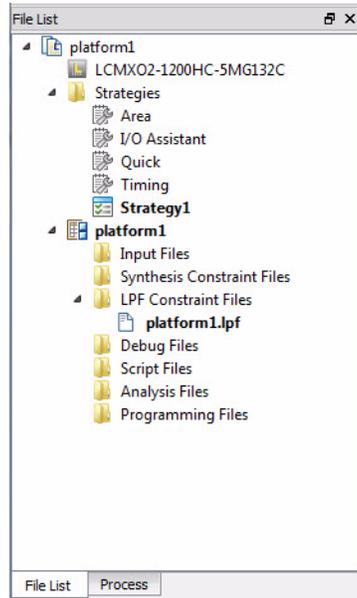
The dialog box should now resemble the illustration in Figure 5.

Figure 5: New Project Wizard – Select a Device Dialog Box

7. Click **Next**, and then click **Finish**.

In the File List, shown in Figure 6, the project name is shown at the top. The implementation name, which has the same name as the project name, is displayed in bold type, with the implementation icon . The project is assigned a default strategy, Strategy1, which is also displayed in bold type with the strategy icon . A strategy is a collection of settings for logic synthesis, place, and route. You can view these settings by double-clicking the strategy name. The platform1 project is also assigned a logical preference file, platform1.lpf.

Figure 6: Diamond File List



Task 2: Create LatticeMico8 Platform

In Task 1, you created a blank Diamond project. The Diamond project is a placeholder for the LatticeMico8 microcontroller platform. You use LatticeMico System Builder (MSB) to create the microcontroller platform. MSB allows you to select components to attach to the microcontroller. Additionally, MSB allows you to customize each of the attached components. After all components are attached to the microcontroller, you use MSB to generate Verilog or VHDL source code that describes a microcontroller-based System-on-a-Chip (SOC). You then enter the HDL source code into the Diamond project in order to create the bitstream used to configure the FPGA.

The steps in this section describe how to build a LatticeMico8 microcontroller SOC that is intended for developing and debugging LatticeMico8-based systems. During system development, the FPGA resources and the firmware are in a state of flux, undergoing many changes. When you deploy a LatticeMico8 microcontroller, as described here, you reduce the impact of the on-going changes in the development environment.

Create a New MSB Platform

Now you will create a new platform in MSB.

To create a new platform:

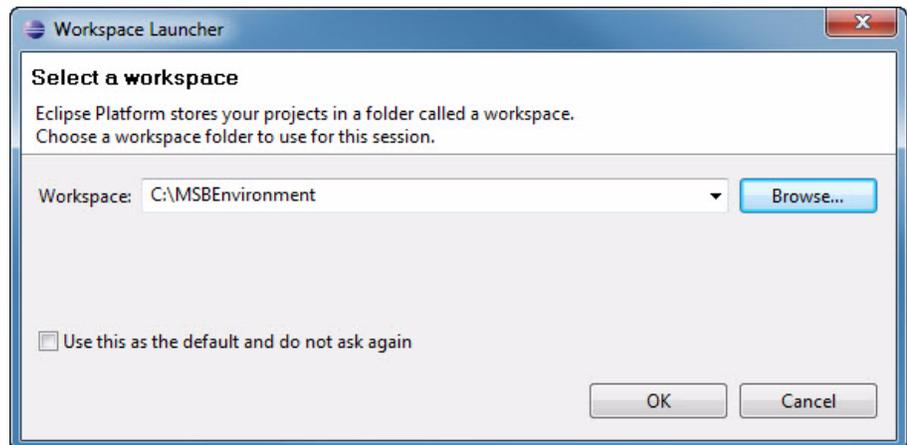
1. In Windows, from the Start menu, choose **Programs > Lattice Diamond > Accessories > LatticeMico System**.

For Linux, on the Linux command line, run the following script:

```
<Diamond_install_path>/micosystem/LatticeMicoLauncher
```

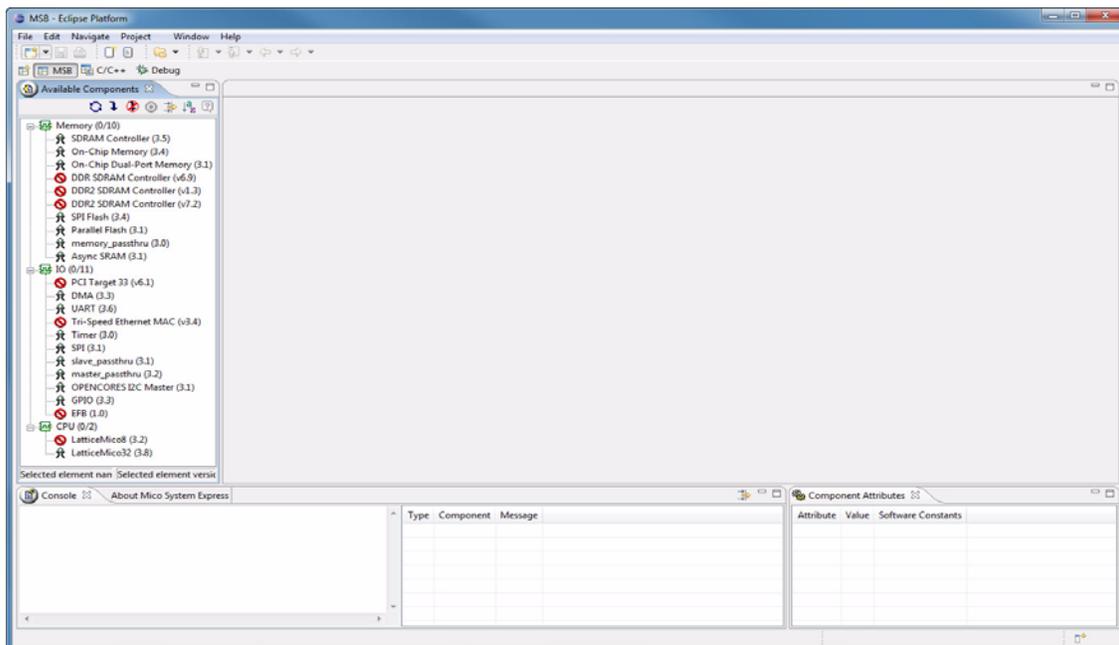
The Workspace Launcher dialog box, shown in Figure 7, displays a default workspace location for the platform.

Figure 7: Workspace Launcher Dialog Box



2. Accept the default location, or click the Browse button to select a different location. To keep the same workspace for future sessions, select the “Use this as the default and do not ask again” option.
3. Click **OK**. The LatticeMico System interface now appears, as shown in Figure 8.

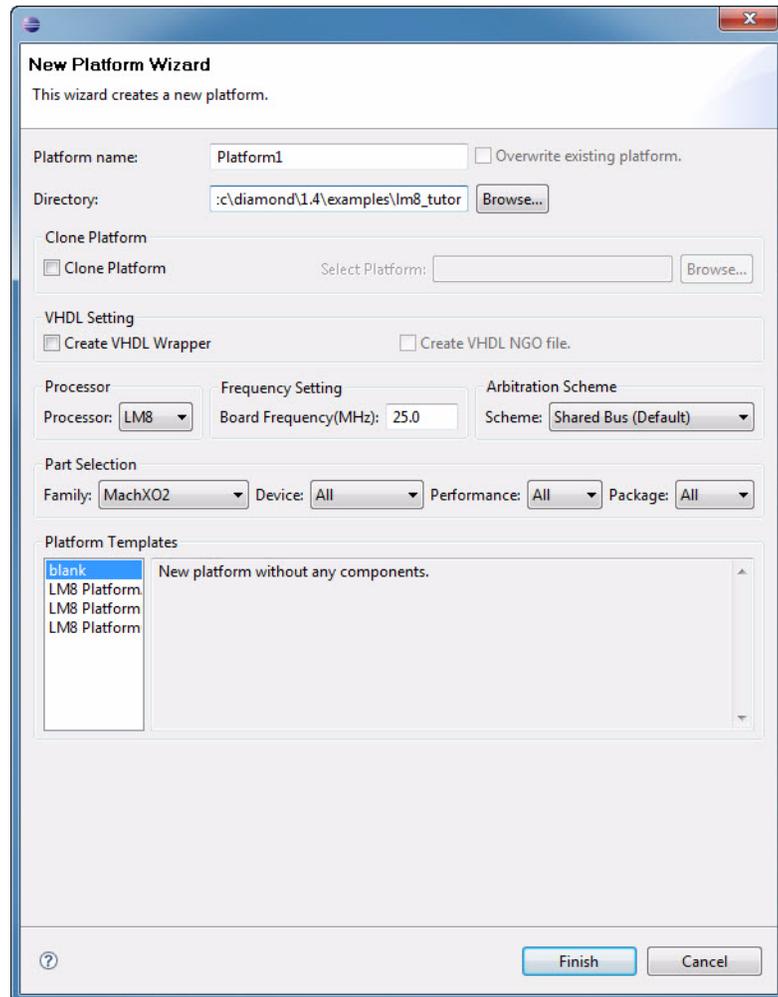
Figure 8: LatticeMico System Interface



4. In the upper left-hand corner of the graphical user interface, select **MSB**, if it is not already selected, to open the MSB perspective.
5. Choose **File > New Platform**.
6. In the New Platform Wizard dialog box, make the following selections:
 - a. In the Platform Name box, enter **platform1**.
 - b. In the Directory box, browse to the `lm8_tutor` directory and click **OK**:

- ▶ For Windows, <Diamond_install_path>\examples\lm8_tutor
 - ▶ For Linux, ~/LatticeMico/lm8_tutor
- c. Do one of the following:
- ▶ If you are generating a platform in Verilog, leave the Create VHDL Wrapper unselected.
 - ▶ If you are generating a platform in mixed Verilog/VHDL, select only Create VHDL Wrapper.
- d. In the Arbitration scheme box, select **Shared Bus (Default)** from the drop-down menu, if it is not already selected.
- e. In the Processor box, select **LM8**.
- f. In the Device Family section, select **MachXO2** from the Family menu and **All** from the Device menu.
- g. In the Platform Templates box, select **blank**.

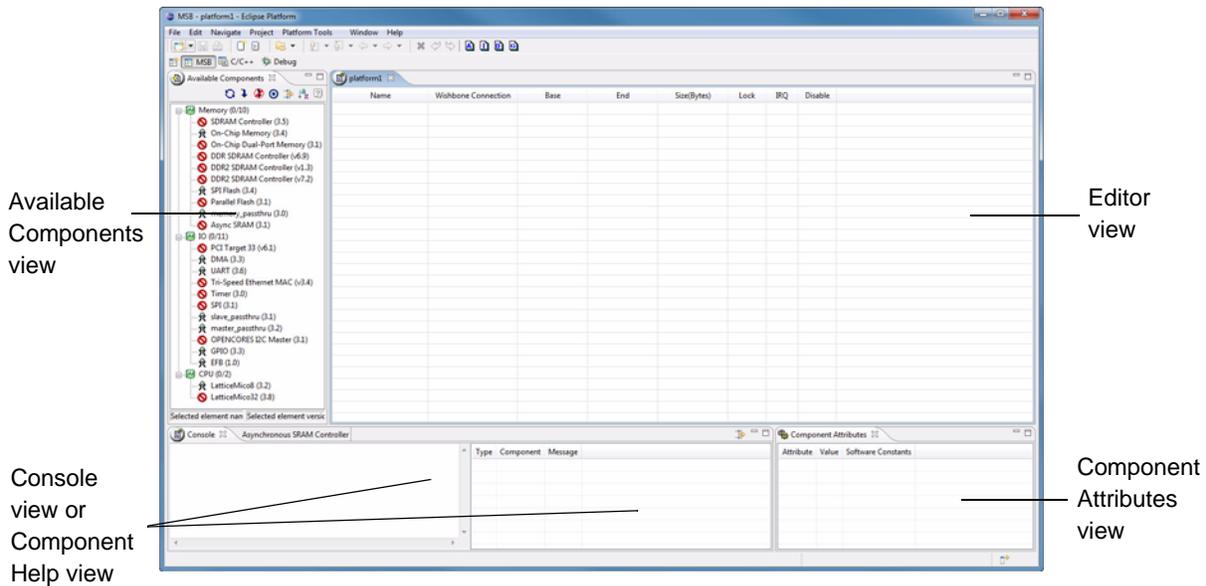
The New Platform Wizard dialog box should look like the illustration in Figure 9.

Figure 9: New Platform Wizard Dialog Box

7. Click **Finish**.

The MSB perspective now appears, as shown in Figure 10.

Figure 10: MSB Perspective



The MSB perspective consists of the following views:

- ▶ Available Components view, which displays all the available components that you can use to create the design:
 - ▶ A list of hardware components: CPUs, memories, components, and bus interfaces. Bus interfaces can be masters or slaves (see “Specify the Connections Between Master and Slave Ports” on page 21 for more information on masters and slaves). The component list shown in Figure 10 is the standard list that is given for each new platform.

Note

Not all components are available for each FPGA device and/or CPU. Depending on the selection made for the Processor and Device Family in Figure 9, some components are disabled. All disabled components are marked in red, as shown in Figure 10.

- ▶ You can double-click on a component to open a dialog box that enables you to customize the component before it is added to the design. The component is then shown in the Editor view.
- ▶ Editor view, which is a table that displays the components that you have chosen in the Available Components view. It includes the following columns:
 - ▶ Name, which displays the names of the chosen components and their ports
 - ▶ Wishbone Connection, which displays the connectivity between master and slave ports
 - ▶ Base, which displays the start addresses for components with slave ports. This field is editable.

- ▶ End, which displays the end addresses for components with slave ports. This field is not editable. The value of the end address is equivalent to the value of the base address plus the value of the size.
- ▶ Size (Bytes), which displays the number of addresses available for component access.
- ▶ Lock, which indicates whether addresses are locked from any assignments. If you lock a component, its address will not change when you select Platform Tools > Generate Address.
- ▶ IRQ, which displays the interrupt priorities of all components that have interrupt lines connected to the LatticeMico8 microcontroller. The LatticeMico8 microcontroller can accept up to 8 external interrupt lines.
- ▶ Disable, which indicates whether components are temporarily excluded from the design
- ▶ Component Help view, which displays information about the component that you selected in the Available Components view. The Help page displays the name of the component—for example, “LatticeMico Timer” or “LatticeMico UART—and gives a brief description of the function of the component. It also provides a list and explanation of the parameters that appear in the dialog box when you double-click the component. If you click the  icon next to the component name, you can view a complete description of the component in a PDF file.
- ▶ Console view, which displays informational and error messages output by MSB

Component Attributes view, which displays the name, parameters, and values of the component selected in the Available Components view or the Editor view. This view is read-only

Add the Microcontroller Core

The first step in building the platform is to add the CPU. In this release, the LatticeMico8 and LatticeMico32 CPUs are available. To add the LatticeMico8 microcontroller core:

1. Under CPU in the Available Components view, click **LatticeMico8** to view the information available about the LatticeMico8 microcontroller.

Information about the LatticeMico8 microcontroller, including the parameters that you can set for it, now appears in the Component Help view and in the Component Attributes view in the lower third of the screen. If you click the  icon in the Component Help view, you can view the LatticeMico8 Reference Manual, which provides a complete description of the microcontroller.

2. Double-click **LatticeMico8** to open the Add LatticeMico8 dialog box. Alternatively, you can select **LatticeMico8**, and then click the Add Component button .

The parameters in the dialog box, shown in Figure 11, correspond to those in the table in the Component Help view.

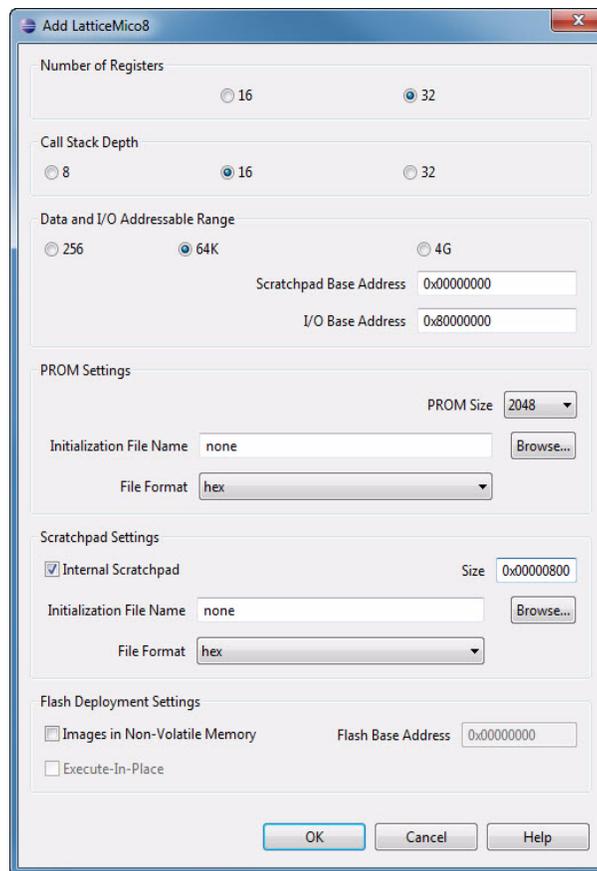
3. In the Add LatticeMico8 dialog box, do the following:
 - a. Set the “Number of Registers” to 32.
 - b. Set the “Call Stack Depth” to 16.
 - c. Set the address ranges for the Scratchpad and Peripheral (I/O) regions of LatticeMico8.
 - ▶ The maximum size of the two regions is defined by the contents of the field “Data and I/O Addressable Range”. Set this value to 64Kbytes for the tutorial.
 - ▶ The base address of the Scratchpad region is identified by the value in the field “Scratchpad Base Address”. Set this value to 0x00000000 for the tutorial. The software developer can use LatticeMico8 instructions – *lsp*, *lspi*, *ssp*, and *sspi* – to access this region. The LatticeMico8 compiler will map all of the software application’s data in to this region by default.
 - ▶ The base address of the Peripheral (I/O) region is identified by the value in the field “I/O Base Address”. Set this value to 0x80000000 for the tutorial. This software developer can use LatticeMico8 instructions – *import*, *importi*, *export*, and *exporti* – or compiler-provided builtin functions – *__builtin_export*, and *__builtin_import* – to access this region. Since the LatticeMico8 compiler does not map data accesses in to this region by default, the software developer must explicitly use the aforementioned instructions or functions to direct the compiler to access this region.
 - d. The LatticeMico8 PROM is either internal to the microcontroller, or located in a non-volatile memory. The following settings determine the PROM settings.
 - ▶ The size of the PROM is defined in the “PROM Size” box and it determines the maximum number of LatticeMico8 instructions that can be stored in it. The size is required regardless of whether the PROM is internal or external. Set this value to 2048 (i.e. 2048 LatticeMico8 assembly instructions).
 - ▶ We need to provide a PROM initialization file that contains the software application’s code image when the PROM is internal. The field “Initialization File Name” will be set up in Task 4.

Note

In this tutorial the LatticeMico8 PROM is internal. If the developer wants the PROM to reside in non-volatile memory such as SPI flash, he must enable the “Images in Non-Volatile Memory” and “Execute-In-Place” options in the LatticeMico8 configuration GUI.

- e. The LatticeMico8 Scratchpad is either internal to the microcontroller, or located in any other volatile memory. The following settings determine the Scratchpad settings.
 - ▶ Select the “Internal Scratchpad” option to locate the Scratchpad within the microcontroller.

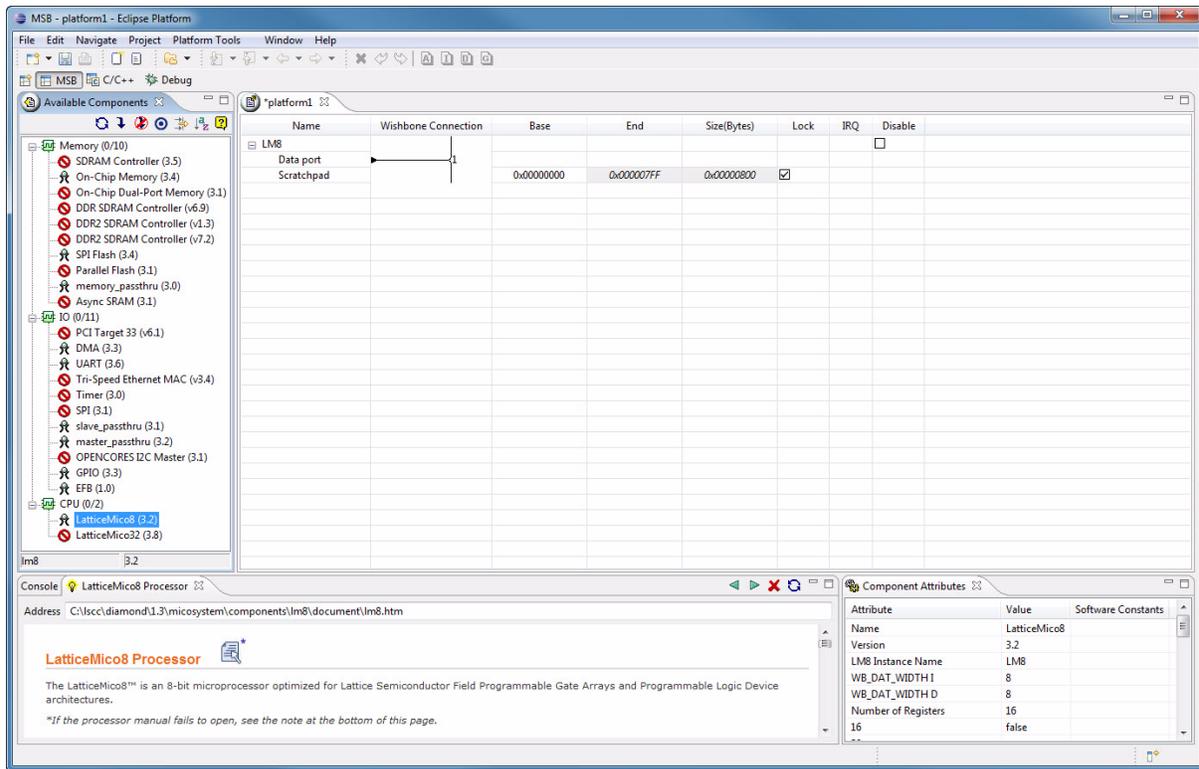
- ▶ The size of the Scratchpad is defined in the “Scratchpad Size”. The size is required regardless of whether the Scratchpad is internal or external since the LatticeMico8 compiler uses this value to determine the maximum number of bytes that can be within the software application’s data. Set this value to 0x00000800 (i.e. 2046 bytes).
 - ▶ We need to provide a Scratchpad initialization file that contains the software application’s data image when the Scratchpad is internal. The field “Initialization File Name” will be set up in Task 4.
- f. Click **OK** to accept the default settings for the rest of the options. Figure 11 shows the settings that are used for LatticeMico8 within the tutorial.

Figure 11: LatticeMico8 Microcontroller Core Dialog Box

Once you click OK, information about the microcontroller now appears in the Name, Wishbone Connection, Base, End, and Size columns of the table in the Editor view.

The MSB perspective now shows the LatticeMico8 Microcontroller in the Editor View, as shown in Figure 12.

Figure 12: MSB Perspective with Microcontroller



The Wishbone Connection column graphically displays the types of ports and connections. Black horizontal lines with outbound arrows indicate master ports, whereas blue horizontal lines with inbound arrows indicate slave ports. The vertical lines are associated with master ports, and the filled circles indicate connections between master and slave ports.

Add the Peripheral Components

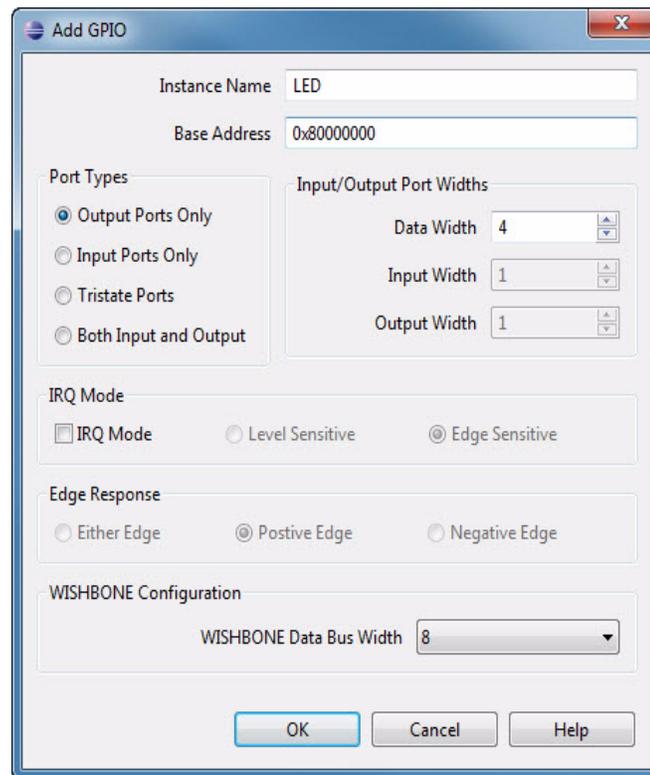
Now you will add the peripheral components to the platform.

Add the GPIO

The first peripheral component that you will add is the LatticeMico GPIO component, which provides a memory-mapped interface between a WISHBONE port and general-purpose I/O ports. The I/O ports connect either to on-chip user logic or to I/O pins that connect to devices external to the FPGA.

To add the GPIO to the platform:

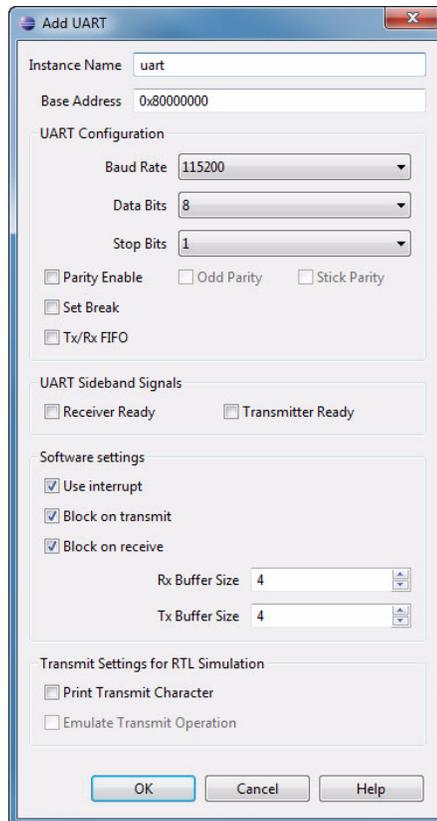
1. Under IO in the Available Components view, double-click **GPIO**. Alternatively, you can select **GPIO**, then click the Add Component button .
2. In the Add GPIO dialog box, shown in Figure 13, do the following:
 - a. In the Instance Name box, change the name of the GPIO to **LED**. For this tutorial, the GPIO block must be named LED. Failure to name the GPIO block LED will cause mismatches in the FPGA I/O pin names. The example C source code uses this instance name to access the GPIO registers.
 - b. Change the setting of the Data Width option to **4**.
 - c. Change the setting of the WISHBONE Data Bus Width to **8**.
 - d. Click **OK** to accept the default settings for the rest of the options.

Figure 13: Add GPIO Dialog Box**Add the UART**

The final component that you will add is a LatticeMico universal asynchronous receiver-transmitter (UART), a core that contains a receiver and a transmitter. The receiver performs serial-to-parallel conversion of the asynchronous data frame received at its serial data input pin. The transmitter performs parallel-to-serial conversion on the 8-bit data received from the CPU.

To add the UART to the platform:

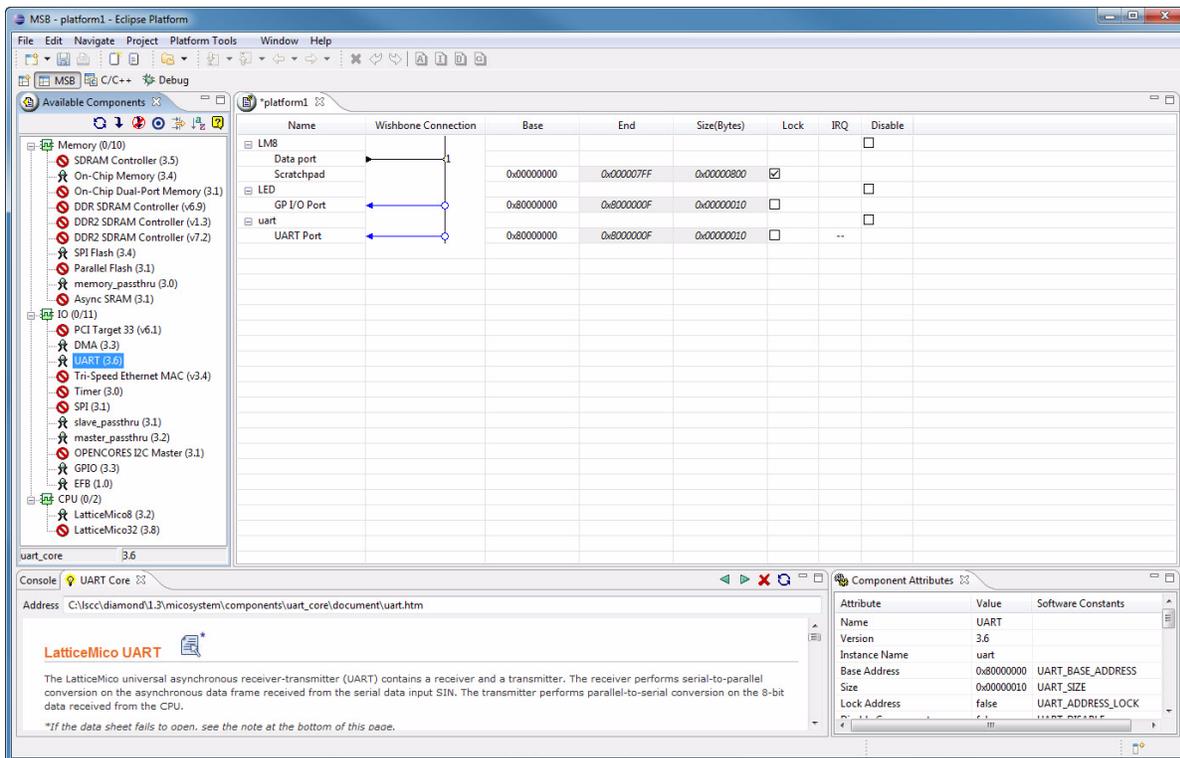
1. Under IO in the Available Components view, double-click **UART** to open the dialog box. Alternatively, you can select **UART**, and then click the Add Component button .
2. In the Add UART dialog box, shown in Figure 14, click **OK** to accept the default settings.

Figure 14: Add UART Dialog Box.

Specify the Connections Between Master and Slave Ports

The MSB perspective now resembles the illustration in Figure 15.

Figure 15: MSB Perspective After Addition Components



The connections that you will make between the master and slave ports in the Editor view will reflect the access scheme shown in Figure 1 on page 4. The following information applies to master and slave ports in the Editor view:

- ▶ There are two types of ports: master ports and slave ports.
 - ▶ A master port can initiate read and write transactions.
 - ▶ A slave port cannot initiate transactions but can respond to transactions initiated by a master port if it determines that it is the targeted component for the initiated transaction.
- ▶ A master port can be connected to one or more slave ports.
- ▶ A component can have one or more master ports, one or more slave ports, or both.
- ▶ Horizontal lines with outbound arrows sourced from a component port indicate a master port.
- ▶ Horizontal lines with inbound arrows targeting a component port indicate a slave port.
- ▶ The vertical lines are associated with horizontal lines with outbound arrows (that is, master ports) to facilitate “connectivity” from a master port to a slave port. A circle represents the intersection of the vertical line and a horizontal line associated with a slave port.
- ▶ A filled circle indicates a connection between the master port represented by the vertical line and the slave port represented by the horizontal line associated with the filled circle.

- ▶ A hollow circle indicates an absence of connection between the master port represented by the vertical line and the slave port represented by the horizontal line associated with the hollow circle. This can be seen in Figure 16.
- ▶ The numbers next to the lines representing the master ports are the priorities in which the master ports can access the slave ports. You can change the priority of these connections by following the instructions in the online Help for LatticeMico System.

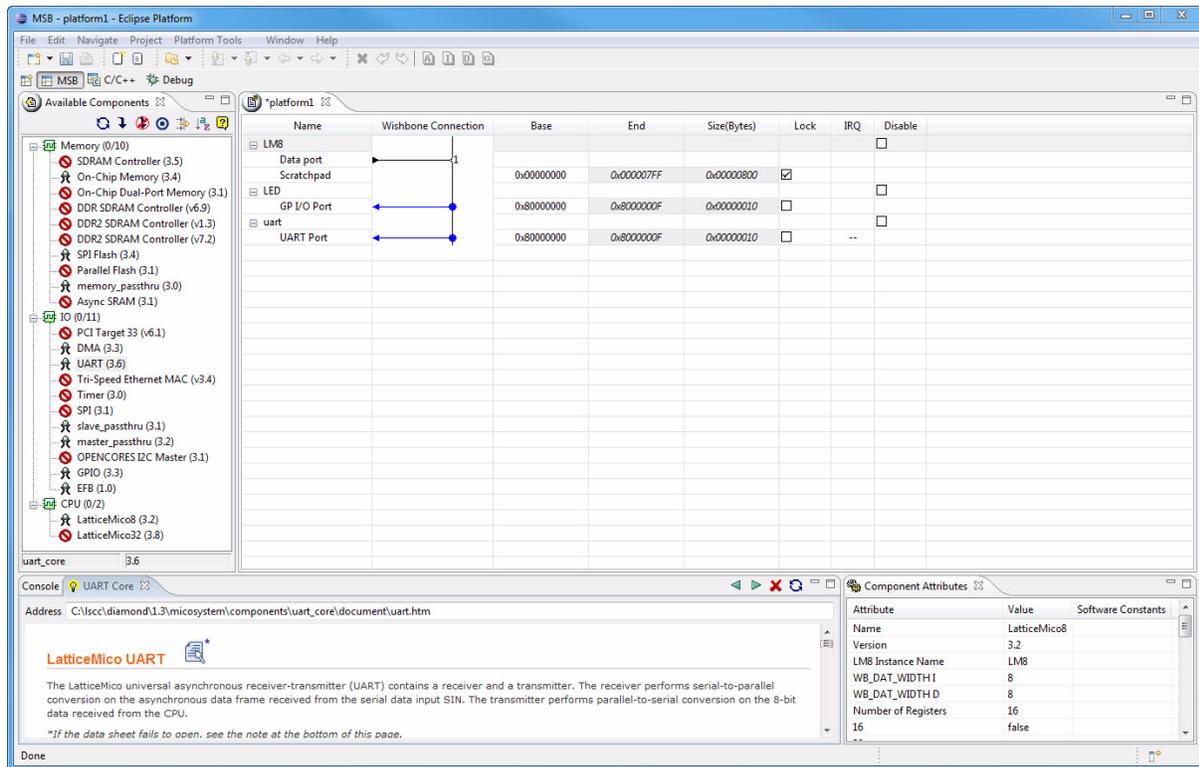
To specify the connections between master and slave ports:

1. Connect the data port of the LatticeMico8 microcontroller to the LatticeMico GPIO slave port by clicking the circle in the WISHBONE Connection column of the GPIO Port row.
2. Connect the data port of the LatticeMico8 microcontroller to the LatticeMico UART slave port by clicking the circle in the WISHBONE Connection column of the UART Port row.

Figure 16 shows the resulting connections in the Editor view. This tutorial example uses the shared-bus arbitration scheme. For information about bus arbitration schemes, refer to the “LatticeMico8 Developer's User Guide.”

MSB automatically generates the arbiter, depending on which arbitration scheme is selected. In the case of the shared-bus arbitration scheme, it generates the microcontroller platform to allow multiple master ports access to multiple slave ports over a single shared bus. In the diagram, the data port accesses the LatticeMico GPIO and the LatticeMico UART.

Figure 16: Resulting Connections



Assign Component Addresses

The next step is for MSB to generate an address for each component with slave ports. Addresses are specified in hexadecimal notation. Components with master ports are not assigned addresses.

Note

You can only edit the addresses in the Base column in the Editor View. You cannot edit the addresses in the End column. The value of the end address is equivalent to the value of the base address plus the value of the size.

You will not assign individual addresses. To automatically assign component addresses:

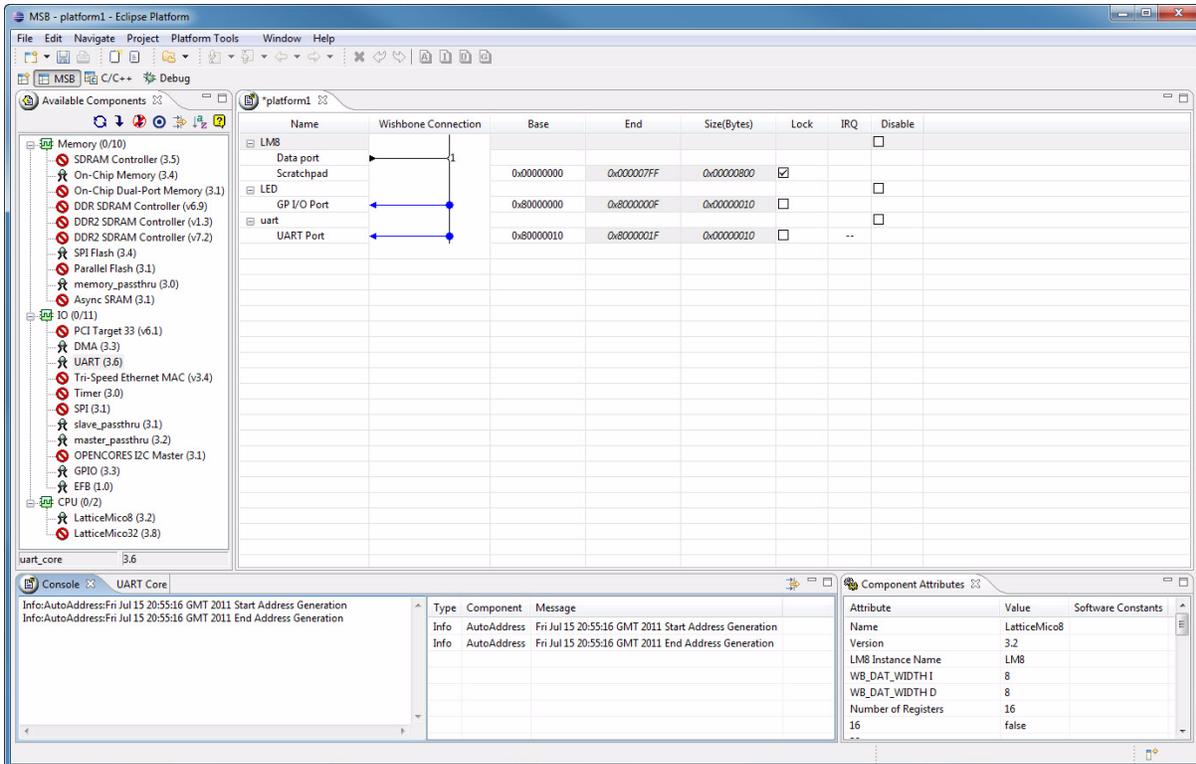
- ▶ Choose **Platform Tools > Generate Address**, or click the **Generate Base Address** button , or right-click in the Editor view and choose **Generate Address** from the pop-up menu. The addresses now appear in the Base and End columns in the Editor view, in hexadecimal notation. Slave components that are not memories are assigned addresses within the 0x80000000-0xFFFFFFFF memory range. The Generate Address command sets A31 of each of the I/O components to '1

Your MSB perspective should now resemble the example shown in Figure 17. The base addresses that you see in your Editor view might be different from those shown.

Note

Address and size values that appear in italic font in the Editor view cannot be changed.

Figure 17: MSB Perspective with Component Addresses Assigned



Assign Interrupt Request Priorities

The interrupt request priority is the order in which hardware components request computing time from the CPU. Now you will assign an interrupt request priority (IRQ) to all components that feature a dash in the IRQ column of the Editor view. You cannot assign interrupt priorities to components lacking this dash in the IRQ column, such as memories and CPUs.

To assign interrupt priorities for all components providing interrupt functionality:

- ▶ Choose **Platform Tools > Generate IRQ**, or click the **Generate IRQ** button , or right-click in the Editor view and choose **Generate IRQ** from the pop-up menu.

Note

To reassign an interrupt priority for a specific component, go to the IRQ column in the row for the component, click on the current interrupt priority number, and choose the new priority number from the drop-down menu. Explicitly assigned interrupt priorities will not be overridden by the interrupt generator tool. The Lock control does not affect IRQ assignment; it only prevents auto-assignment of the Base Address.

If you accidentally assign duplicate priorities, MSB will issue an error message in the Console view when you select **Platform Tools > Generate IRQ**.

Perform a Design Rule Check

You will want to perform a design rule check to verify that components in the platform have valid base addresses, interrupt request values, and other fundamental properties.

To perform a design rule check:

- ▶ Choose **Platform Tools > Run DRC**, or click the **Run DRC** button , or right-click in the Editor view and choose **Run DRC** from the pop-up menu.

In the Console view, MSB shows that there are no errors in the platform.

Generate the LatticeMico8 Platform

You are now ready to generate the microcontroller platform. During the generation process, MSB creates the following files in the "lm8_tutor\platform1\soc" directory:

- ▶ A platform1.msb file, which describes the platform. It is in XML format and contains the configurable parameters and bus interface information for the components.
- ▶ A platform1.v (Verilog) file, which is used by both Verilog and mixed Verilog/VHDL users:
 - ▶ For Verilog users, the platform1.v file is used in both simulation and implementation. It instantiates all the selected components and the interconnect described in the MSB graphical user interface. This file is the top-level simulation and synthesis RTL file

passed to Diamond. It includes the .v files for each component in the design. These .v files are used to synthesize and generate a bitstream to be downloaded to the FPGA. The first time Generate is run, the Verilog source for each component in the platform, which is located in "<Diamond_install_path>/micosystem/components/<component>", is copied into a subdirectory called "components". The components subdirectory is a sibling to the "soc" subdirectory.

- ▶ For mixed Verilog/VHDL users, the platform1.v file is used in simulation only. A mixed-mode Verilog and VHDL simulator, such as Aldec® Active-HDL™, is needed for functional simulation.
- ▶ A platform1_vhd.vhd (VHDL) file is created if you selected the "Create VHDL Wrapper" option in the New Platform Wizard dialog box. It is intended to be used only to incorporate the Verilog-based platform into a mixed Verilog/VHDL design. The platform1_vhd.vhd contains the top-level design used for synthesis. This top-level design file instantiates the platform1 component.
- ▶ A platform1.ngo file, if you selected both the Create VHDL Wrapper and the Create VHDL NGO File options in the New Platform Wizard dialog box. The .ngo file is required for Linux Verilog/VHDL users. It is a synthesized version of platform1.v.

The contents of the platform1.msb file are used by the C/C++ development tools. The C/C++ source code build process extracts the base address information and the size of each component and uses the information to build GNU LD linker files. Each time the Generate function is run, it causes the C/C++ compiler to consider the C/C++ source code to be out of date. This means that the source code will be rebuilt from scratch after each Generate process.

To generate the microprocessor platform:

- ▶ Click anywhere in the Editor view and choose **Platform Tools > Run Generator**, or click the **Run Generator** button , or right-click and choose **Run Generator** from the pop-up menu.

The Console view displays the output as MSB processes the design. If you are using Verilog, you will see *Finish Generator* in the Console view when the generator is finished. If the project was created with the "Create VHDL Wrapper" option selected, the project is a mixed Verilog/VHDL flow and the generator silently launches Synplify synthesis and Diamond to create the wrapper. If you are using mixed Verilog/VHDL, you must wait for the *Finish VHDL Wrapper* message to appear in the Console view.

The MSB perspective now looks like the illustration in Figure 18. The assigned addresses for the components might differ. As shown in Figure 19, MSB generates a platform1_inst.v file, which contains the Verilog instantiation template for use in a design where the platform is not the top-level module. For a mixed Verilog/VHDL project, no equivalent file is generated.

Figure 18: MSB Perspective

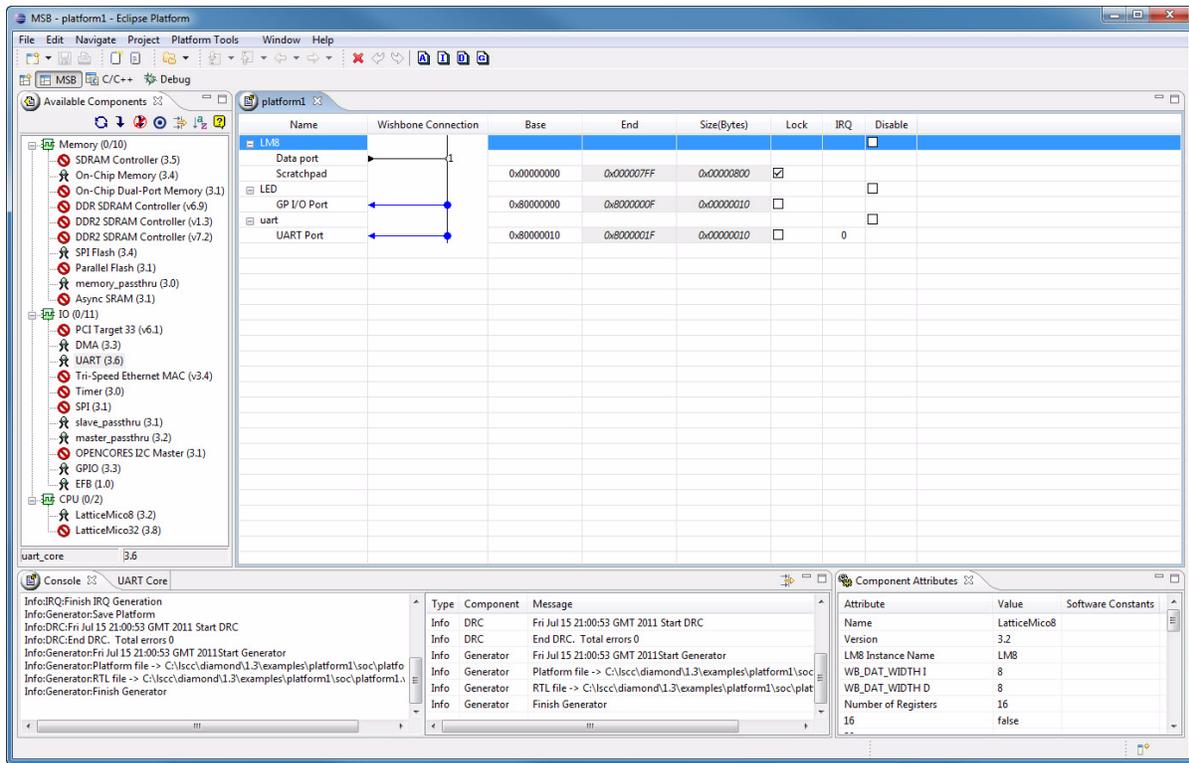


Figure 19: platform1_inst.v file

```

`include "../components/lm8/rtl/verilog/lm8_include_all.v"
`include "../components/gpio/rtl/verilog/gpio.v"
`include "../components/gpio/rtl/verilog/tpio.v"
`include "../components/uart_core/rtl/verilog/uart_core.v"
platform1 platform1_u (
    .clk_i(clk_i),
    .reset_n(reset_n)
    , .LEDPIO_OUT(LEDPIO_OUT) // [4-1:0]
    , .uartSIN(uartSIN) //
    , .uartSOUT(uartSOUT) //
);

```

Create a User Top-Level Module

As mentioned in the previous section, the platform that is generated by MSB may not be the top-level module. In this scenario, you can use the contents of `platform1_inst.v`, as shown in Figure 19, within your top-level module. This instantiates the platform generated by MSB within your top-level module.

You must now create a top level file name `platform1_top.v` to instantiate the MSB-generated platform. Create the file as follows:

1. Copy and paste the contents of Figure 20 into a text editor.
2. Name the file platform1_top.v, and save the file in the following directory:
 - ▶ For Windows,
 - <Diamond_install_path>\examples\lm8_tutor\platform1\soc
 - ▶ For Linux, ~/LatticeMico/lm8_tutor/platform1/soc

You will import this file into your Diamond project, in the section “Importing the Source Files” on page 38.

Figure 20: User Top Level File

```

`include "../soc/platform1.v"

module platform1_top
(
    input reset_n,
    input uartSIN,
    output uartSOUT,
    output [3:0] LEDPIO_OUT
);

// MachX02 internal oscillator generates platform clock
wire clk_in;
OSCH OSCH_inst (.OSC(clk_in), .SEDSTDBY(), .STDBY(1'b0));
defparam OSCH_inst.NOM_FREQ = "24.18";

platform1 platform1_u
(
    .clk_i (clk_in),
    .reset_n (reset_n),
    .LEDPIO_OUT (LEDPIO_OUT),
    .uartSIN (uartSIN),
    .uartSOUT (uartSOUT)
);

endmodule

```

Task 3: Create Software Application Code

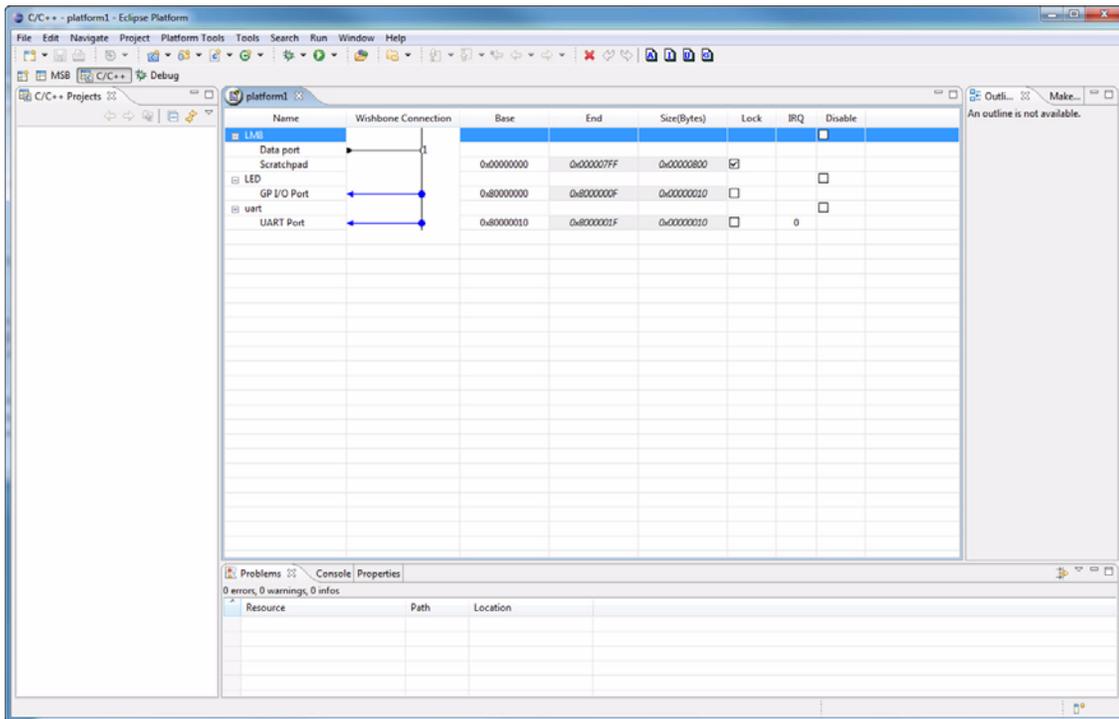
In this task, you create the software application by using C/C++ in the LatticeMico System Software Project Environment (C/C++ SPE). The software application is the code that runs on the LatticeMico8 microcontroller to control the components, the bus, and the memories. The application is written in C.

C/C++ SPE is based on the Eclipse environment and provides an integrated development environment for developing, debugging, and deploying C/C++ applications. C/C++ SPE uses the GNU C tool chain (compiler, assembler, linker, debugger, and other necessary utilities) that has been customized for the LatticeMico8 microcontroller. C/C++ SPE uses the same LatticeMico

System interface as MSB, but it uses a different perspective called the C/C++ perspective.

To activate the C/C++ perspective select **C/C++** in the upper left-hand corner of MSB graphical user interface. The C/C++ perspective is shown in Figure 21.

Figure 21: Software Project Environment



The C/C++ perspective consists of the following views:

- ▶ C/C++ Projects view, which lists C/C++ SPE projects that have been created
- ▶ Navigator view, which shows all of the file system's files under the workspace directory
- ▶ Editor view, which is similar to the Editor view in the MSB perspective
- ▶ Outline view, which displays the structure of the file currently open in the Editor view
- ▶ Problems view, which displays any error, warning, or informational messages output by C/C++ SPE
- ▶ Console view, which displays informational messages output by the C/C++ SPE build process
- ▶ Properties view, which displays the attributes of the item currently selected in the C/C++ Projects view. This view is read-only.
- ▶ Search view, which displays the results of a search when you choose **Search > File**.
- ▶ Tasks view, which shows the tasks running concurrently in the background

- ▶ Make Targets view, which is not used in LatticeMico8 C projects

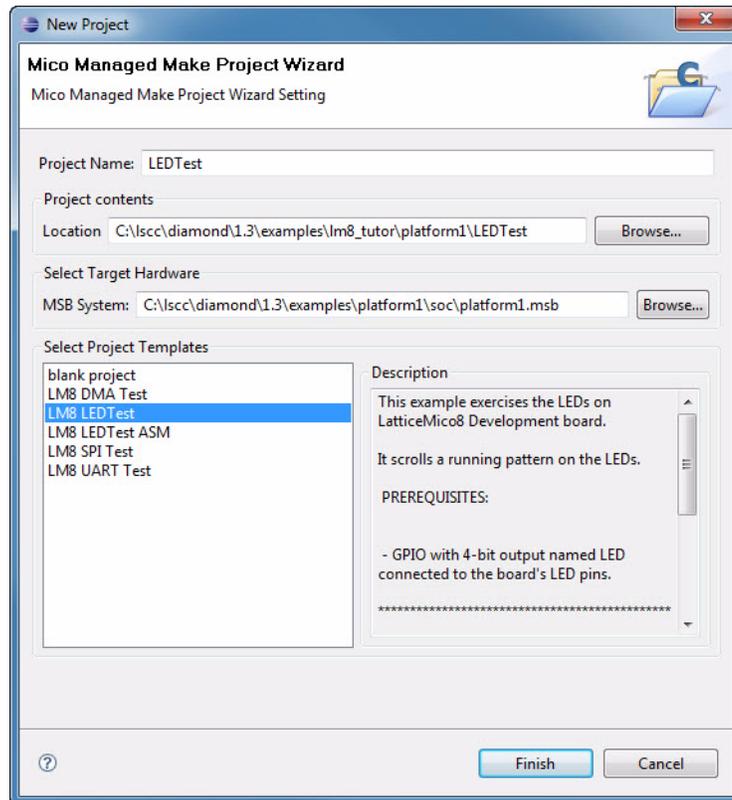
Create a New C/C++ SPE Project

You will create a new project in C/C++ SPE, import the platform1.msb file into the project, select the application code template to use so that you do not have to write the code yourself, and compile the code.

To create a new C/C++ SPE project:

1. In the C/C++ perspective, choose **File > New > Mico Managed Make C Project**.
2. In the New Project dialog box, make the following selections:
 - a. In the Project Name box, enter **LEDTest**.
 - b. In the Location box, browse to the following directory:
 - ▶ For Windows,
<Diamond_install_path>\examples\lm8_tutor\platform1
 - ▶ For Linux, ~/LatticeMico/lm8_tutor/platform1
 - c. In the MSB System box, browse to the following location, select the platform1.msb file in the dialog box, and click **Open**.
 - ▶ For Windows, lm8_tutor\platform1\soc\platform1.msb
 - ▶ For Linux, lm8_tutor/platform1/soc/platform1.msb
 - d. In the Select Project Templates box, select **LM8 LEDTest** as the template for the application code.

The New Project dialog box should resemble the figure shown in Figure 22.

Figure 22: New Project Dialog Box**Note**

The directory shown in the Location box in the Project Contents field is where the software project directory will be created. Your user files will be placed in this directory.

3. Click **Finish**.

Now you see the source code in the middle pane of the C/C++ perspective, as shown in Figure 23.

an executable file image, as required by the GNU linker. C/C++ SPE uses this information to ensure that the program code and data are located at the correct addresses. Although it is not covered in this tutorial, the LatticeMico C/C++ SPE enables you to easily specify a custom linker script to be used in lieu of the generated script for the managed build.

- ▶ A LatticeMico8 software executable linked formal file (.elf). The .elf file contains the LatticeMico8 instructions and information about the pre-initialized data. This tutorial generates a file called platform1.elf.

These files are included within the directory LEDTest and are generated by the C/C++ SPE in the background. All the contents of this directory except user's C application – in this case it is LEDTest.c – are dynamically generated, and any changes to them are overwritten from build to build. The platform1 library directory (LEDTest\platform1\) contains platform-specific information for the building of an application.

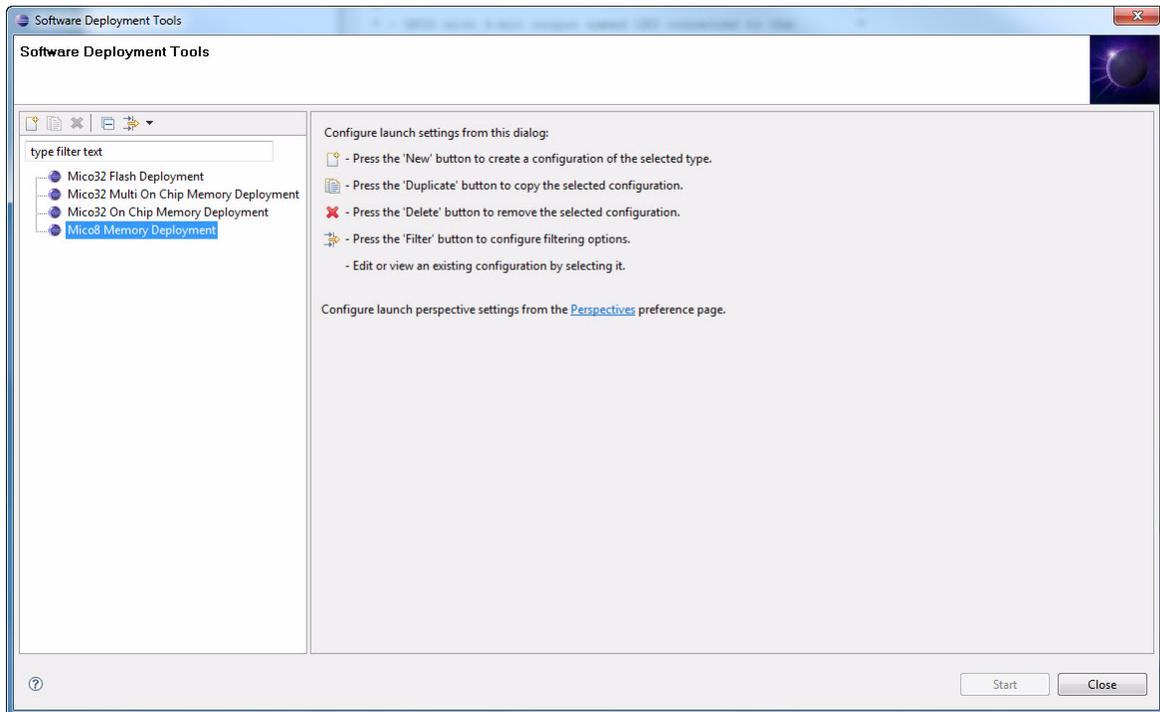
Task 4: Deploy Software Application Code to LatticeMico8

The LEDTest application is in an executable linked format (ELF) that contains both, code and data. This ELF file must be converted in to hexadecimal initialization files, one each for code and data. The initialization file that contains LEDTest code is called prom_init.hex and is used to initialize the LatticeMico8 PROM. The initialization file that contains LEDTest data is called scratchpad_init.hex and is used to initialize the LatticeMico8 Scratchpad.

C/C++ SPE provides an easy-to-use interface for preparing LEDTest for deployment to the LatticeMico8 PROM and Scratchpad. Consult the "LatticeMico8 User Guide" for functional details on the deployment utility.

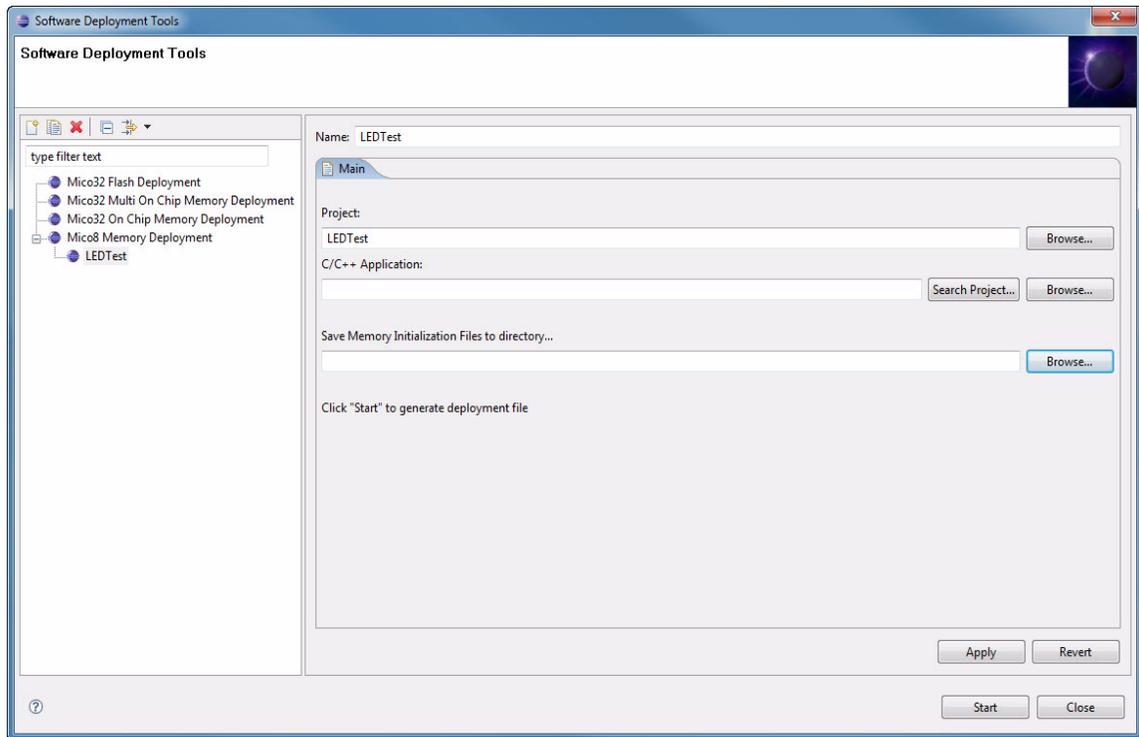
To prepare LEDTest for deployment:

1. In the C/C++ perspective, select **LEDTest** and choose **Tools > Software Deployment**. The Software Deployment Tools dialog box appears, as shown in Figure 24.

Figure 24: Software Deployment Tools Dialog Box

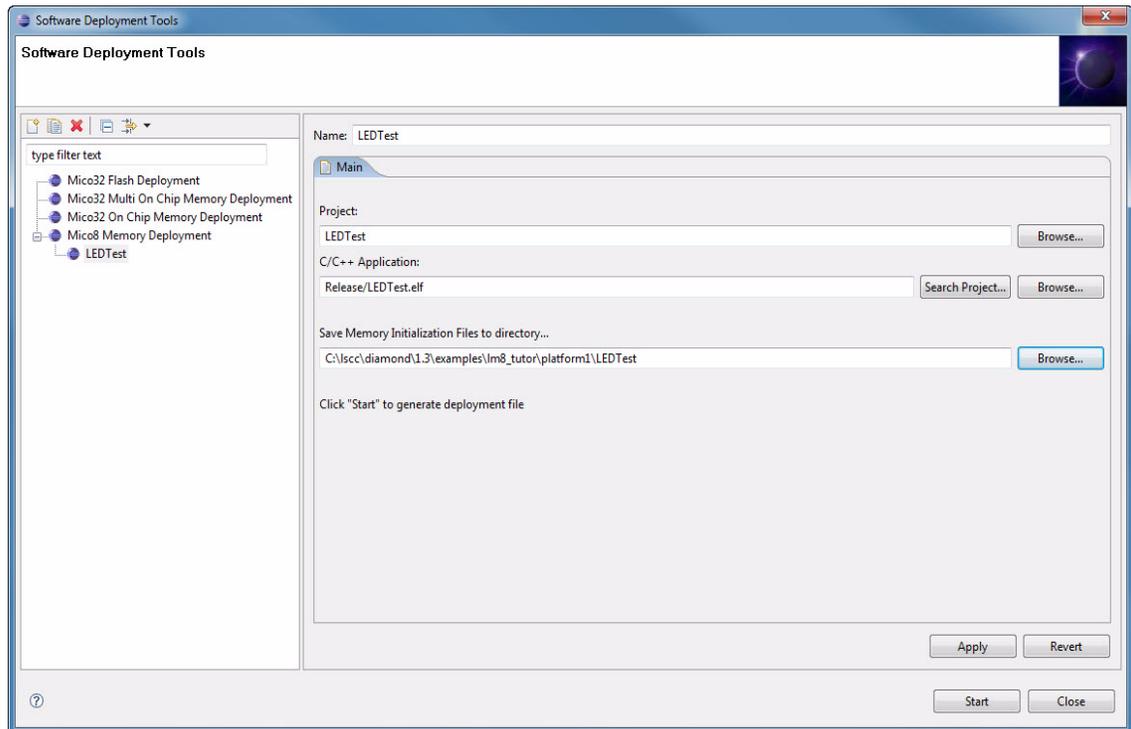
2. Select **Mico8 Memory Deployment**, and click on **New launch configuration** button. The Software Deployment Tools dialog now displays a Mico8 memory deployment configuration settings pane similar to that shown in Figure 26.
 - a. In the Name box, enter the value LEDTest
 - b. Browse to LEDTest and select it as the value for the Project box
 - c. Click on the Search Project button and select LEDTest.elf as the value for the C/C++ Application box
 - d. Select the folder to which the initialization files are saved.

Figure 25: Mico8 Configuration Options



3. The configuration options within the Mico8 memory deployment configuration settings pane should look like Figure 26.

Figure 26: Mico8 Memory Deployment Configuration Settings

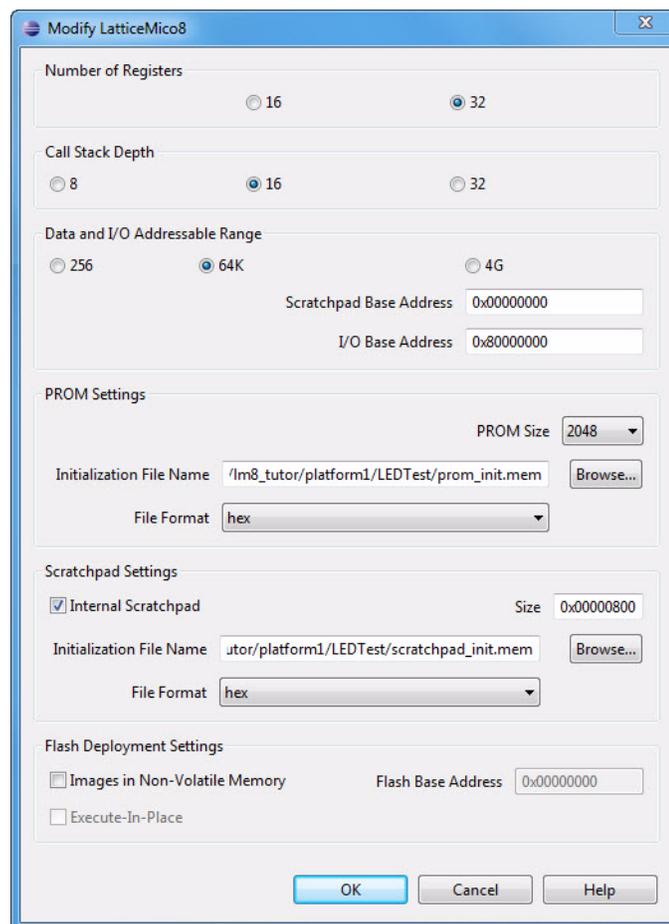


4. Click **Apply** and then click **Start**.
5. The two initialization files prom_init.mem and scratchpad_init.mem should now be saved within the output folder specified in Step 2.d above.

To initialize LatticeMico8 with LEDTest initialization files:

1. Return to the MSB perspective.
2. Double-click on LM8 instance within the platform.
 - a. Initialize the PROM in the **Initialization File Name** box in **PROM Settings** by browsing to the prom_init.mem file.
 - b. Initialize the Scratchpad in the **Initialization File Name** box in **Scratchpad Settings** by browsing to the scratchpad_init.mem file.
3. The LM8 instance GUI should look like Figure 27. Click **OK**.

Figure 27: LatticeMico8 Instance GUI



4. Regenerate the hardware platform by clicking anywhere in the Editor view and choose **Platform Tools > Run Generator**, or clicking the **Run Generator** button, or right-clicking and choose **Run Generator** from the pop-up menu.

The LatticeMico8 Verilog source code is now configured to use the `prom_init.mem` and `data_init.mem` files that implement LEDTest.

Task 5: Generate Platform Bitstream

The next step in the flow is to generate the microcontroller bitstream. This bitstream is then downloaded to the FPGA on the circuit board. To generate the bitstream, return to Diamond.

Import the MSB Output File

First, you must import the Verilog file output by MSB, the Verilog and VHDL files for mixed Verilog/VHDL, or the EDIF file created by the synthesis tool into Diamond. The process of importing the generated platform file into Diamond is the same for Verilog and mixed Verilog/VHDL, except that you must import the VHDL wrapper file in addition to the Verilog file for mixed Verilog/VHDL.

Configure the Lattice Diamond Environment

The Diamond build process has the ability to operate in two different modes. One is to copy all HDL source files into the Diamond project directory, and the other is to reference them in their current directory structure. The LatticeMico8 build requires that the source files remain in the directory structure created by MSB. The default Diamond behavior is to leave the files where they are, but it is advisable to verify that Diamond is configured correctly.

1. In Diamond, choose **Tools > Options**.
2. Under Environment, in the left pane of the Options dialog box, select **General**.
3. If the option **Copy file to Implementation's Source directory when adding existing file** is selected, clear this option and click **OK**.

Importing the Source Files

You import the top-level file, `platform1_top.v`, into Diamond. To import the Verilog or Verilog/VHDL files for the tutorial example:

1. In Diamond, choose **File > Add > Existing File**.
2. In the dialog box, browse to the directory containing the top level Verilog source code.
 - ▶ For Windows, `<Diamond_install_path>\examples\lm8_tutor\platform1\soc`
 - ▶ For Linux, `~/LatticeMico/lm8_tutor/platform1/soc/`

3. Do one of the following:
 - ▶ Select the platform1_top.v file (Verilog), and click **Add**.
 - ▶ If your design is mixed Verilog/VHDL, select the platform1_top.v, and the platform1_vhd.vhd files and click **Add**.
4. If your design is mixed Verilog/VHDL, perform the following additional steps:
 - a. Choose **Project > Property Pages**.
 - b. In the dialog box, select the project name that appears in bold type next to the implementation icon .
 - c. In the right pane, click inside the Value cell for “Top-Level Unit” and select platform1_top from the drop-down menu.
 - d. Click inside the Value cell for “Verilog Include Search Path,” and then click the browse button to open the “Verilog Include Search Path” dialog box.
 - e. In the dialog box, click the **New Search Path** button , browse to the <platform1>\soc directory, and click **OK**.
 - f. Click **OK** to add the path to the Project Properties and close the “Verilog Include Search Path” dialog box.
 - g. Click **OK** to return to the Diamond main window.

Connect the Microcontroller to the FPGA Pins

You have two options for connecting the microcontroller to the FPGA pins:

- ▶ Manually create the pin constraints and import them into Diamond.
- ▶ Import a preconfigured preference file into Diamond.

For this tutorial, you will import a preconfigured pin preference file into Diamond. Create a preconfigured pin preference file named “untitled.lpf” as follows:

1. Copy and paste the contents of Figure 28 into a text editor.
2. Name the file untitled.lpf, and save the file in the following directory:
 - ▶ For Windows, <Diamond_install_path>\examples\lm8_tutor
 - ▶ For Linux, ~/LatticeMico/lm8_tutor/

To import this preconfigured pin preference file:

1. In Diamond, select the File List tab and double-click **Strategy1**.
2. In the **Strategies** dialog box, select Translate Design in the left pane.
3. In the right pane, double-click the cell in the Value column for Consistent Bus Name Conversion.

Figure 28: Preconfigured Pin Preference File

```

AUTOMOTIVE ;
BLOCK RESETPATHS ;
BLOCK ASYNCPATHS ;
BLOCK INTERCLOCKDOMAIN PATHS ;

LOCATE COMP "reset_n" SITE "K2" ;
IOBUF PORT "reset_n" IO_TYPE=LVCOS18 ;
#-----
# LEDs assumes the component instance name is LED & has
# a width of 4
#-----
LOCATE COMP "LEDPIO_OUT_0_" SITE "F1" ;
LOCATE COMP "LEDPIO_OUT_1_" SITE "F2" ;
LOCATE COMP "LEDPIO_OUT_2_" SITE "F3" ;
LOCATE COMP "LEDPIO_OUT_3_" SITE "H2" ;
IOBUF PORT "LEDPIO_OUT_0_" IO_TYPE=LVCOS18 ;
IOBUF PORT "LEDPIO_OUT_1_" IO_TYPE=LVCOS18 ;
IOBUF PORT "LEDPIO_OUT_2_" IO_TYPE=LVCOS18 ;
IOBUF PORT "LEDPIO_OUT_3_" IO_TYPE=LVCOS18 ;
#-----
# UART assumes the component instance name is uart
#-----
LOCATE COMP "uartSIN" SITE "H1" ;
LOCATE COMP "uartSOUT" SITE "H3" ;
IOBUF PORT "uartSIN" IO_TYPE=LVCOS18 ;
IOBUF PORT "uartSOUT" IO_TYPE=LVCOS18 ;
BLOCK JTAGPATHS;

```

4. Choose **Lattice** from the drop-down menu, and click **OK**.
5. In Diamond, choose **File > Add > Existing File**.
6. In the Add Existing File dialog box, do the following:
 - a. Select Constraint Files (*.lpf) from the Files of type menu.
 - b. Select the option Copy file to Implementation's Source directory.
 - c. Navigate to the directory containing the untitled.lpf file for the File Name box, and click Add.
Diamond adds the .lpf file to the project and displays file name and path in the File List.
7. In the File List pane, right-click **untitled.lpf** and choose **Set as Active Preference File**.
8. Diamond displays the untitled.lpf file and path in bold type, indicating that the untitled.lpf will now be used instead of the platform1.lpf file.

Perform Functional Simulation

You can optionally simulate the functionality of the output top-level platform1.v or platform1_vhd.vhd module by using a simulator such as Active-HDL in Diamond. See the Active-HDL online Help in Diamond for more information on this procedure.

- ▶ For Verilog simulation, you use platform1.v and all the Verilog files for each attached component.
- ▶ For mixed Verilog/VHDL simulation, you use platform1_vhd.vhd, platform1.v, and all the Verilog files for each attached component. You must use a mixed-language simulator such as ModelSim® SE or Aldec Active-HDL.

Perform Timing Simulation

You can optionally validate the timing of your design by performing timing simulation. Because timing simulation is a complex topic, it is not addressed in this tutorial. For information on timing simulation, see the Achieving Timing Closure in FPGA Designs Tutorial, the “Design Verification” topic in the Diamond online Help, or the “Strategies for Timing Closure” chapter of the FPGA Design Guide.

The timing simulation process automatically builds a database and maps, places, and routes the design.

Generate the Bitstream

Now you will generate a bitstream to download the microprocessor platform to the FPGA. If you did not perform timing simulation, the bitstream generation process will automatically synthesize, map, place, and route the design before it generates the bitstream.

To generate a bitstream:

1. In Diamond, select the Process tab.
2. In the Export Files section, double-click **JEDEC File**.

Diamond now generates a bitstream, platform1_platform1.jed, that is ready to be downloaded into the device. This process takes several minutes.

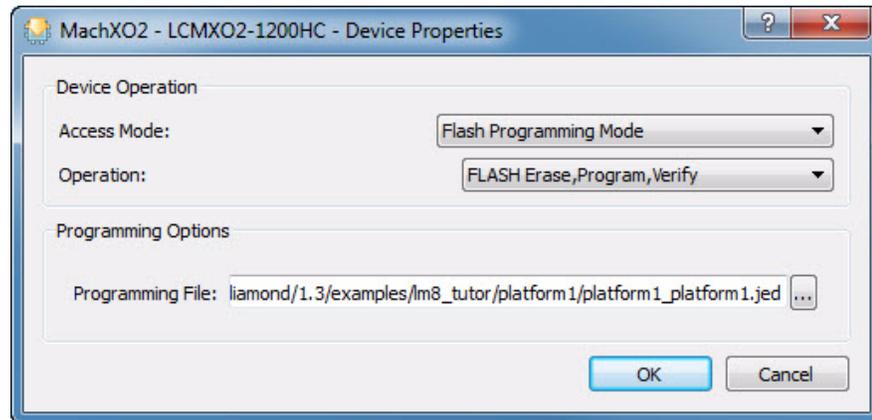
Task 6: Download Platform Bitstream to FPGA

The bitstream generated in the previous section contains all the information required to program the MachXO2 FPGA. Diamond Programmer sends the programming bitstream to the FPGA over a parallel port or USB port communications link.

Now you will use Programmer to download the hardware bitstream that you generated in the previous section to the FPGA on the board.

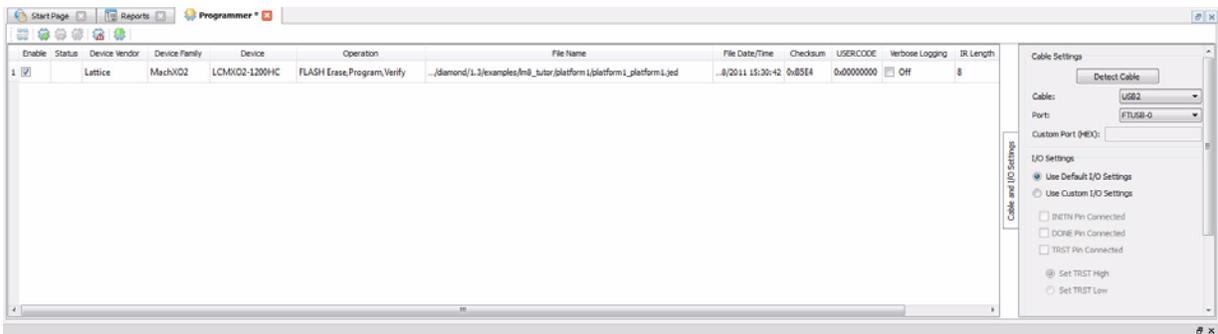
To download the bitstream to the FPGA on the board:

1. Remove any Lattice USB Programming cables from your system.
2. Connect the power supply to the development board.
3. Connect a USB cable from your computer to the MachXO2 Control Board. The USB cable must be connected to the USB target connector adjacent to the keypad. Give the computer a few seconds to detect the USB device on the MachXO2 Control board before moving to step 4.
4. In Diamond, choose **Tools > Programmer**.
5. In the Getting Started dialog box, choose **Create a new Blank Project**. Leave the **Import File to Current Implementation** box checked, and click **OK**. Programmer scans the device database, and then the Programmer view displays in Diamond.
6. In the Cable Settings dialog box on the right side of the Programmer window, do the following:
 - a. In the Cable box, select **USB2**.
 - b. In the Port box, choose the only setting available in the drop-down menu, **FTUSB-0**.
7. Double-click the File Name column. Click  to display the Open File dialog box, and browse to the platform1_platform1.jed file in the following directory:
 - ▶ For Windows,
<Diamond_install_path>\examples\lm8_tutor\platform1_platform1.jed
 - ▶ For Linux, ~/LatticeMico/lm8_tutor/platform1_platform1.jed
8. Click **Open**.
9. Double-click the Operation column to display the Device Properties dialog box, as shown in Figure 29, and choose the following settings:
 - ▶ For Access Mode, choose **Flash Programming Mode** from the pull-down menu.
 - ▶ For Operation, choose **Flash, Erase, Program, Verify** from the pull-down menu.

Figure 29: Device Properties Dialog Box.

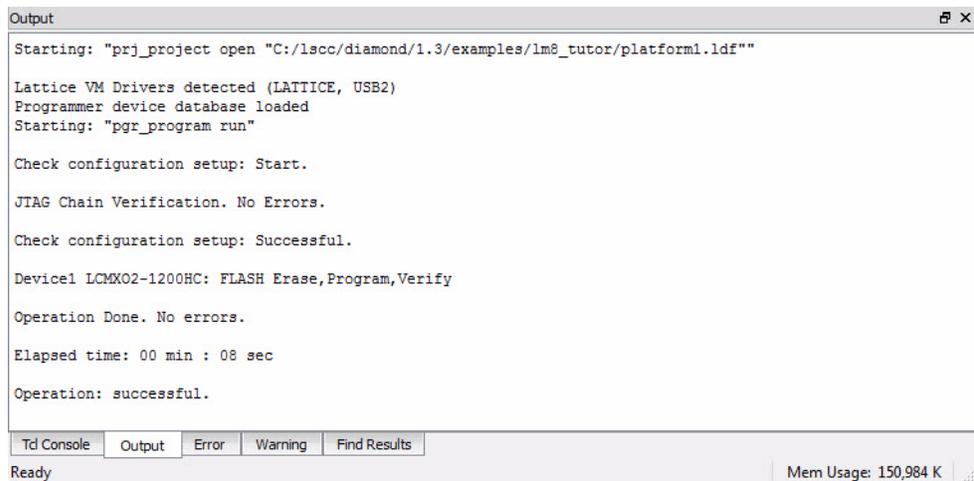
10. Click **OK**.

11. The Programmer view should look as shown in Figure 30.

Figure 30: Diamond Programmer

12. Click the Program button  on the Programmer toolbar to initiate the download.

13. Check the Programmer output console to see if the download passed, as shown in Figure 31. If the programming process succeeded, you will see a green-shaded PASS in the Programmer Status column.

Figure 31: Programmer Output Console

```
Output
Starting: "prj_project open "C:/lsc/diamond/1.3/examples/lm8_tutor/platform1.ldf""

Lattice VM Drivers detected (LATTICE, USB2)
Programmer device database loaded
Starting: "pgr_program run"

Check configuration setup: Start.

JTAG Chain Verification. No Errors.

Check configuration setup: Successful.

Device1 LCMXO2-1200HC: FLASH Erase,Program,Verify

Operation Done. No errors.

Elapsed time: 00 min : 08 sec

Operation: successful.
```

Td Console Output Error Warning Find Results

Ready Mem Usage: 150,984 K

14. At the end of this process, the FPGA is loaded with the microcontroller hardware configuration. If the design is successfully downloaded onto the MachXO2 device, the four LEDs on the MachXO2 Control Board should blink in sequence and repeat the pattern.
15. In Diamond, choose **File > Save**. In the Save .xcf File As dialog box, enter platform1.xcf in the File Name box, and click **Save**.

Summary

You have finished the LatticeMico8 Tutorial. In this tutorial, you have learned how to do the following:

- ▶ Set up a Lattice Diamond FPGA project.
- ▶ Create microcontroller platform for the LatticeMico8 embedded microcontroller in MSB.
- ▶ Create the software application code for the microcontroller platform with C/C++ SPE.
- ▶ Deploy the software application code to the microcontroller's internal PROM and Scratchpad.
- ▶ Generate a bitstream of the microcontroller platform in Diamond.
- ▶ Download the hardware bitstream to the FPGA on the board using Diamond Programmer.