



1 Introduction

This tutorial presents an introduction to the Quartus® II CAD system. It gives a general overview of a typical CAD flow for designing circuits that are implemented by using FPGA devices, and shows how this flow is realized in the Quartus II software. The design process is illustrated by giving step-by-step instructions for using the Quartus II software to implement a very simple circuit in an Altera FPGA device.

The Quartus II system includes full support for all of the popular methods of entering a description of the desired circuit into a CAD system. This tutorial makes use of the Verilog design entry method, in which the user specifies the desired circuit in the Verilog hardware description language. Two other versions of this tutorial are also available; one uses the VHDL hardware description language and the other is based on defining the desired circuit in the form of a schematic diagram.

The last step in the design process involves configuring the designed circuit in an actual FPGA device. To show how this is done, it is assumed that the user has access to the Altera DE-series Development and Education board connected to a computer that has Quartus II software installed. A reader who does not have access to the DE-series board will still find the tutorial useful to learn how the FPGA programming and configuration task is performed.

The screen captures in the tutorial were obtained using the Quartus II version 13.0; if other versions of the software are used, some of the images may be slightly different.

Contents:

- Typical CAD Flow
- Getting Started
- Starting a New Project
- Verilog Design Entry
- Compiling the Design
- Pin Assignment
- Simulating the Designed Circuit
- Programming and Configuring the FPGA Device
- Testing the Designed Circuit

2 Background

Computer Aided Design (CAD) software makes it easy to implement a desired logic circuit by using a programmable logic device, such as a field-programmable gate array (FPGA) chip. A typical FPGA CAD flow is illustrated in Figure 1.

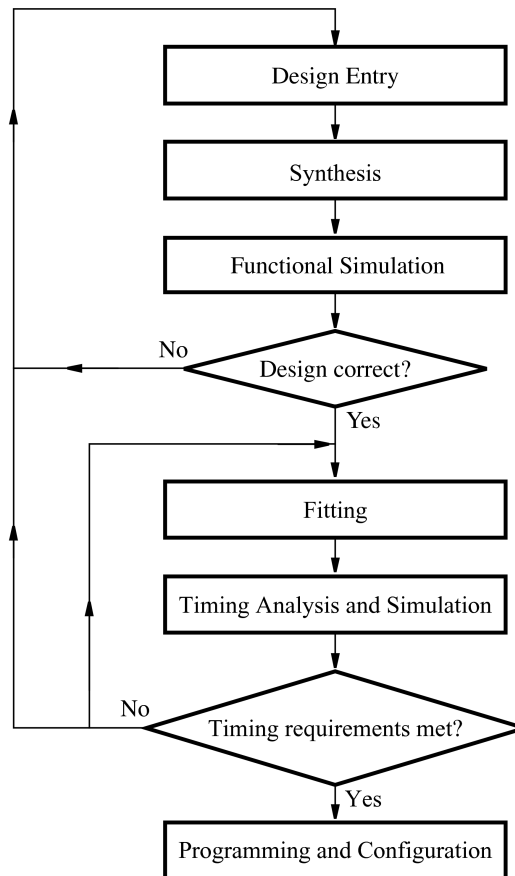


Figure 1. Typical CAD flow.

The CAD flow involves the following steps:

- **Design Entry** – the desired circuit is specified either by means of a schematic diagram, or by using a hardware description language, such as Verilog or VHDL
- **Synthesis** – the entered design is synthesized into a circuit that consists of the logic elements (LEs) provided in the FPGA chip
- **Functional Simulation** – the synthesized circuit is tested to verify its functional correctness; this simulation does not take into account any timing issues

- **Fitting** – the CAD Fitter tool determines the placement of the LEs defined in the netlist into the LEs in an actual FPGA chip; it also chooses routing wires in the chip to make the required connections between specific LEs
- **Timing Analysis** – propagation delays along the various paths in the fitted circuit are analyzed to provide an indication of the expected performance of the circuit
- **Timing Simulation** – the fitted circuit is tested to verify both its functional correctness and timing
- **Programming and Configuration** – the designed circuit is implemented in a physical FPGA chip by programming the configuration switches that configure the LEs and establish the required wiring connections

This tutorial introduces the basic features of the Quartus II software. It shows how the software can be used to design and implement a circuit specified by using the Verilog hardware description language. It makes use of the graphical user interface to invoke the Quartus II commands. Doing this tutorial, the reader will learn about:

- Creating a project
- Design entry using Verilog code
- Synthesizing a circuit specified in Verilog code
- Fitting a synthesized circuit into an Altera FPGA
- Assigning the circuit inputs and outputs to specific pins on the FPGA
- Simulating the designed circuit
- Programming and configuring the FPGA chip on Altera's DE-series board

3 Getting Started

Each logic circuit, or subcircuit, being designed with Quartus II software is called a *project*. The software works on one project at a time and keeps all information for that project in a single directory (folder) in the file system. To begin a new logic circuit design, the first step is to create a directory to hold its files. To hold the design files for this tutorial, we will use a directory *introtutorial*. The running example for this tutorial is a simple circuit for two-way light control.

Start the Quartus II software. You should see a display similar to the one in Figure 2. This display consists of several windows that provide access to all the features of Quartus II software, which the user selects with the computer mouse. Most of the commands provided by Quartus II software can be accessed by using a set of menus that are located below the title bar. For example, in Figure 2 clicking the left mouse button on the menu named **File** opens the menu shown in Figure 3. Clicking the left mouse button on the entry **Exit** exits from Quartus II software. In general, whenever the mouse is used to select something, the *left* button is used. Hence we will not normally specify which button to press. In the few cases when it is necessary to use the *right* mouse button, it will be specified explicitly.

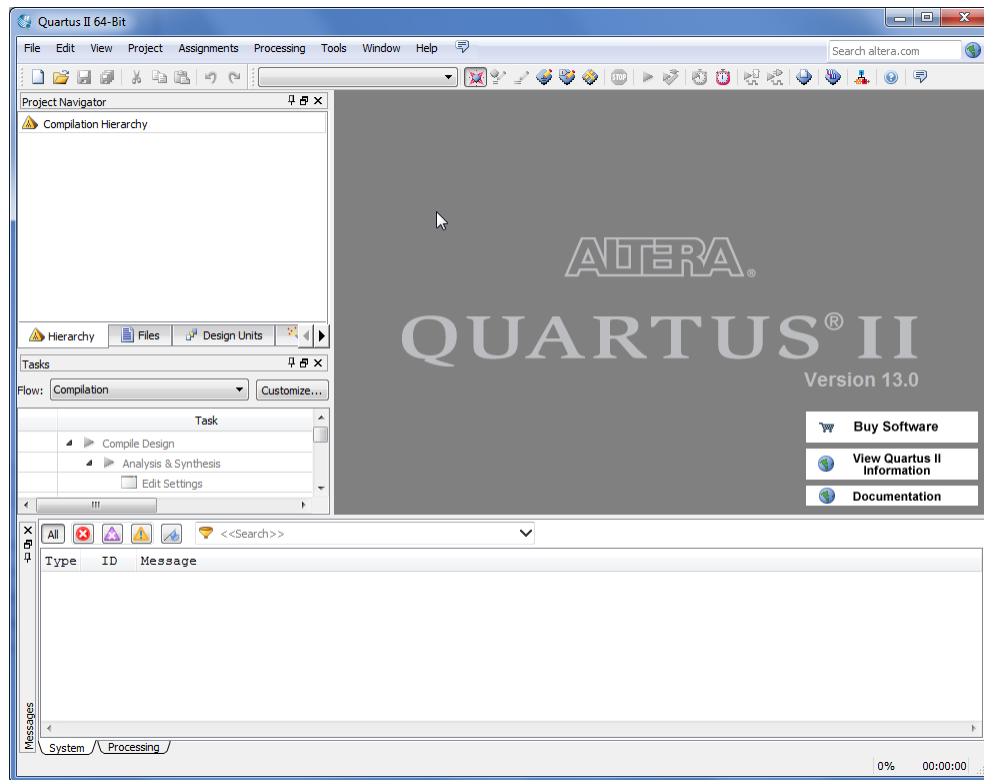


Figure 2. The main Quartus II display.

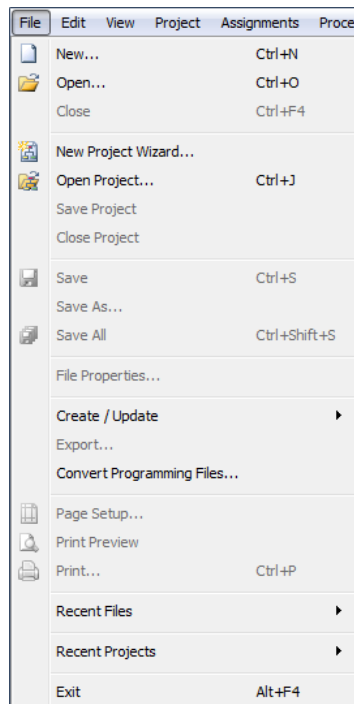


Figure 3. An example of the File menu.

For some commands it is necessary to access two or more menus in sequence. We use the convention **Menu1 > Menu2 > Item** to indicate that to select the desired command the user should first click the left mouse button on **Menu1**, then within this menu click on **Menu2**, and then within **Menu2** click on **Item**. For example, **File > Exit** uses the mouse to exit from the system. Many commands can be invoked by clicking on an icon displayed in one of the toolbars. To see the command associated with an icon, position the mouse over the icon and a tooltip will appear that displays the command name.

3.1 Quartus II Online Help

Quartus II software provides comprehensive online documentation that answers many of the questions that may arise when using the software. The documentation is accessed from the **Help** menu. To get some idea of the extent of documentation provided, it is worthwhile for the reader to browse through the **Help** menu.

If no web browser is specified, Quartus will complain with an error message. To specify a web browser, go to **Tools > Options... > General > Internet Connectivity**. Specify a path to a web browser in the web browser field.

The user can quickly search through the Help topics by selecting **Help > Search**, which opens a dialog box into which keywords can be entered. Another method, context-sensitive help, is provided for quickly finding documentation for specific topics. While using most applications, pressing the F1 function key on the keyboard opens a Help display that shows the commands available for the application.

4 Starting a New Project

To start working on a new design we first have to define a new *design project*. Quartus II software makes the designer's task easy by providing support in the form of a *wizard*. Create a new project as follows:

1. Select File > New Project Wizard and click Next to reach the window in Figure 4, which asks for the name and directory of the project.

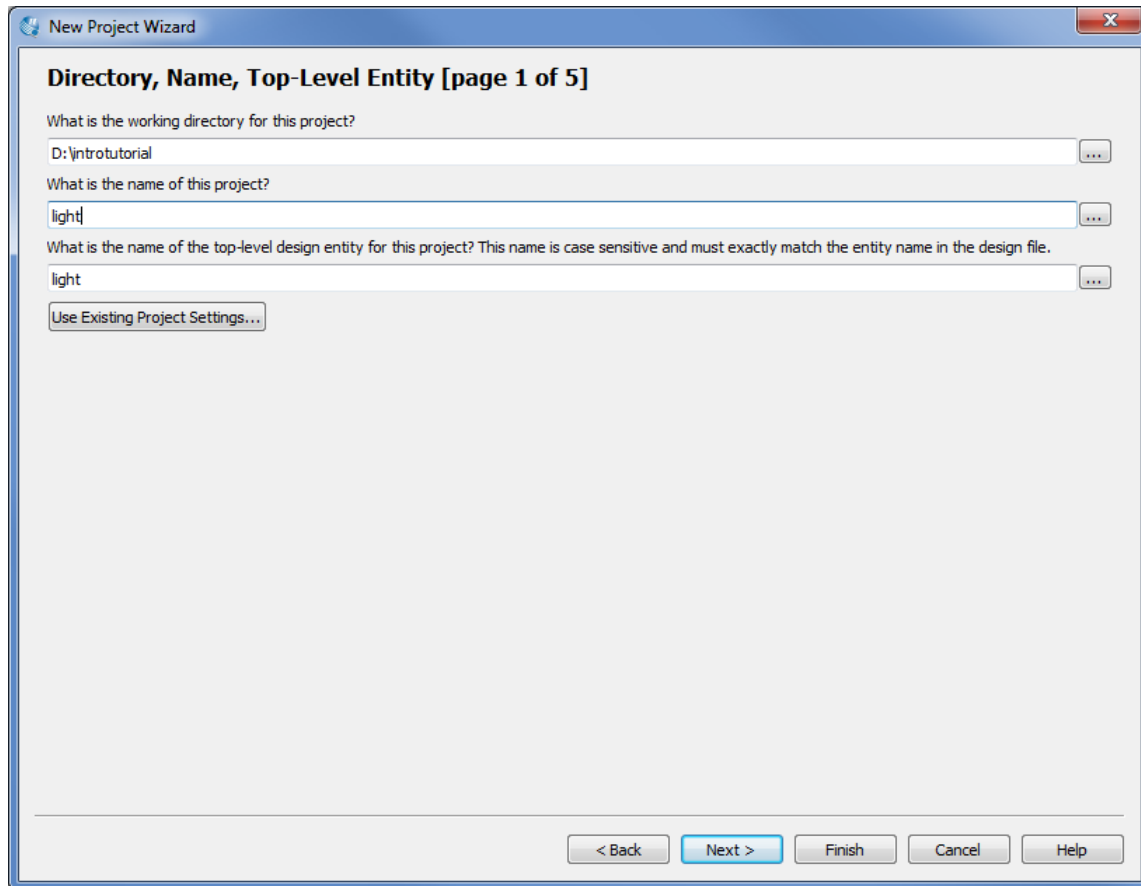


Figure 4. Creation of a new project.

2. Set the working directory to be *introtutorial*; of course, you can use some other directory name of your choice if you prefer. The project must have a name, which is usually the same as the top-level design entity that will be included in the project. Choose *light* as the name for both the project and the top-level entity, as shown in Figure 4. Press Next. Since we have not yet created the directory *introtutorial*, Quartus II software displays the pop-up box in Figure 5 asking if it should create the desired directory. Click Yes, which leads to the window in Figure 6.

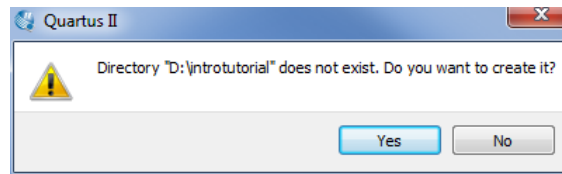


Figure 5. Quartus II software can create a new directory for the project.

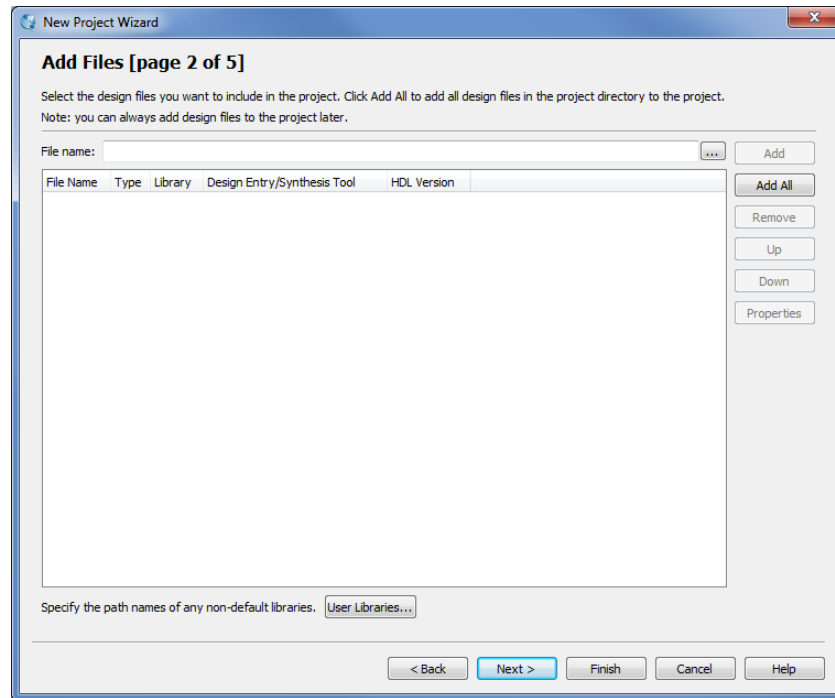


Figure 6. The wizard can include user-specified design files.

3. The wizard makes it easy to specify which existing files (if any) should be included in the project. Assuming that we do not have any existing files, click **Next**, which leads to the window in Figure 7.

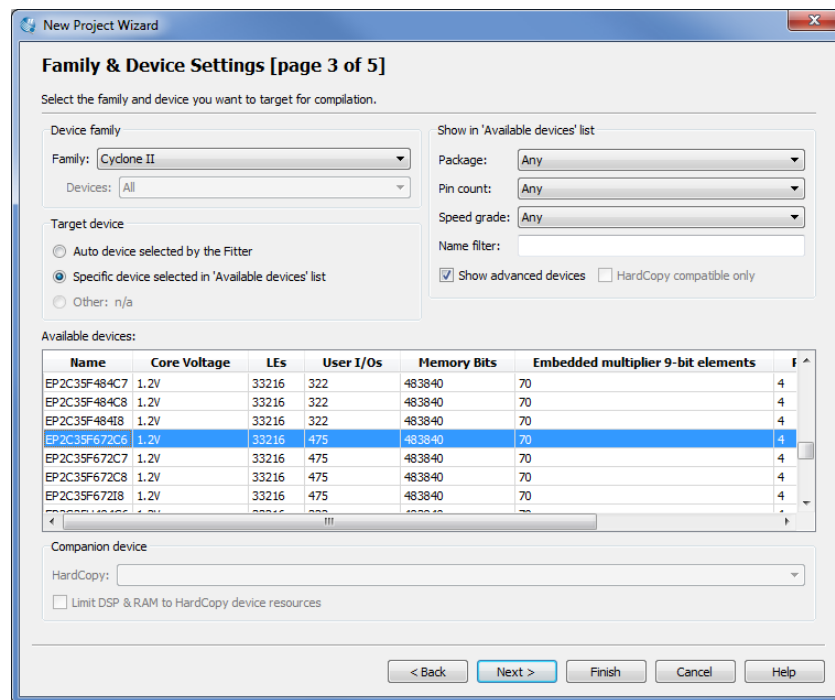


Figure 7. Choose the device family and a specific device.

4. We have to specify the type of device in which the designed circuit will be implemented. Choose the Cyclone-series device family for your DE-series board. We can let Quartus II software select a specific device in the family, or we can choose the device explicitly. We will take the latter approach. From the list of available devices, choose the appropriate device name for your DE-series board. A list of devices names on DE-series boards can be found in Table 1. Press Next, which opens the window in Figure 8.

Board	Device Name
DE0	Cyclone III EP3C16F484C6
DE0-Nano	Cyclone IVE EP4CE22F17C6
DE1	Cyclone II EP2C20F484C7
DE2	Cyclone II EP2C35F672C6
DE2-70	Cyclone II EP2C70F896C6
DE2-115	Cyclone IVE EP4CE115F29C7

Table 1. DE-series FPGA device names

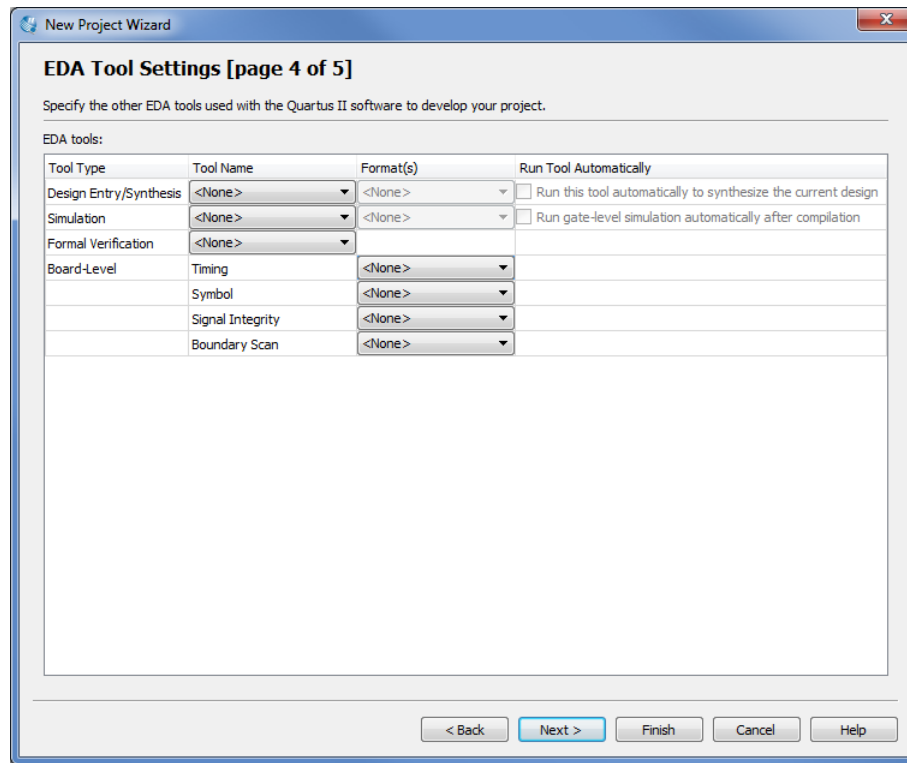


Figure 8. Other EDA tools can be specified.

5. The user can specify any third-party tools that should be used. A commonly used term for CAD software for electronic circuits is *EDA tools*, where the acronym stands for Electronic Design Automation. This term is used in Quartus II messages that refer to third-party tools, which are the tools developed and marketed by companies other than Altera. Since we will rely solely on Quartus II tools, we will not choose any other tools. Press **Next**.
6. A summary of the chosen settings appears in the screen shown in Figure 9. Press **Finish**, which returns to the main Quartus II window, but with *light* specified as the new project, in the display title bar, as indicated in Figure 10.

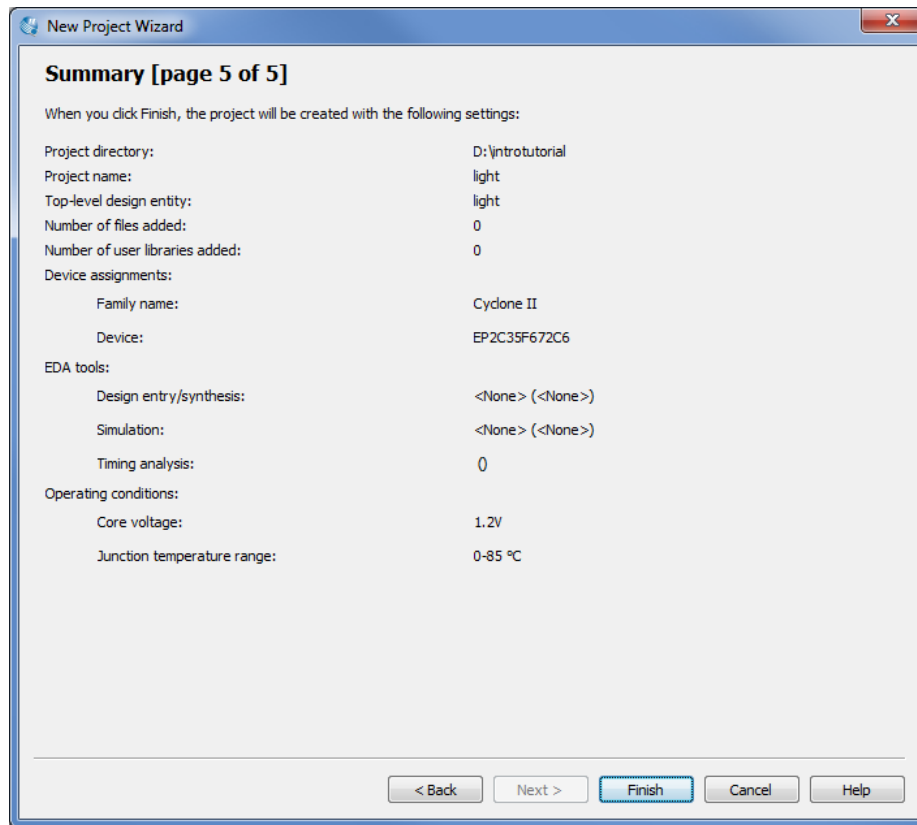


Figure 9. Example summary of a DE2 board project settings.

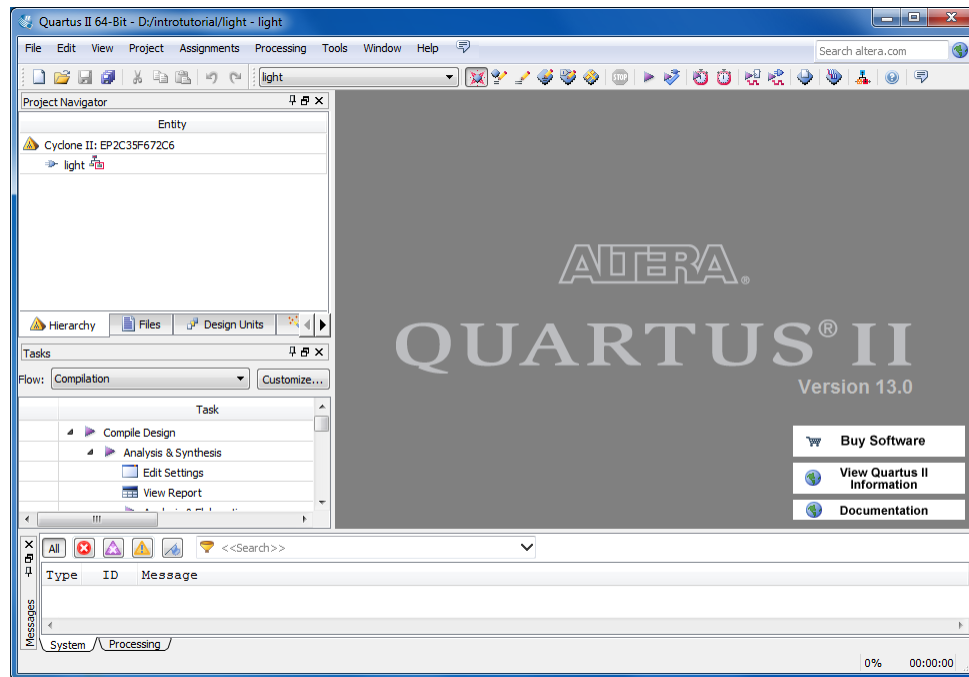


Figure 10. The Quartus II display for created project on a DE2 board.

5 Design Entry Using Verilog Code

As a design example, we will use the two-way light controller circuit shown in Figure 11. The circuit can be used to control a single light from either of the two switches, x_1 and x_2 , where a closed switch corresponds to the logic value 1. The truth table for the circuit is also given in the figure. Note that this is just the Exclusive-OR function of the inputs x_1 and x_2 , but we will specify it using the gates shown.

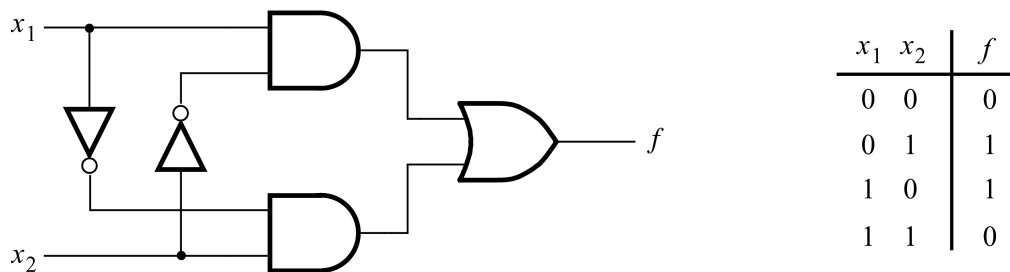


Figure 11. The light controller circuit.

The required circuit is described by the Verilog code in Figure 12. Note that the Verilog module is called *light* to match the name given in Figure 4, which was specified when the project was created. This code can be typed into a file by using any text editor that stores ASCII files, or by using the Quartus II text editing facilities. While the file

can be given any name, it is a common designers' practice to use the same name as the name of the top-level Verilog module. The file name must include the extension *v*, which indicates a Verilog file. So, we will use the name *light.v*.

```
module light (x1, x2, f);  
    input x1, x2;  
    output f;  
    assign f = (x1 & ~x2) | (~x1 & x2);  
endmodule
```

Figure 12. Verilog code for the circuit in Figure 11.

5.1 Using the Quartus II Text Editor

This section shows how to use the Quartus II Text Editor. You can skip this section if you prefer to use some other text editor to create the Verilog source code file, which we will name *light.v*.

Select **File > New** to get the window in Figure 13, choose **Verilog HDL File**, and click **OK**. This opens the Text Editor window. The first step is to specify a name for the file that will be created. Select **File > Save As** to open the pop-up box depicted in Figure 14. In the box labeled **Save as type** choose **Verilog HDL File**. In the box labeled **File name** type *light*. Put a checkmark in the box **Add file to current project**. Click **Save**, which puts the file into the directory *introtutorial* and leads to the Text Editor window shown in Figure 15. Enter the Verilog code in Figure 12 into the Text Editor and save the file by typing **File > Save**, or by typing the shortcut **Ctrl-s**.

Most of the commands available in the Text Editor are self-explanatory. Text is entered at the *insertion point*, which is indicated by a thin vertical line. The insertion point can be moved either by using the keyboard arrow keys or by using the mouse. Two features of the Text Editor are especially convenient for typing Verilog code. First, the editor can display different types of Verilog statements in different colors, which is the default choice. Second, the editor can automatically indent the text on a new line so that it matches the previous line. Such options can be controlled by the settings in **Tools > Options > Text Editor**.

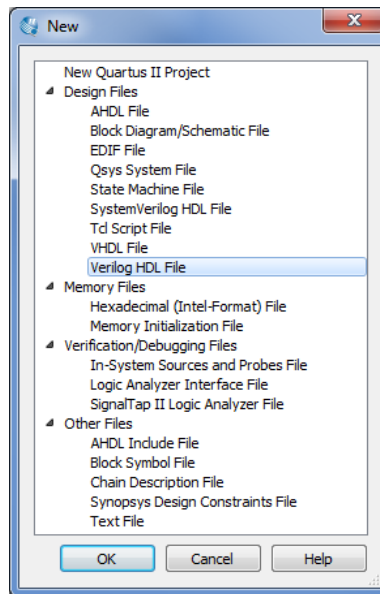


Figure 13. Choose to prepare a Verilog file.

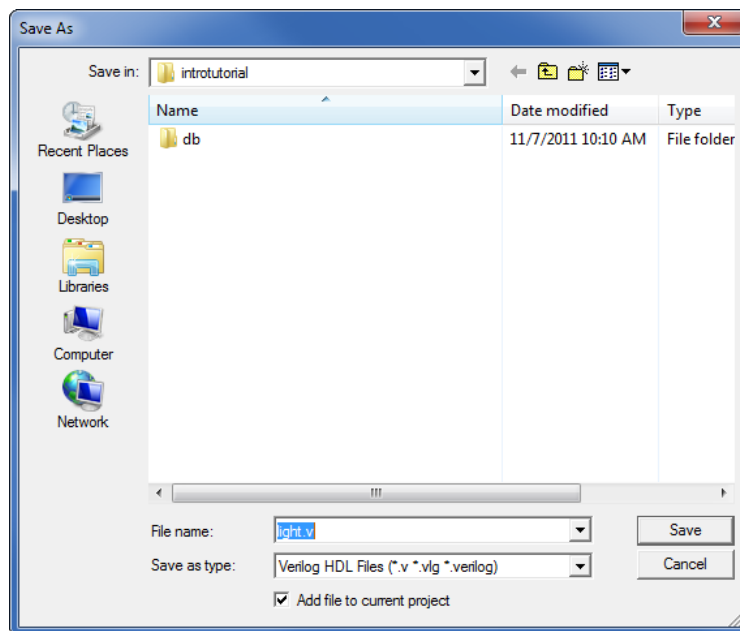


Figure 14. Name the file.

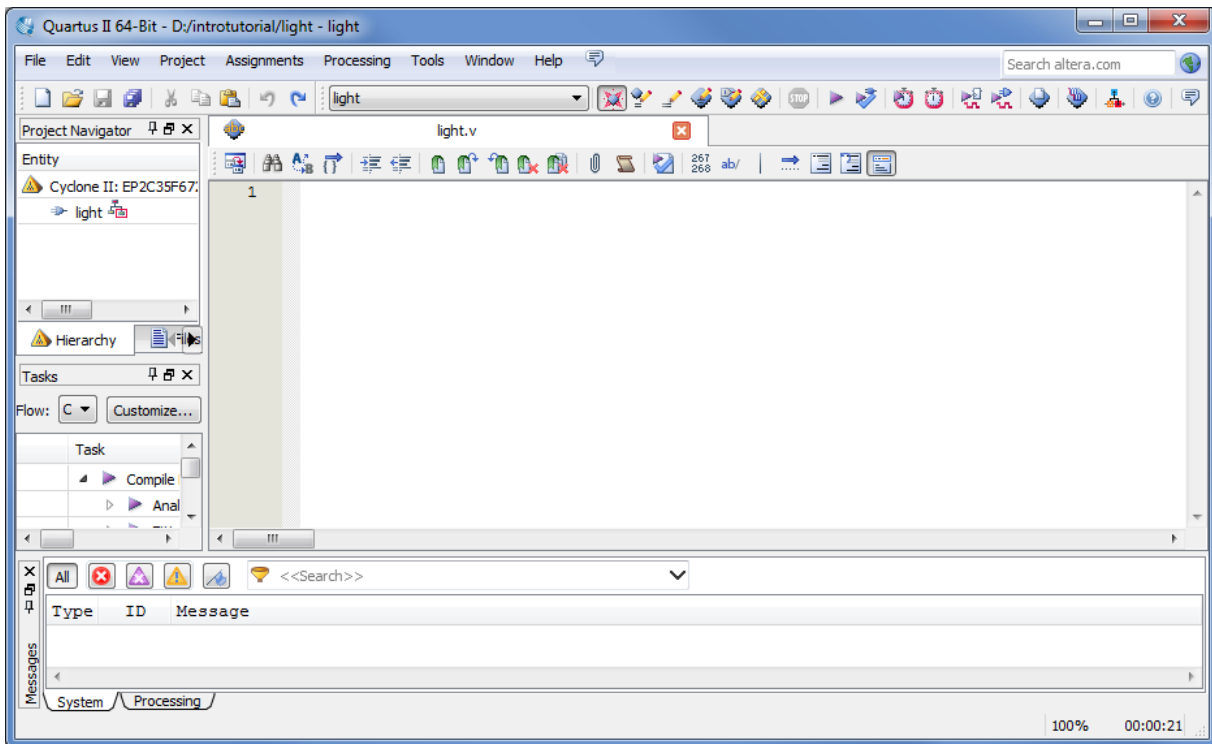


Figure 15. Text Editor window.

5.1.1 Using Verilog Templates

The syntax of Verilog code is sometimes difficult for a designer to remember. To help with this issue, the Text Editor provides a collection of *Verilog templates*. The templates provide examples of various types of Verilog statements, such as a **module** declaration, an **always** block, and assignment statements. It is worthwhile to browse through the templates by selecting **Edit > Insert Template > Verilog HDL** to become familiar with this resource.

5.2 Adding Design Files to a Project

As we indicated when discussing Figure 6, you can tell Quartus II software which design files it should use as part of the current project. To see the list of files already included in the *light* project, select **Assignments > Settings**, which leads to the window in Figure 16. As indicated on the left side of the figure, click on the item **Files**. An alternative way of making this selection is to choose **Project > Add/Remove Files in Project**.

If you used the Quartus II Text Editor to create the file and checked the box labeled **Add file to current project**, as described in Section 5.1, then the *light.v* file is already a part of the project and will be listed in the window in Figure 16. Otherwise, the file must be added to the project. So, if you did not use the Quartus II Text Editor, then place a copy of the file *light.v*, which you created using some other text editor, into the directory *introtutorial*. To add this file to the project, click on the ... button next to the box labelled **File name** in Figure 16 to get the pop-up window in Figure 17. Select the *light.v* file and click **Open**. The selected file is now indicated in the **File name** box

in Figure 16. Click Add then OK to include the *light.v* file in the project. We should mention that in many cases the Quartus II software is able to automatically find the right files to use for each entity referenced in Verilog code, even if the file has not been explicitly added to the project. However, for complex projects that involve many files it is a good design practice to specifically add the needed files to the project, as described above.

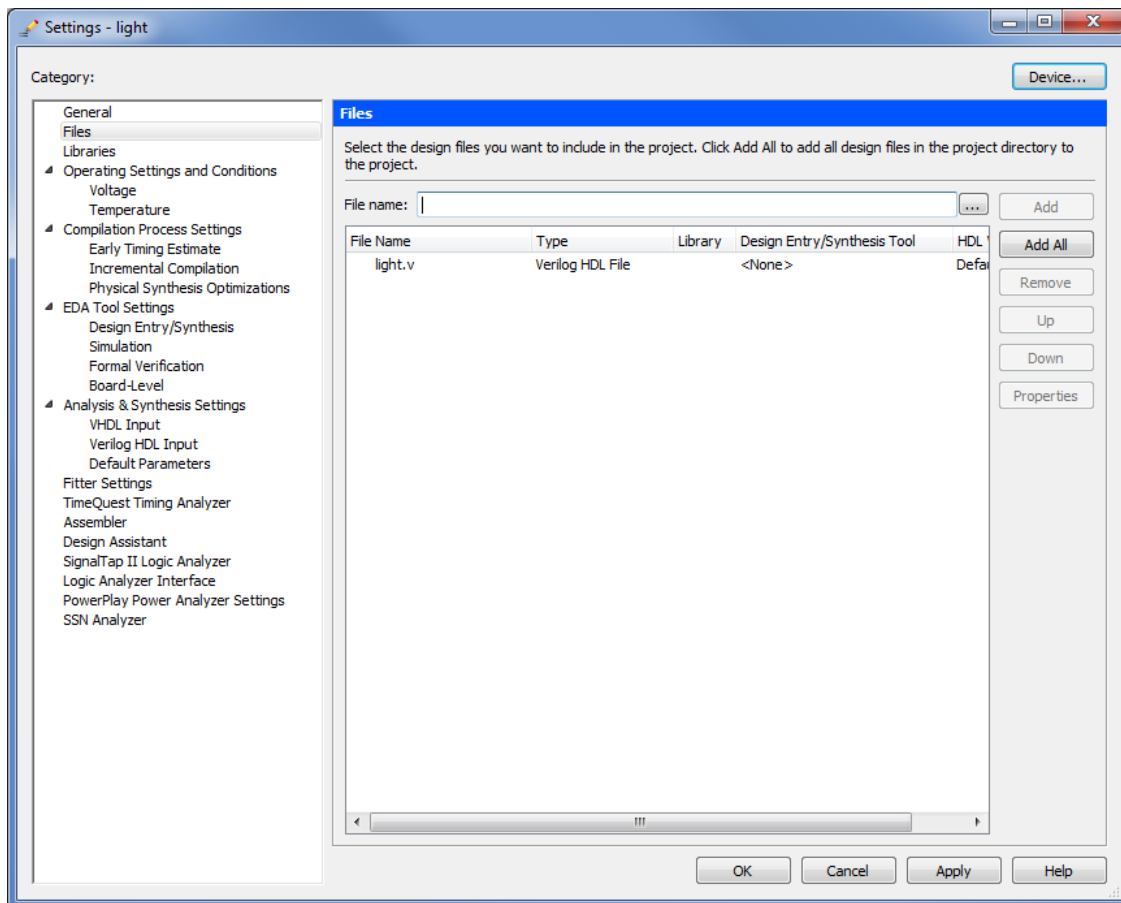


Figure 16. Settings window.

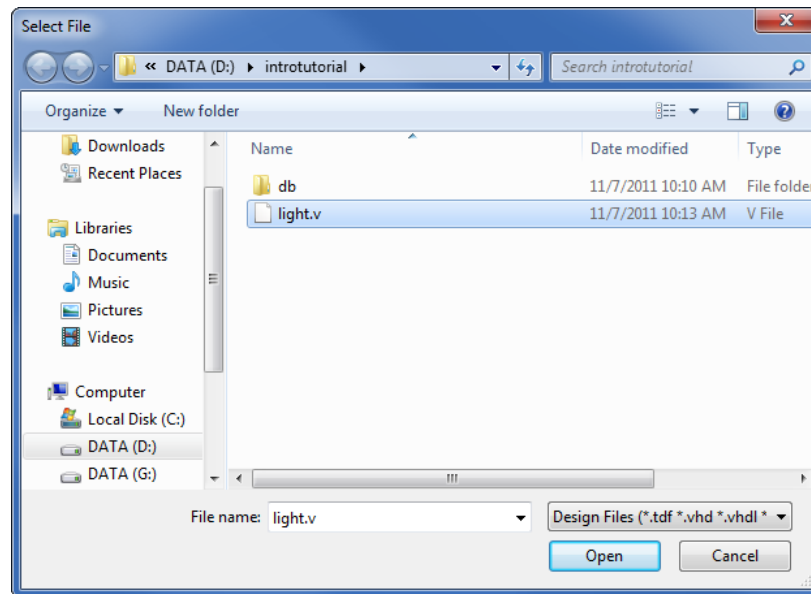



Figure 17. Select the file.

6 Compiling the Designed Circuit

The Verilog code in the file *light.v* is processed by several Quartus II tools that analyze the code, synthesize the circuit, and generate an implementation of it for the target chip. These tools are controlled by the application program called the *Compiler*.

Run the Compiler by selecting **Processing > Start Compilation**, or by clicking on the toolbar icon  that looks like a purple triangle. Your project must be saved before compiling. As the compilation moves through various stages, its progress is reported in a window on the left side of the Quartus II display. Successful (or unsuccessful) compilation is indicated in a pop-up box. Acknowledge it by clicking **OK**, which leads to the Quartus II display in Figure 18. In the message window, at the bottom of the figure, various messages are displayed. In case of errors, there will be appropriate messages given.

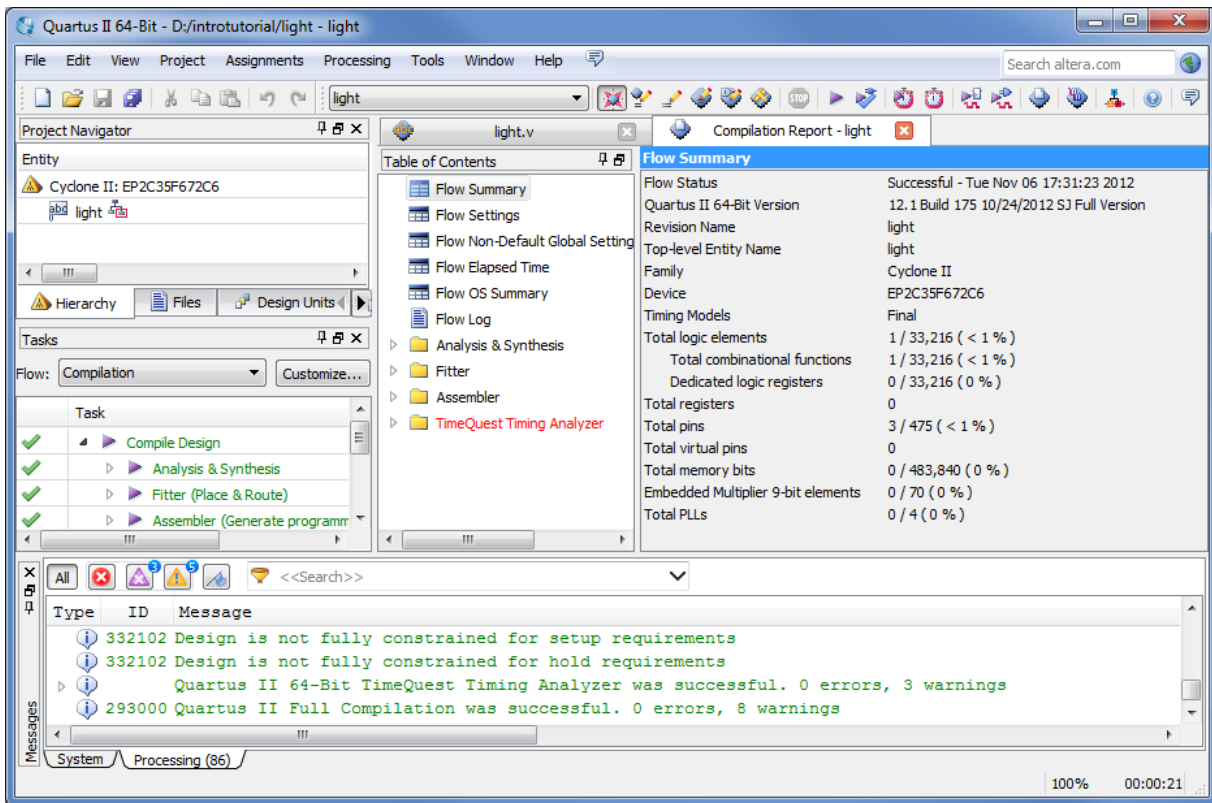




Figure 18. Display after a successful compilation.

When the compilation is finished, a compilation report is produced. A tab showing this report is opened automatically, as seen in Figure 18. The tab can be closed in the normal way, and it can be opened at any time either by selecting **Processing > Compilation Report** or by clicking on the icon . The report includes a number of sections listed on the left side. Figure 18 displays the Compiler Flow Summary section, which indicates that only one logic element and three pins are needed to implement this tiny circuit on the selected FPGA chip.

6.1 Errors

Quartus II software displays messages produced during compilation in the Messages window. If the Verilog design file is correct, one of the messages will state that the compilation was successful and that there are no errors.

If the Compiler does not report zero errors, then there is at least one mistake in the Verilog code. In this case a message corresponding to each error found will be displayed in the Messages window. Double-clicking on an error message will highlight the offending statement in the Verilog code in the Text Editor window. Similarly, the Compiler may display some warning messages. Their details can be explored in the same way as in the case of error messages. The user can obtain more information about a specific error or warning message by selecting the message and pressing the F1 function key.

To see the effect of an error, open the file *light.v*. Remove the semicolon in the **assign** statement, illustrating a typographical error that is easily made. Compile the erroneous design file by clicking on the  icon. A pop-up box will ask if the changes made to the *light.v* file should be saved; click **Yes**. After trying to compile the circuit, Quartus II software will display a pop-up box indicating that the compilation was not successful. Acknowledge it by clicking **OK**. The compilation report summary, given in Figure 19, now confirms the failed result. In the Table of Contents panel, expand the Analysis & Synthesis part of the report and then select **Messages** to have the messages displayed as shown in Figure 20. The Compilation Report can be displayed as a separate window as in Figure 20 by right-clicking its tab and selecting **Detach Window**, and can be reattached by clicking **Window > Attach Window**. Double-click on the first error message. Quartus II software responds by opening the *light.v* file and highlighting the statement which is affected by the error, as shown in Figure 21. Correct the error and recompile the design.

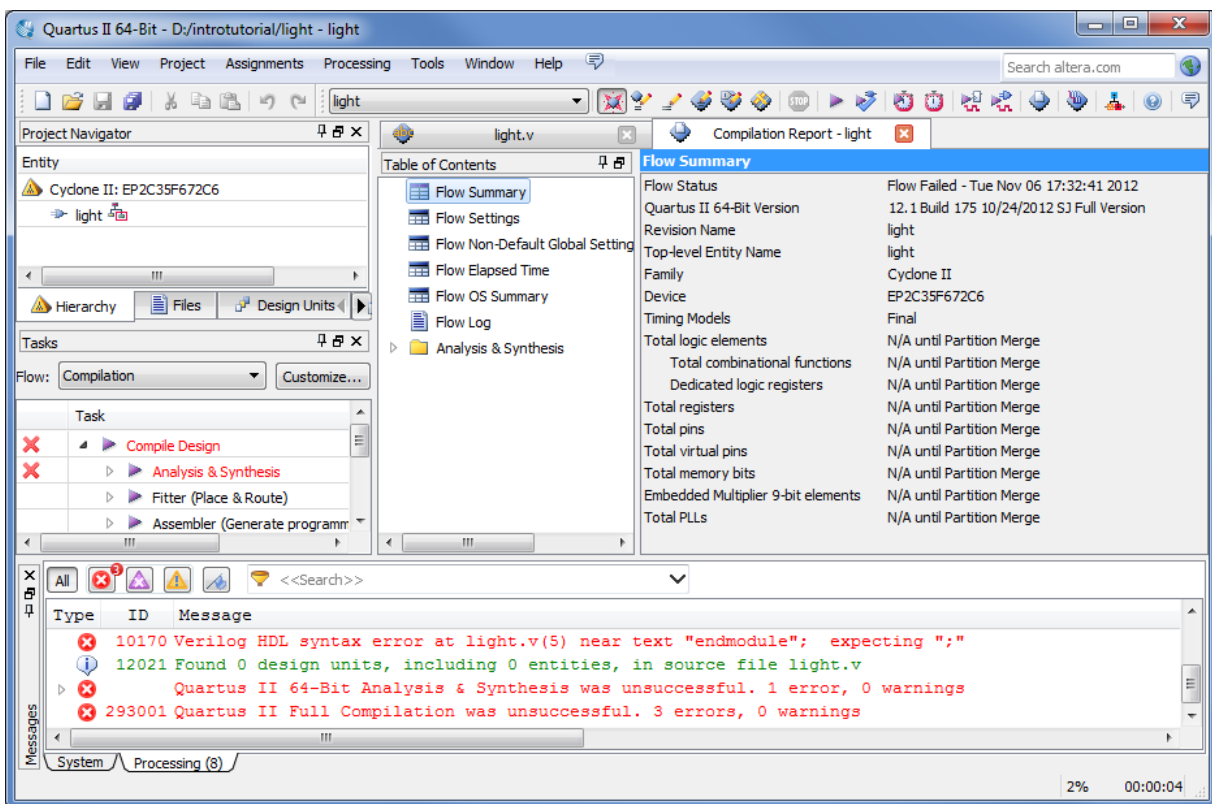


Figure 19. Compilation report for the failed design.

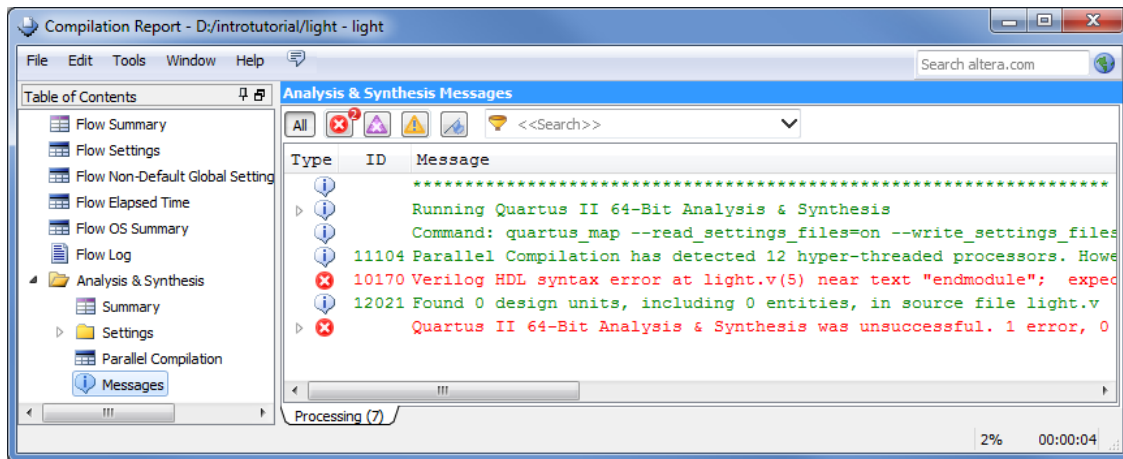


Figure 20. Error messages.

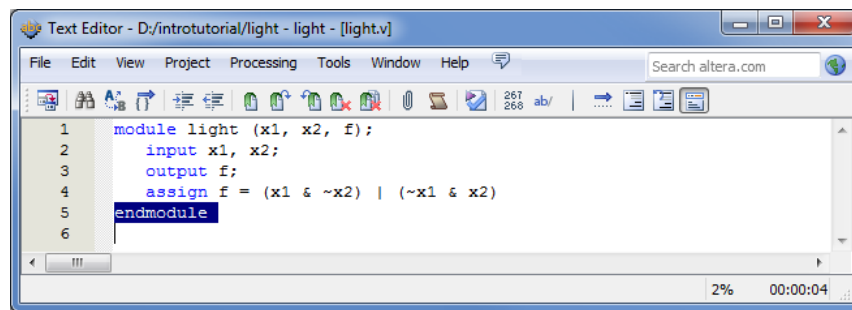


Figure 21. Identifying the location of the error.

7 Pin Assignment

During the compilation above, the Quartus II Compiler was free to choose any pins on the selected FPGA to serve as inputs and outputs. However, the DE-series board has hardwired connections between the FPGA pins and the other components on the board. We will use two toggle switches, labeled SW_0 and SW_1 , to provide the external inputs, x_1 and x_2 , to our example circuit. These switches are connected to the FPGA pins listed in Table 2. We will connect the output f to the green light-emitting diode labeled $LEDG_0$. Its FPGA pin assignment can also be found in Table 2.

Component	DE0	DE0-Nano	DE1	DE2	DE2-70	DE2-115
SW_0	PIN_J6	PIN_M1	PIN_L22	PIN_N25	PIN_AA23	PIN_AB28
SW_1	PIN_H5	PIN_T8	PIN_L21	PIN_N26	PIN_AB26	PIN_AC28
$LEDG_0$	PIN_J1	PIN_A15	PIN_U22	PIN_AE22	PIN_W27	PIN_E21

Table 2. DE-Series Pin Assignments

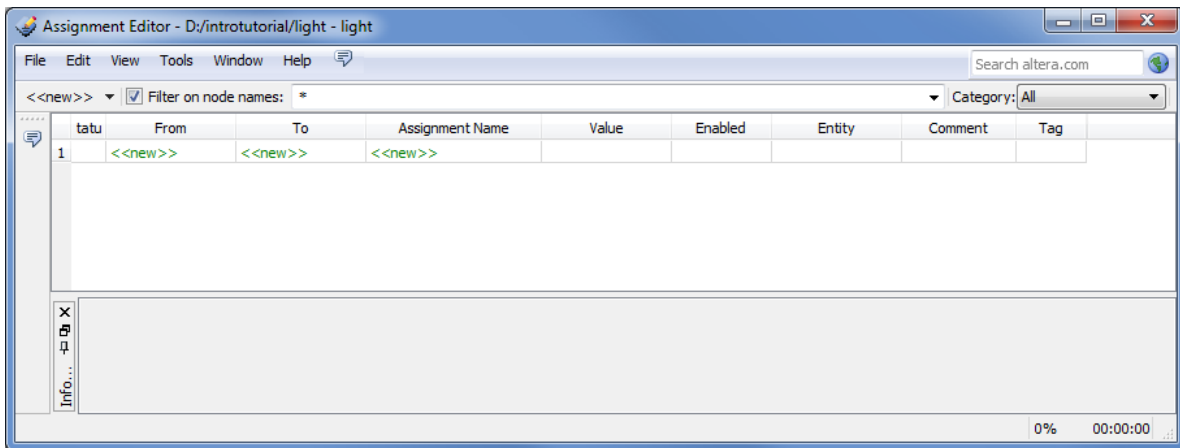




Figure 22. The Assignment Editor window.

Pin assignments are made by using the Assignment Editor. Select **Assignments > Assignment Editor** to reach the window in Figure 22 (shown here as a detached window). In the **Category** drop-down menu select **All**. Click on the **<<new>>** button located near the top left corner to make a new item appear in the table. Double click the box under the column labeled **To** so that the Node Finder button  appears. Click on the button (not the drop down arrow) to reach the window in Figure 23. Click on  to show more search options. In the **Filter** drop-down menu select **Pins: all**. Then click the **List** button to display the input and output pins to be assigned: f , $x1$, and $x2$. Click on $x1$ as the first pin to be assigned and click the **>** button; this will enter $x1$ in the **Selected Nodes** box. Click **OK**. $x1$ will now appear in the box under the column labeled **To**. Alternatively, the node name can be entered directly by double-clicking the box under the **To** column and typing in the node name.

Follow this by double-clicking on the box to the right of this new $x1$ entry, in the column labeled **Assignment Name**. Now, the drop-down menu in Figure 24 appears. Scroll down and select **Location (Accepts wildcards/groups)**. Instead of scrolling down the menu to find the desired item, you can just type the first letter of the item in the **Assignment Name** box. In this case the desired item happens to be the first item beginning with **L**. Finally, double-click the box in the column labeled **Value**. Type the pin assignment corresponding to SW_0 for your DE-series board, as listed in Table 2.

Use the same procedure to assign input $x2$ and output f to the appropriate pins listed in Table 2. An example using a DE2 board is shown in Figure 25. To save the assignments made, choose **File > Save**. You can also simply close the Assignment Editor window, in which case a pop-up box will ask if you want to save the changes to assignments; click **Yes**. Recompile the circuit, so that it will be compiled with the correct pin assignments.

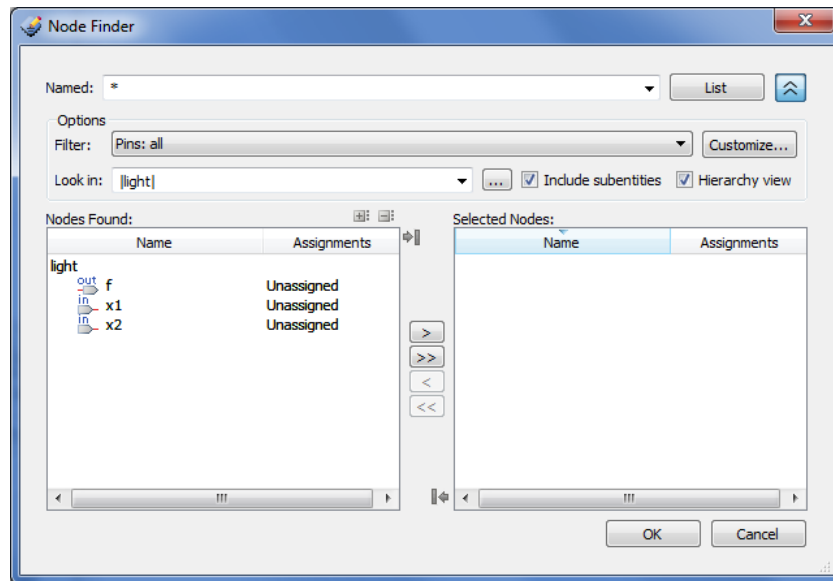


Figure 23. The Node Finder displays the input and output names.

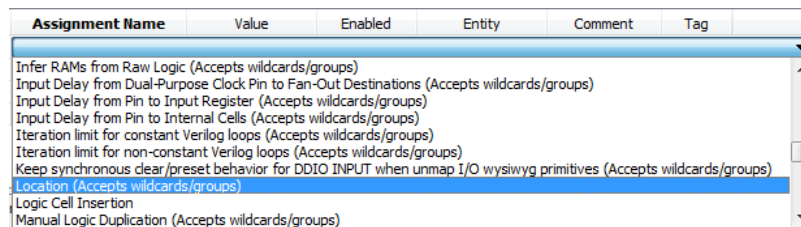


Figure 24. The available assignment names for a DE2 board.

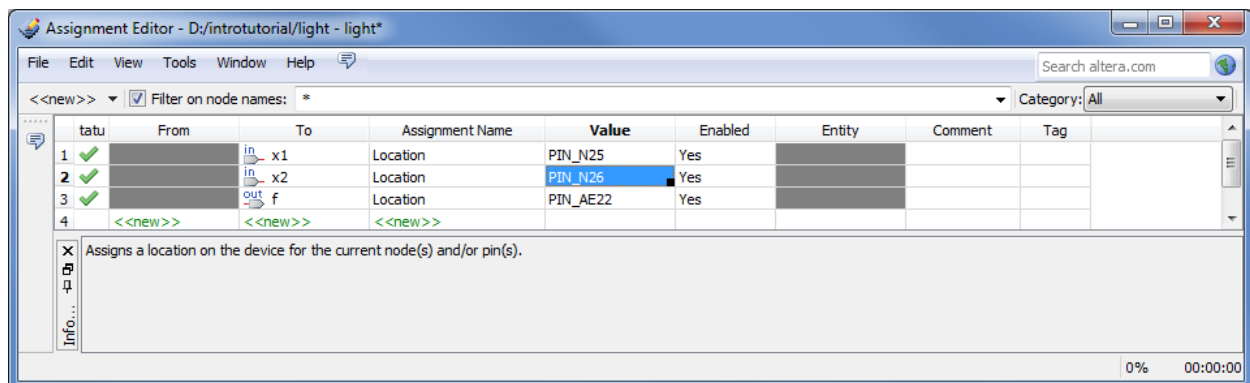


Figure 25. The complete assignment on a DE2 board.

The DE-series board has fixed pin assignments. Having finished one design, the user will want to use the same pin assignment for subsequent designs. Going through the procedure described above becomes tedious if there are many pins used in the design. A useful Quartus II feature allows the user to both export and import the pin assignments from a special file format, rather than creating them manually using the Assignment Editor. A simple file format that can be used for this purpose is the *Quartus II Settings File (QSF)* format. The format for the file for our simple project (on a DE2 board) is

```
set_location_assignment PIN_N25 -to x1
set_location_assignment PIN_N26 -to x2
set_location_assignment PIN_AE22 -to f
```

By adding lines to the file, any number of pin assignments can be created. Such *qsf* files can be imported into any design project.

If you created a pin assignment for a particular project, you can export it for use in a different project. To see how this is done, open again the Assignment Editor to reach the window in Figure 25. Select **Assignments > Export Assignment** which leads to the window in Figure 26. Here, the file *light.qsf* is available for export. Click on OK. If you now look in the directory, you will see that the file *light.qsf* has been created.

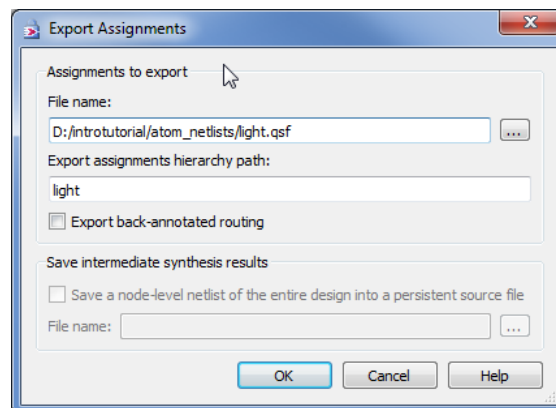


Figure 26. Exporting the pin assignment.

You can import a pin assignment by choosing **Assignments > Import Assignments**. This opens the dialogue in Figure 27 to select the file to import. Type the name of the file, including the *qsf* extension and the full path to the directory that holds the file, in the File Name box and press OK. Of course, you can also browse to find the desired file.

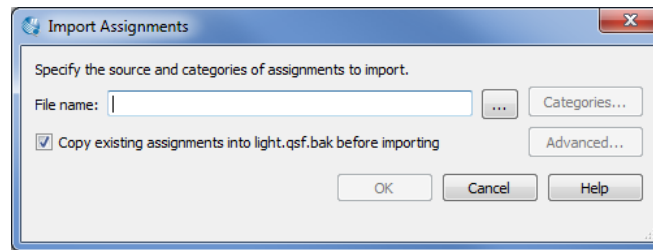


Figure 27. Importing the pin assignment.

For convenience when using large designs, all relevant pin assignments for the DE-series board are given in individual files. For example, the DE2 pin assignments can be found in the *DE2_pin_assignments.qsf* file, in the directory *tutorials\design_files*, which is included on the CD-ROM that accompanies the DE-series board and can also be found on Altera's DE-series web pages. This file uses the names found in the *DE2 User Manual*. If we wanted to make the pin assignments for our example circuit by importing this file, then we would have to use the same names in our Verilog design file; namely, *SW[0]*, *SW[1]* and *LEDG[0]* for *x1*, *x2* and *f*, respectively. Since these signals are specified in the *qsf* file as elements of vectors *SW* and *LEDG*, we must refer to them in the same way in the Verilog design file. For example, in the *qsf* file the 18 toggle switches are called *SW[17]* to *SW[0]*. In Verilog code, they can also be referred to as a vector *SW[17:0]*.

8 Simulating the Designed Circuit

Before implementing the designed circuit in the FPGA chip on the DE-series board, it is prudent to simulate it to ascertain its correctness. Quartus II's Simulation Waveform Editor tool can be used to simulate the behavior of a designed circuit. Before the circuit can be simulated, it is necessary to create the desired waveforms, called *test vectors*, to represent the input signals. It is also necessary to specify which outputs, as well as possible internal points in the circuit, the designer wishes to observe. The simulator applies the test vectors to a model of the implemented circuit and determines the expected response. We will use the Simulation Waveform Editor to draw the test vectors, as follows:

1. In the main Quartus II window, select **File > New > Verification/Debugging Files > University Program VWF** to open the Simulation Waveform Editor.
2. The Simulation Waveform Editor window is depicted in Figure 28. Save the file under the name *light.vwf*; note that this changes the name in the displayed window. Set the desired simulation to run from 0 to 200 ns by selecting **Edit > Set End Time** and entering 200 ns in the dialog box that pops up. Selecting **View > Fit in Window** displays the entire simulation range of 0 to 200 ns in the window, as shown in Figure 29. You may wish to resize the window to its maximum size.

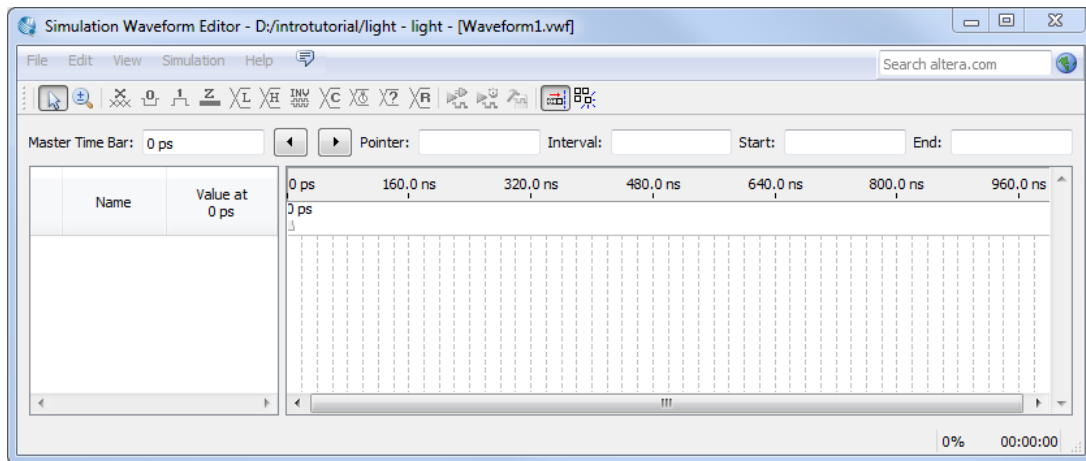


Figure 28. The Waveform Editor window.

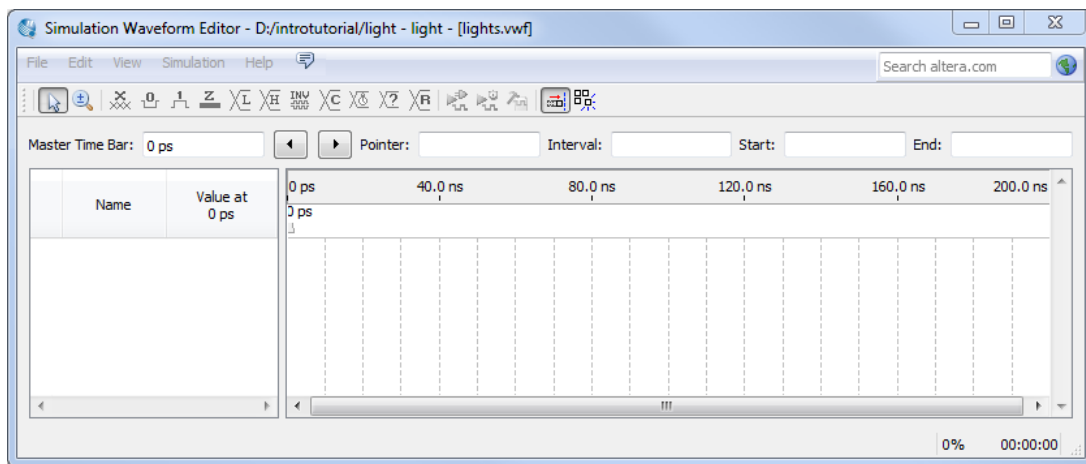


Figure 29. The augmented Waveform Editor window.

3. Next, we want to include the input and output nodes of the circuit to be simulated. Click **Edit > Insert > Insert Node or Bus** to open the window in Figure 30. It is possible to type the name of a signal (pin) into the Name box, or use the Node Finder to search your project for the signals. Click on the button labeled **Node Finder** to open the window in Figure 31. The Node Finder utility has a filter used to indicate what type of nodes are to be found. Since we are interested in input and output pins, set the filter to **Pins: all**. Click the **List** button to find the input and output nodes as indicated on the left side of the figure.

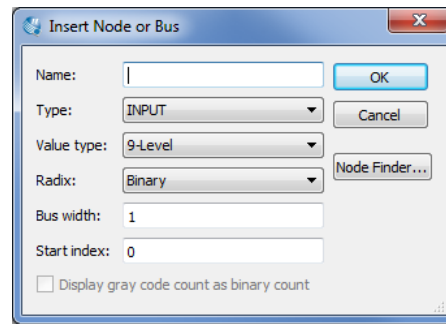


Figure 30. The Insert Node or Bus dialogue.

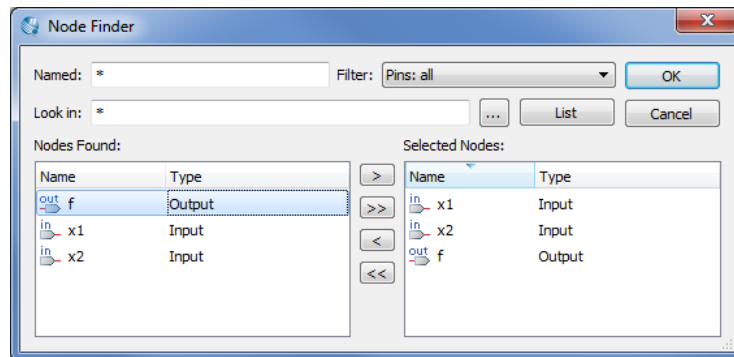


Figure 31. Selecting nodes to insert into the Waveform Editor.

Click on the *x1* signal in the Nodes Found box in Figure 31, and then click the > sign to add it to the Selected Nodes box on the right side of the figure. Do the same for *x2* and *f*. Click OK to close the Node Finder window, and then click OK in the window of Figure 30. This leaves a fully displayed Waveform Editor window, as shown in Figure 32. If you did not select the nodes in the same order as displayed in Figure 32, it is possible to rearrange them. To move a waveform up or down in the Waveform Editor window, click within the node's row (i.e. on its name, icon, or value) and drag it up or down in the Waveform Editor.

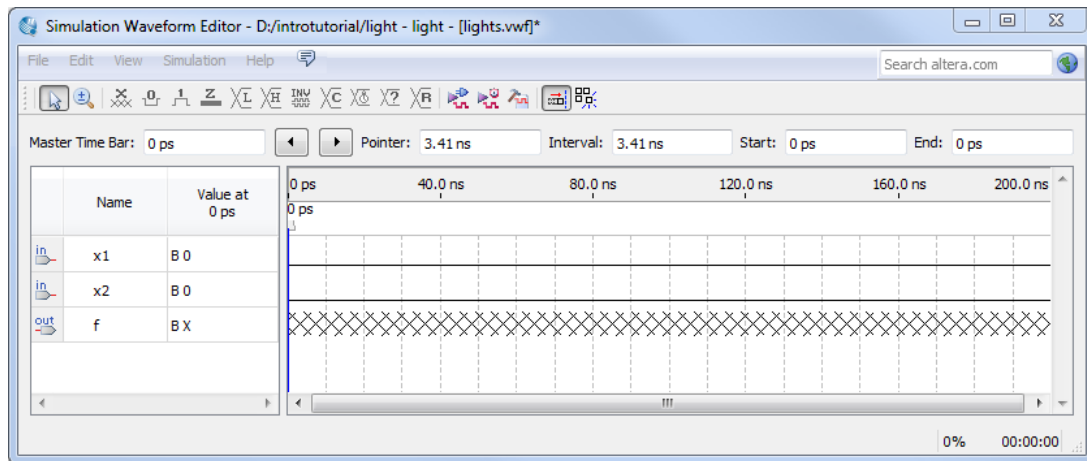



Figure 32. The nodes needed for simulation.

- We will now specify the logic values to be used for the input signals *x1* and *x2* during simulation. The logic values at the output *f* will be generated automatically by the simulator. To make it easy to draw the desired waveforms, the Waveform Editor displays (by default) vertical guidelines and provides a drawing feature that snaps on these lines (which can otherwise be invoked by choosing **Edit > Snap to Grid**). Observe also a solid vertical line, which can be moved by pointing to its top and dragging it horizontally. This reference line is used in analyzing the timing of a circuit; move it to the *time* = 0 position. The waveforms can be drawn using the Selection Tool, which is activated by selecting the icon  in the toolbar.

To simulate the behavior of a large circuit, it is necessary to apply a sufficient number of input valuations and observe the expected values of the outputs. In a large circuit the number of possible input valuations may be huge, so in practice we choose a relatively small (but representative) sample of these input valuations. However, for our tiny circuit we can simulate all four input valuations given in Figure 11. We will use four 50-ns time intervals to apply the four test vectors.

We can generate the desired input waveforms as follows. Click on the waveform for the *x1* node. Once a waveform is selected, the editing commands in the Waveform Editor can be used to draw the desired waveforms. Commands are available for setting a selected signal to 0, 1, unknown (X), high impedance (Z), weak low (L), weak high (H), a count value (C), an arbitrary value, a random value (R), inverting its existing value (INV), or defining a clock waveform. Each command can be activated by using the **Edit > Value** command, or via the toolbar for the Waveform Editor. The Value menu can also be opened by right-clicking on a selected waveform.

Set *x1* to 0 in the time interval 0 to 100 ns, which is probably already set by default. Next, set *x1* to 1 in the time interval 100 to 200 ns. Do this by pressing the mouse at the start of the interval and dragging it to its end, which highlights the selected interval, and choosing the logic value 1 in the toolbar. Make *x2* = 1 from 50 to 100 ns and also from 150 to 200 ns, which corresponds to the truth table in Figure 11. This should produce the image in Figure 33. Observe that the output *f* is displayed as having an unknown value at this time, which is indicated by a hashed pattern; its value will be determined during simulation. Save the file.

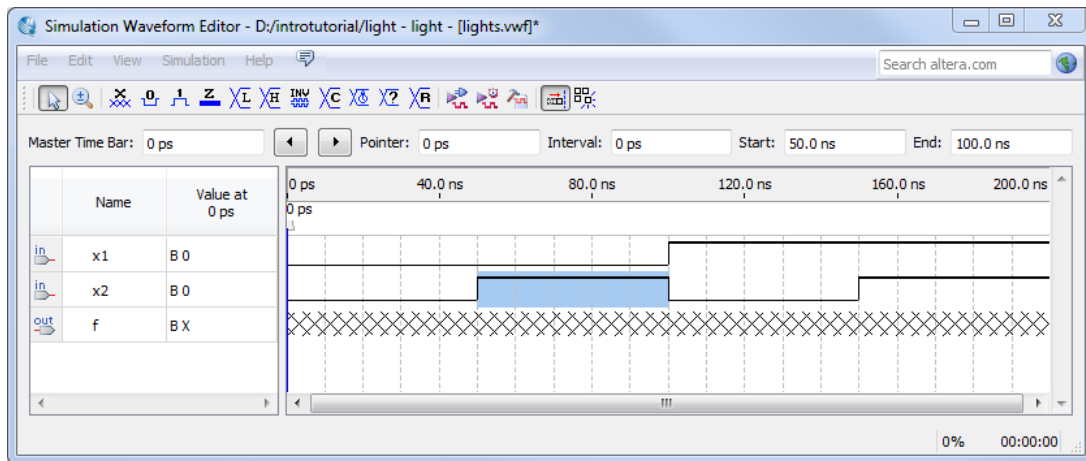




Figure 33. Setting of test values.

8.1 Performing the Simulation

A designed circuit can be simulated in two ways. The simplest way is to assume that logic elements and interconnection wires in the FPGA are perfect, thus causing no delay in propagation of signals through the circuit. This is called *functional simulation*. A more complex alternative is to take all propagation delays into account, which leads to *timing simulation*. Typically, functional simulation is used to verify the functional correctness of a circuit as it is being designed.

8.1.1 Functional Simulation

Before running a functional simulation it is necessary to run Analysis and Synthesis on your design by selecting the  icon in the main Quartus II window. Note that Analysis and Synthesis gets run as a part of the main compilation flow. If you compiled your design in Section 6, then it is not necessary to run Analysis and Synthesis again.

To perform the functional simulation, select **Simulation > Run Functional Simulation** or select the  icon. A pop-up window will show the progress of the simulation then automatically close when it is complete. At the end of the simulation, a second Waveform Editor window will open the results of the simulation as illustrated in Figure 34. Observe that the output *f* is as specified in the truth table of Figure 11.

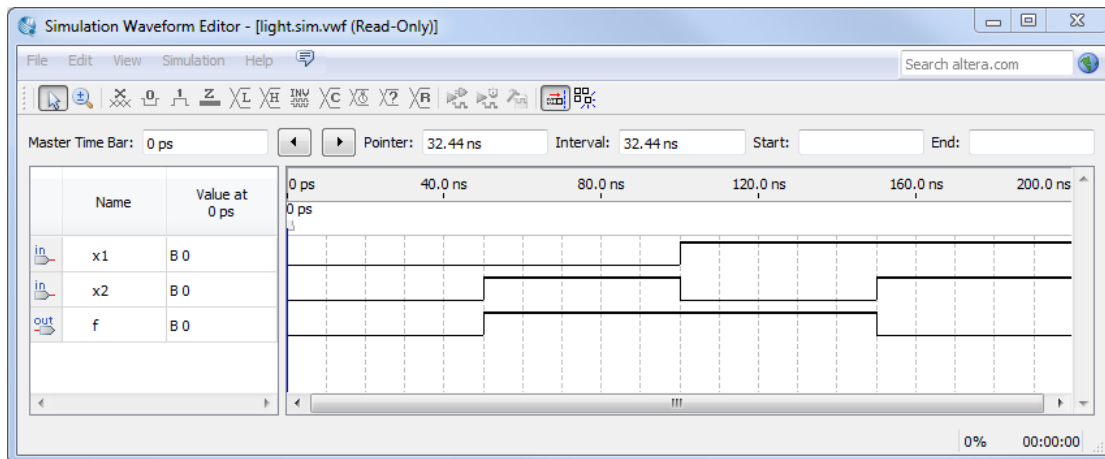




Figure 34. The result of functional simulation.

8.1.2 Timing Simulation

Having ascertained that the designed circuit is functionally correct, we should now perform the timing simulation to see how it will behave when it is actually implemented in the chosen FPGA device. Before running a timing simulation, it is necessary to compile your design by selecting the  icon in the main Quartus II window. Unlike functional simulations, timing simulations require the full compilation of your design, not just Analysis and Synthesis.

To perform the timing simulation, select **Simulation > Run Timing Simulation** or select the  icon. The simulation should produce the waveforms in Figure 35. Observe that there is a delay of about 5 ns in producing a change in the signal f from the time when the input signals, x_1 and x_2 , change their values. This delay is due to the propagation delays in the logic element and the wires in the FPGA device.

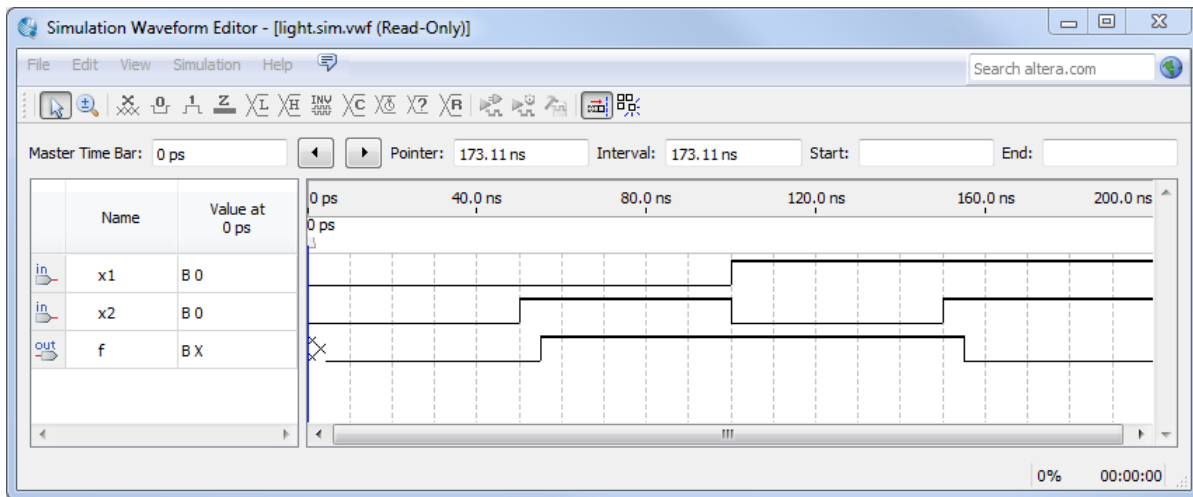


Figure 35. The result of timing simulation.

9 Programming and Configuring the FPGA Device

The FPGA device must be programmed and configured to implement the designed circuit. The required configuration file is generated by the Quartus II Compiler's Assembler module. Altera's DE-series board allows the configuration to be done in two different ways, known as JTAG and AS modes. The configuration data is transferred from the host computer (which runs the Quartus II software) to the board by means of a cable that connects a USB port on the host computer to the leftmost USB connector on the board. To use this connection, it is necessary to have the USB-Blaster driver installed. If this driver is not already installed, consult the tutorial *Getting Started with Altera's DE-Series Boards* for information about installing the driver. Before using the board, make sure that the USB cable is properly connected and turn on the power supply switch on the board.

In the JTAG mode, the configuration data is loaded directly into the FPGA device. The acronym JTAG stands for Joint Test Action Group. This group defined a simple way for testing digital circuits and loading data into them, which became an IEEE standard. If the FPGA is configured in this manner, it will retain its configuration as long as the power remains turned on. The configuration information is lost when the power is turned off. The second possibility is to use the Active Serial (AS) mode. In this case, a configuration device that includes some flash memory is used to store the configuration data. Quartus II software places the configuration data into the configuration device on the DE-series board. Then, this data is loaded into the FPGA upon power-up or reconfiguration. Thus, the FPGA need not be configured by the Quartus II software if the power is turned off and on. The choice between the two modes is made by the RUN/PROG switch on the DE-series board. The RUN position selects the JTAG mode, while the PROG position selects the AS mode.

9.1 JTAG Programming

The programming and configuration task is performed as follows. Flip the RUN/PROG switch into the RUN position. Select Tools > Programmer to reach the window in Figure 36. Here it is necessary to specify the programming

hardware and the mode that should be used. If not already chosen by default, select JTAG in the Mode box. Also, if the USB-Blaster is not chosen by default, press the **Hardware Setup...** button and select the USB-Blaster in the window that pops up, as shown in Figure 37.

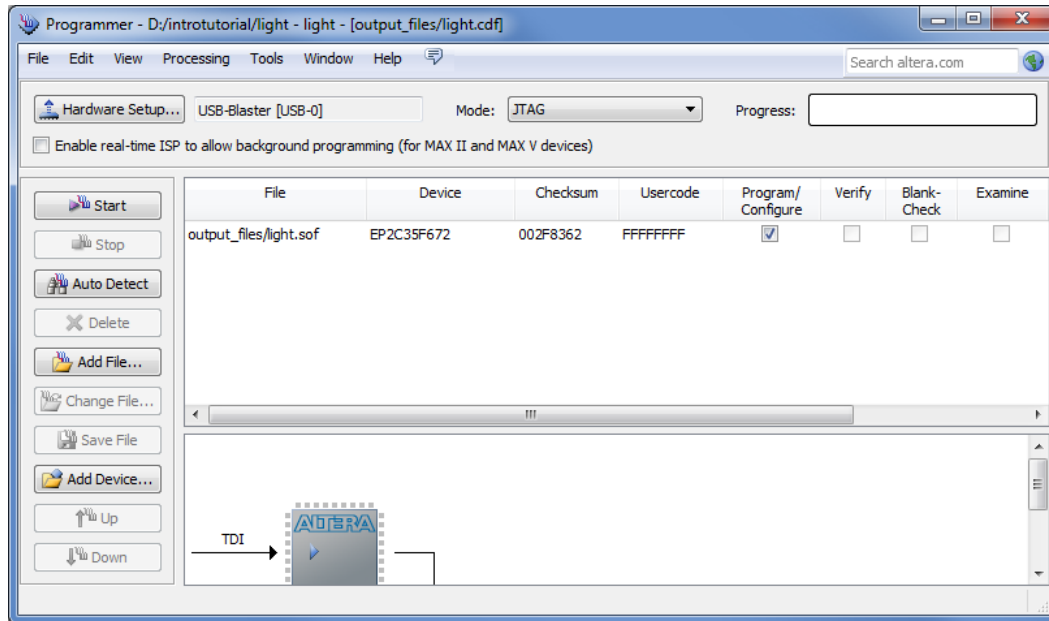


Figure 36. The Programmer window.

Observe that the configuration file *light.sof* is listed in the window in Figure 36. If the file is not already listed, then click **Add File** and select it. This is a binary file produced by the Compiler's Assembler module, which contains the data needed to configure the FPGA device. The extension *.sof* stands for SRAM Object File. Note also that the device selected is EP2C35F672, which is the FPGA device used on the DE2 board. Click on the **Program/Configure** check box, as shown in Figure 38.

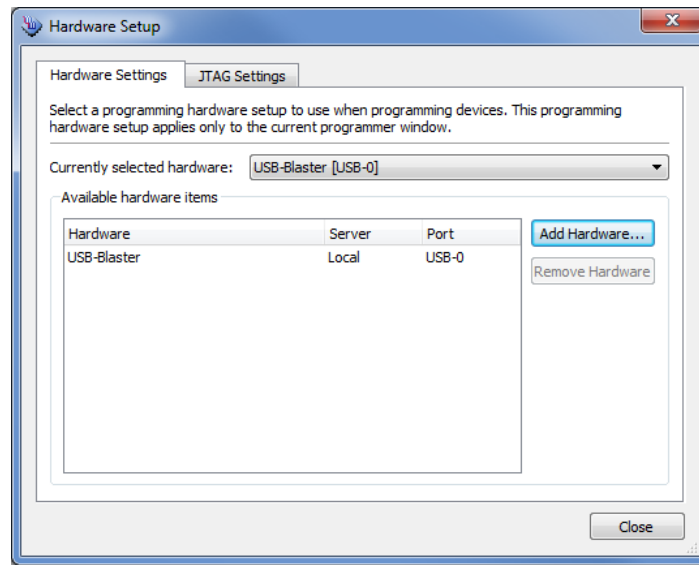


Figure 37. The Hardware Setup window.

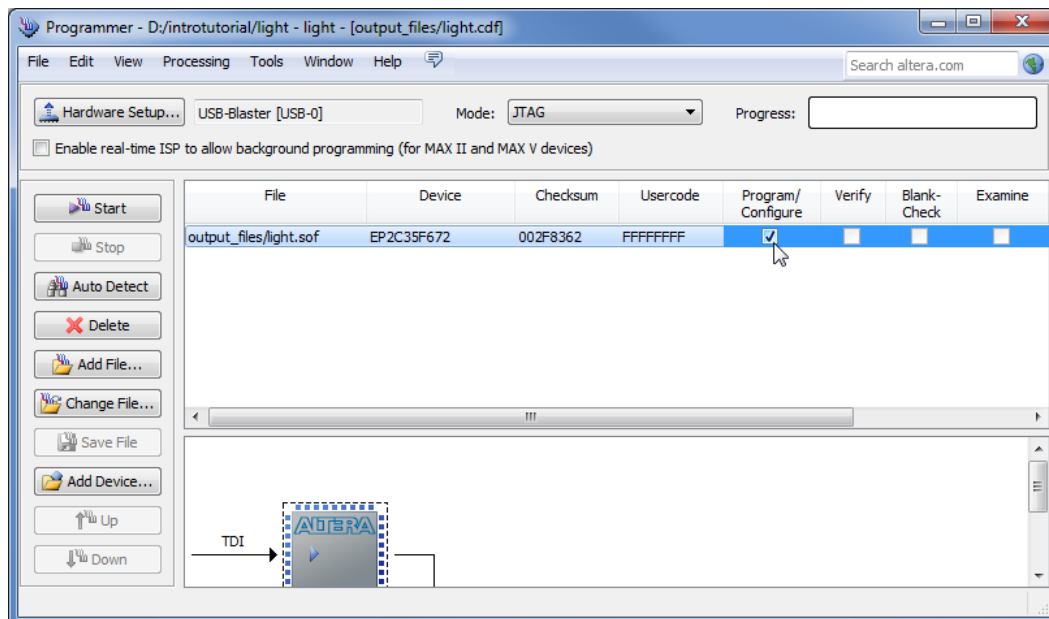


Figure 38. The updated Programmer window.

Now, press **Start** in the window in Figure 38. An LED on the board will light up when the configuration data has been downloaded successfully. If you see an error reported by Quartus II software indicating that programming failed, then check to ensure that the board is properly powered on.

9.2 Active Serial Mode Programming

In this case, the configuration data has to be loaded into the configuration device on the DE-series board. Refer to Table 3 for a list of configuration devices on DE-series boards. To specify the required configuration device select **Assignments > Device**, which leads to the window in Figure 39.

Click on the **Device and Pin Options** button to reach the window in Figure 40. Now, click on the **Configuration** tab to obtain the window in Figure 41. In the Configuration device box (which may be set to Auto) choose the correct configuration device name and click **OK**. Upon returning to the window in Figure 39, click **OK**. Recompile the designed circuit.

Board	Configuration Device
DE0	EPCS4
DE0-Nano	EPCS64
DE1	EPCS4
DE2	EPCS16
DE2-70	EPCS64
DE2-115	EPCS64

Table 3. DE-series Configuration Device Names

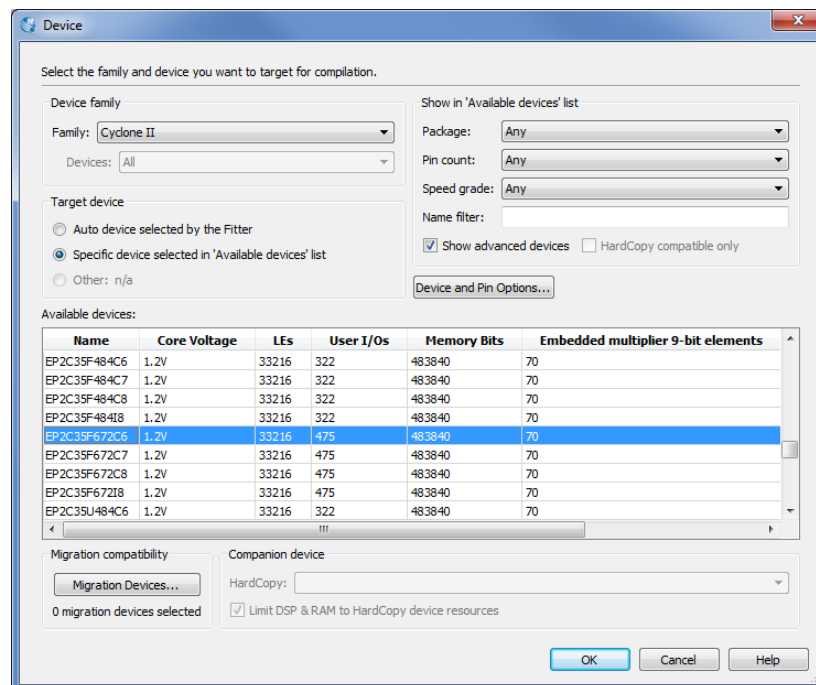


Figure 39. The Device Settings window.

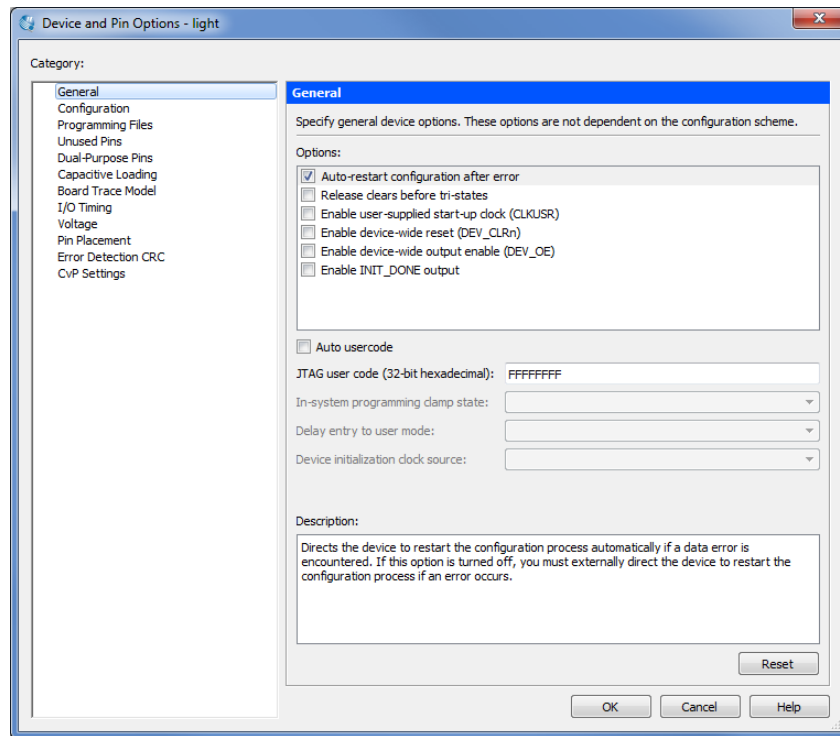


Figure 40. The Options window.

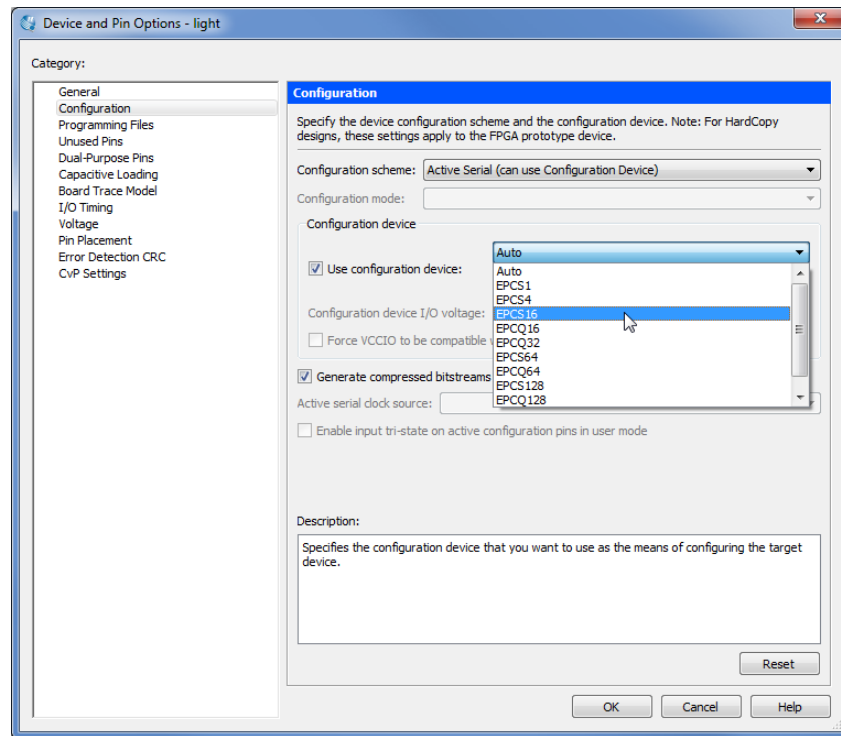


Figure 41. Specifying the configuration device.

The rest of the procedure is similar to the one described above for the JTAG mode. Select **Tools > Programmer** to reach the window in Figure 36. In the Mode box select **Active Serial Programming**. If you are changing the mode from the previously used JTAG mode, the pop-up box in Figure 42 will appear, asking if you want to clear all devices. Click **Yes**. Now, the Programmer window shown in Figure 43 will appear. Make sure that the Hardware Setup indicates the USB-Blaster. If the configuration file is not already listed in the window, press **Add File**. The pop-up box in Figure 44 will appear. Select the file *light.pof* in the directory *introtutorial/output_files* and click **Open**. As a result, the configuration file *light.pof* will be listed in the window. This is a binary file produced by the Compiler's Assembler module, which contains the data to be loaded into the configuration device on the DE-series board. The extension *.pof* stands for Programmer Object File. Upon returning to the Programmer window, click on the **Program/Configure** check box, as shown in Figure 45.

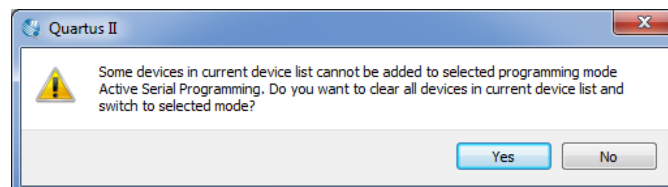


Figure 42. Clear the previously selected devices.

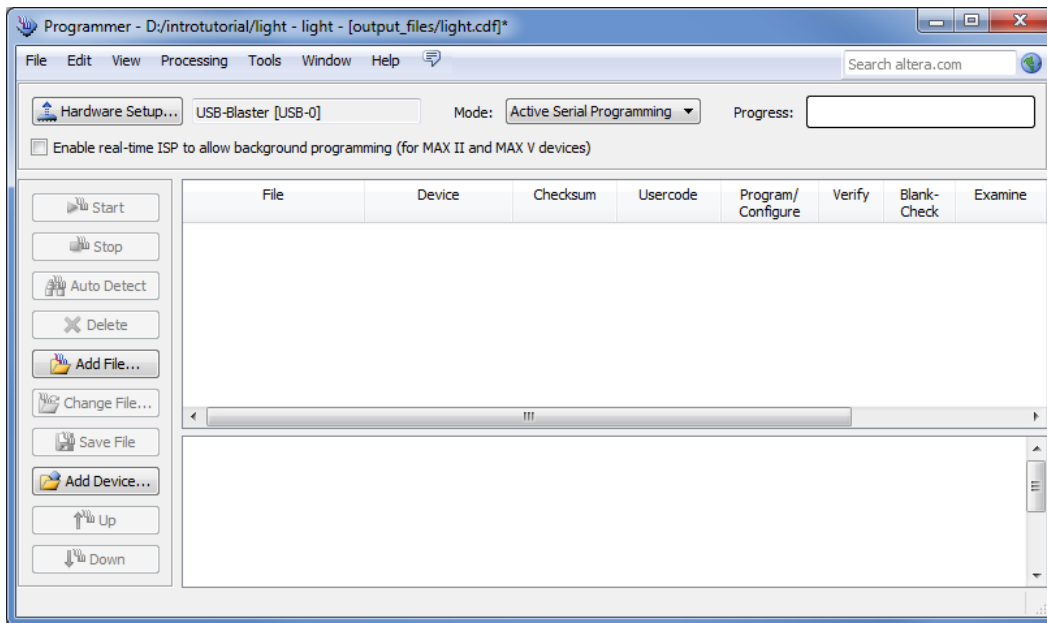


Figure 43. The Programmer window with Active Serial Programming selected.

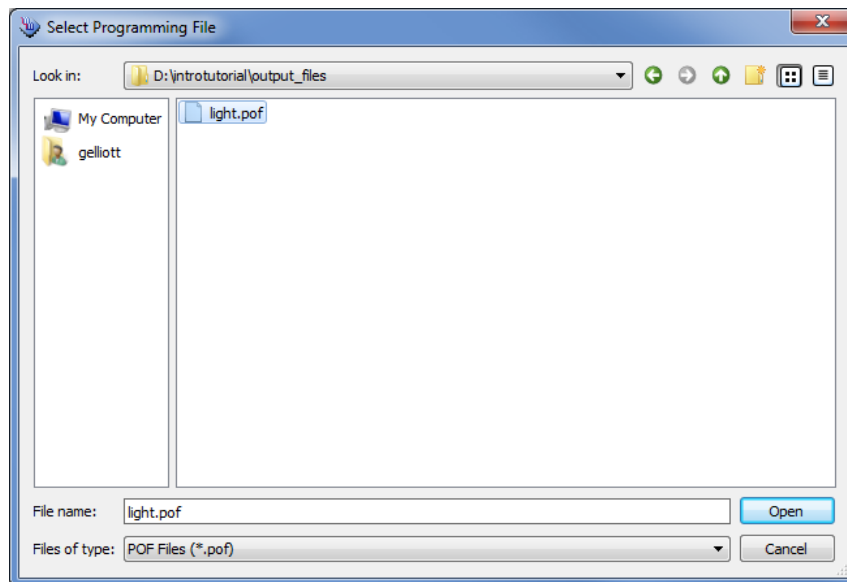


Figure 44. Choose the configuration file.

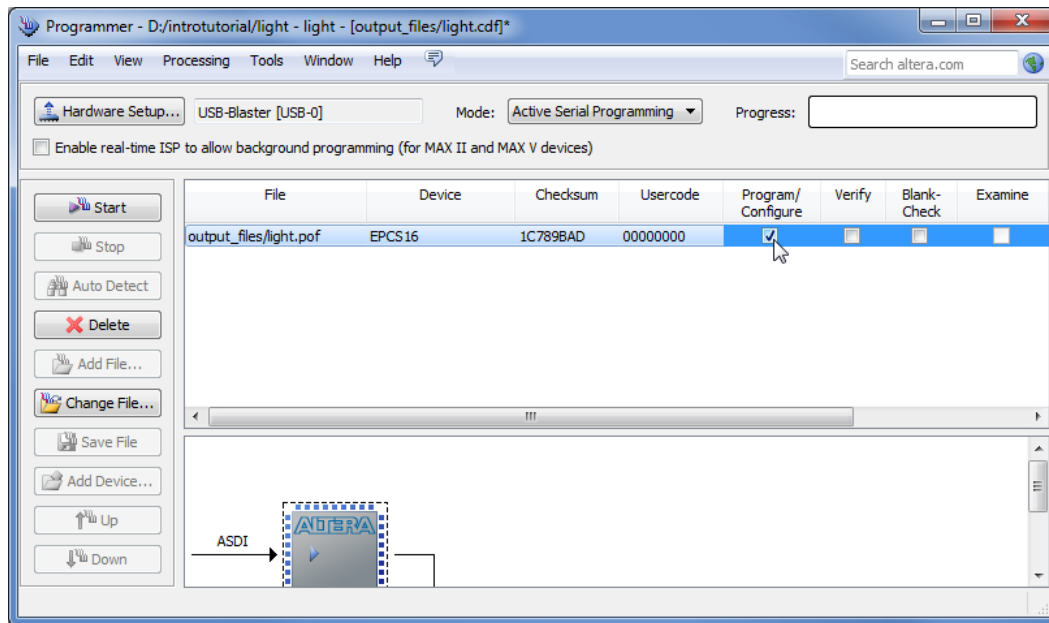


Figure 45. The updated Programmer window.

Flip the RUN/PROG switch on the DE-series board to the PROG position. Press Start in the window in Figure 45. An LED on the board will light up when the configuration data has been downloaded successfully. Also, the Progress box in Figure 45 will indicate when the configuration and programming process is completed, as shown in Figure 46.

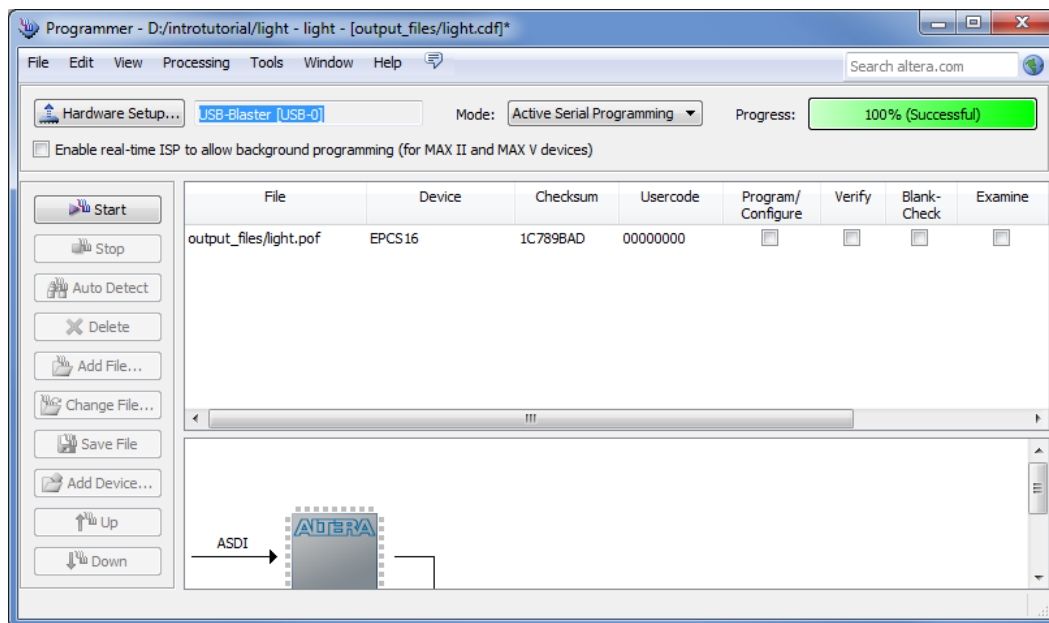


Figure 46. The Programmer window upon completion of programming.

10 Testing the Designed Circuit

Having downloaded the configuration data into the FPGA device, you can now test the implemented circuit. Flip the RUN/PROG switch to RUN position. Try all four valuations of the input variables x_1 and x_2 , by setting the corresponding states of the switches SW_1 and SW_0 . Verify that the circuit implements the truth table in Figure 11.

If you want to make changes in the designed circuit, first close the Programmer window. Then make the desired changes in the Verilog design file, compile the circuit, and program the board as explained above.

Copyright ©1991-2013 Altera Corporation. All rights reserved. Altera, The Programmable Solutions Company, the stylized Altera logo, specific device designations, and all other words and logos that are identified as trademarks and/or service marks are, unless noted otherwise, the trademarks and service marks of Altera Corporation in the U.S. and other countries. All other product or service names are the property of their respective holders. Altera products are protected under numerous U.S. and foreign patents and pending applications, mask work rights, and copyrights. Altera warrants performance of its semiconductor products to current specifications in accordance with Altera's standard warranty, but reserves the right to make changes to any products and services at any time without notice. Altera assumes no responsibility or liability arising out of the application or use of any information, product, or service described herein except as expressly agreed to in writing by Altera Corporation. Altera customers are advised to obtain the latest version of device specifications before relying on any published information and before placing orders for products or services.

This document is being provided on an "as-is" basis and as an accommodation and therefore all warranties, representations or guarantees of any kind (whether express, implied or statutory) including, without limitation, warranties of merchantability, non-infringement, or fitness for a particular purpose, are specifically disclaimed.