



# Quartus Prime Introduction Using Schematic Designs

*For Quartus Prime 16.1*

## 1 Introduction

This tutorial presents an introduction to the Quartus® Prime 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 Prime software. The design process is illustrated by giving step-by-step instructions for using the Quartus Prime software to implement a very simple circuit in an FPGA device.

The Quartus Prime 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 schematic design entry method, in which the user draws a graphical diagram of the circuit. Two other versions of this tutorial are also available, which use the Verilog and VHDL hardware description languages, respectively.

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 DE-series Development and Education board connected to a computer that has Quartus Prime 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 Prime version 16.1 Standard Edition; other versions of the software may be slightly different.

### **Contents:**

- Typical CAD Flow
- Getting Started
- Starting a New Project
- Schematic 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.

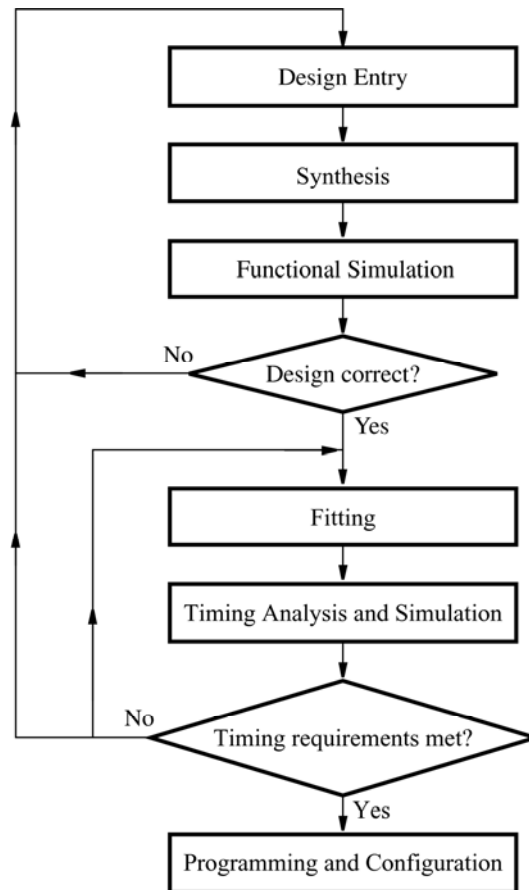


Figure 1. Typical CAD flow.

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 Prime software. It shows how the software can be used to design and implement a circuit specified by means of a schematic diagram. It makes use of the graphical user interface to invoke the Quartus Prime commands. Doing this tutorial, the reader will learn about:

- Creating a project
- Entering a schematic diagram
- Synthesizing a circuit from the schematic diagram
- Fitting a synthesized circuit into an FPGA
- Assigning the circuit inputs and outputs to specific pins on the FPGA
- Simulating the designed circuit
- Programming and configuring the FPGA chip on 's DE-series board

### 3 Getting Started

Each logic circuit, or subcircuit, being designed with Quartus Prime 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 Prime 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 Prime software, which the user selects with the computer mouse. Most of the commands provided by Quartus Prime 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 Prime 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.

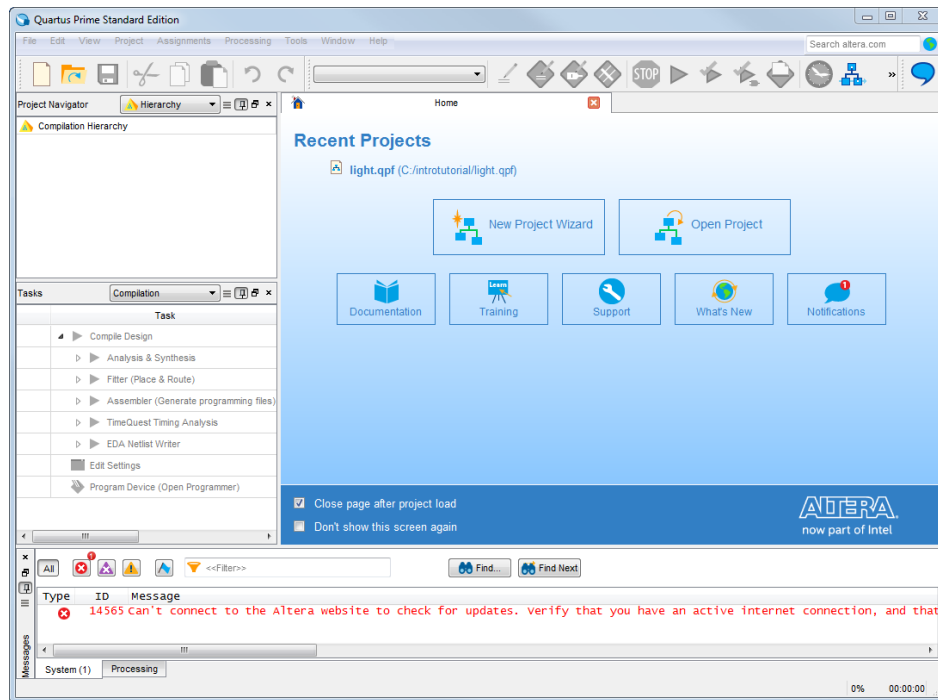


Figure 2. The main Quartus Prime display.

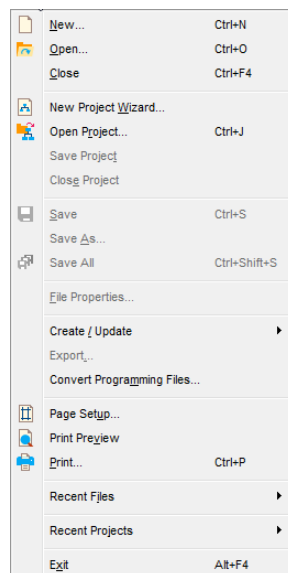


Figure 3. An example of the File menu.

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 Prime Online Help

Quartus Prime 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.

The user can quickly search through the Help topics by using the dialog box in the top right corner of the main Quartus display. 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 Prime 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.
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 Prime 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.

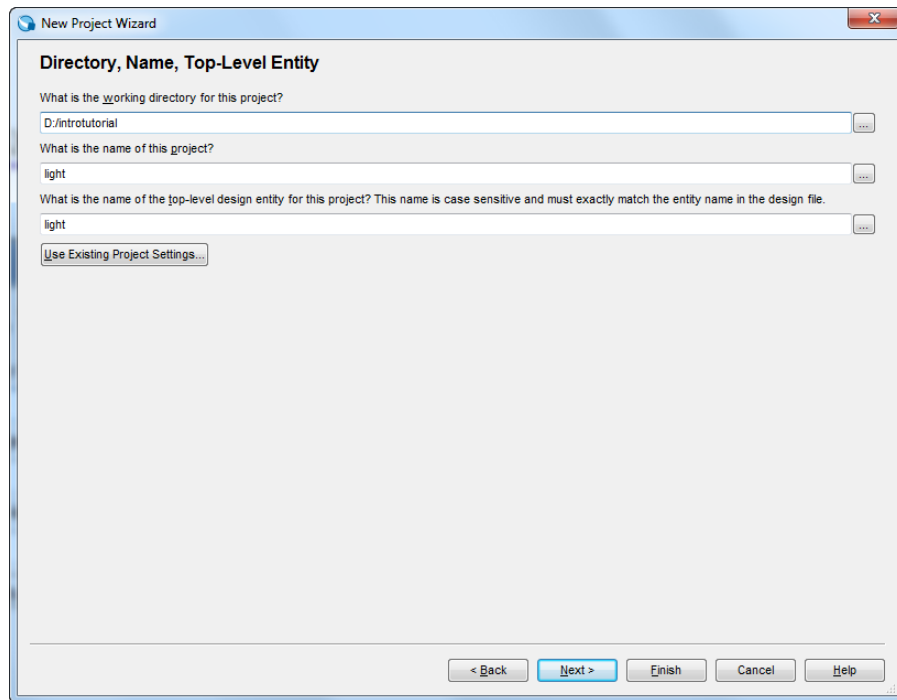


Figure 4. Creation of a new project.

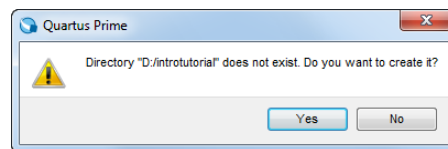


Figure 5. Quartus Prime software can create a new directory for the project.

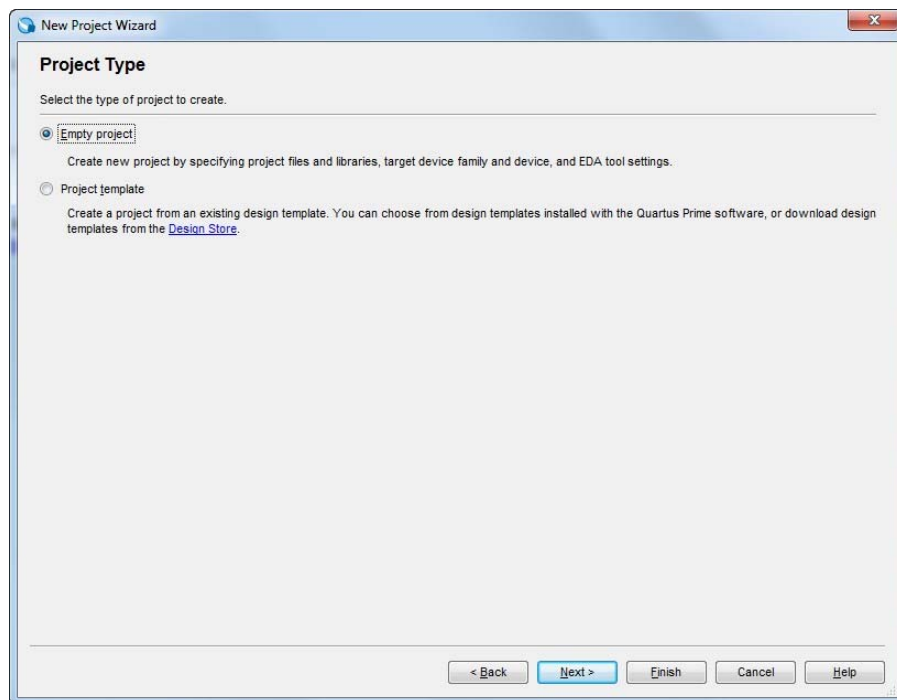


Figure 6. Choosing the project type.

3. The Project Type window, shown in Figure 6, allows you to choose from the Empty project and the Project template options. For this tutorial, choose Empty project as we will be creating a project from scratch, and press Next which leads to the window in Figure 7.

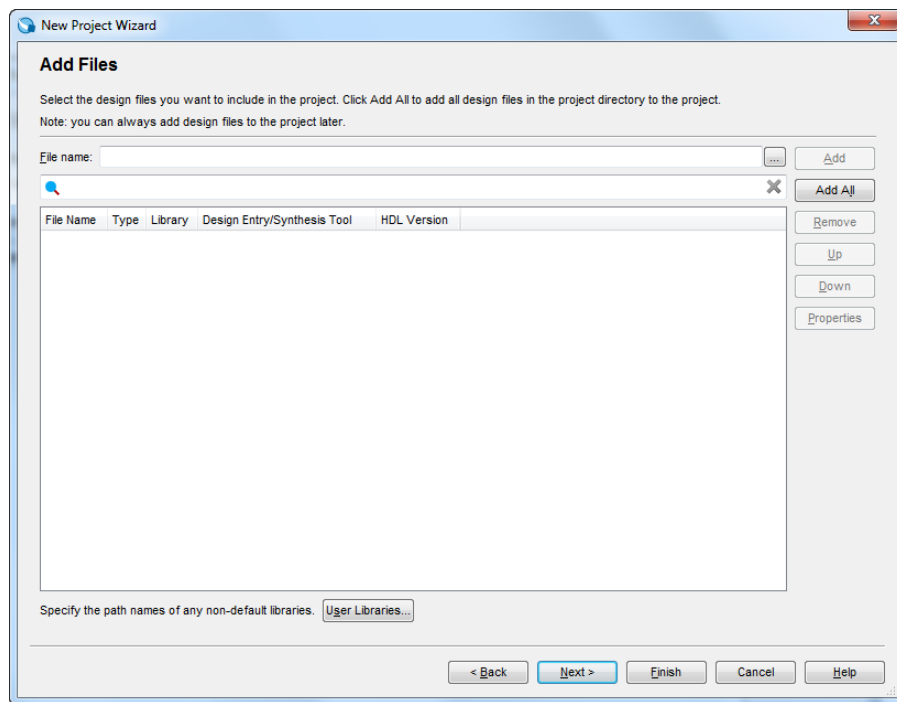


Figure 7. The wizard can include user-specified design files.

4. 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 8.



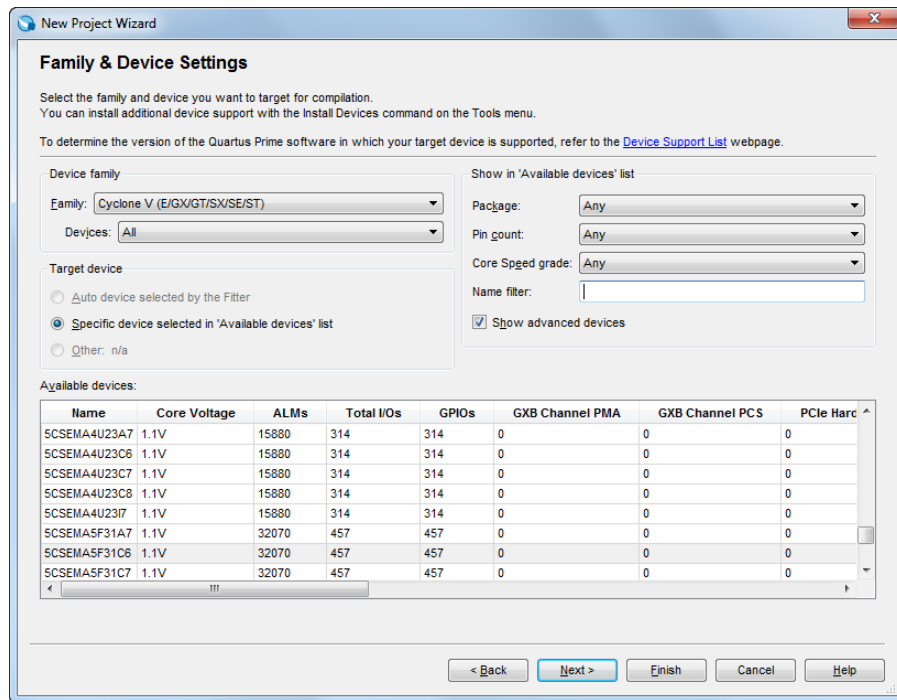


Figure 8. Choose the device family and a specific device.

- 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 Prime 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 9.

Board	Device Name
DE0-CV	Cyclone V 5CEBA4F23C7
DE0-Nano	Cyclone IVE EP4CE22F17C6
DE0-Nano-SoC	Cyclone V SoC 5CSEMA4U23C6
DE1-SoC	Cyclone V SoC 5CSEMA5F31C6
DE2-115	Cyclone IVE EP4CE115F29C7
DE10-Lite	Max 10 10M50DAF484C7G
DE5a-NET	Arria 10 10AX115N3F45E2SG

Table 1. DE-series FPGA device names

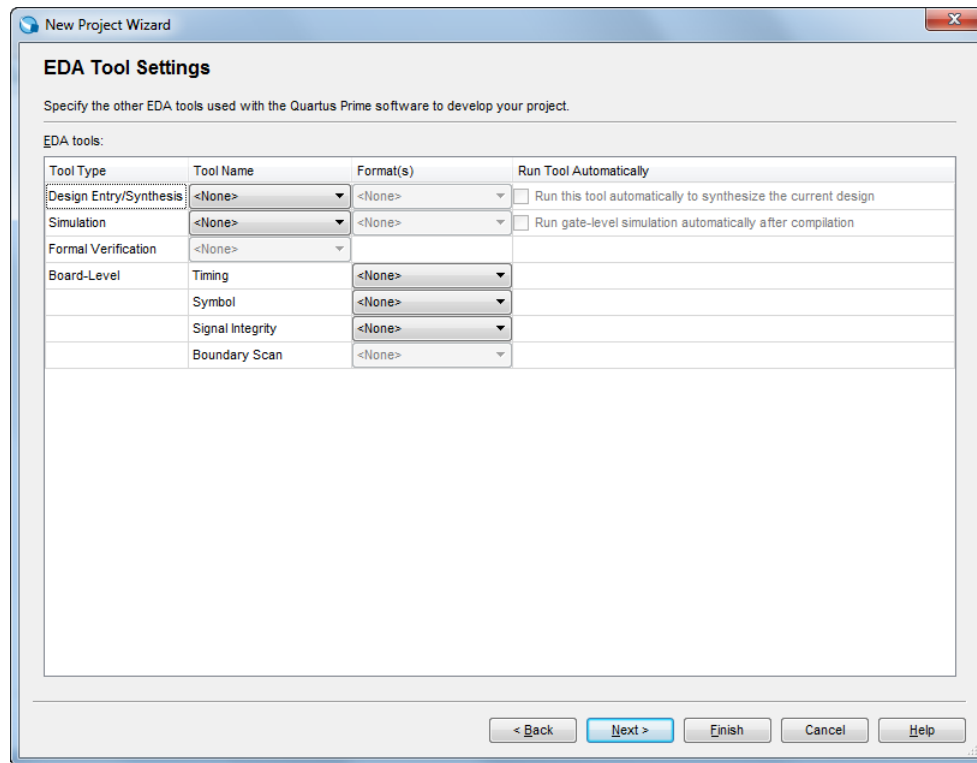


Figure 9. Other EDA tools can be specified.

6. 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 Prime messages that refer to third-party tools, which are the tools developed and marketed by companies other than. Since we will rely solely on Quartus Prime tools, we will not choose any other tools. Press **Next**.
7. A summary of the chosen settings appears in the screen shown in Figure 10. Press **Finish**, which returns to the main Quartus Prime window, but with *light* specified as the new project, in the display title bar, as indicated in Figure 11.

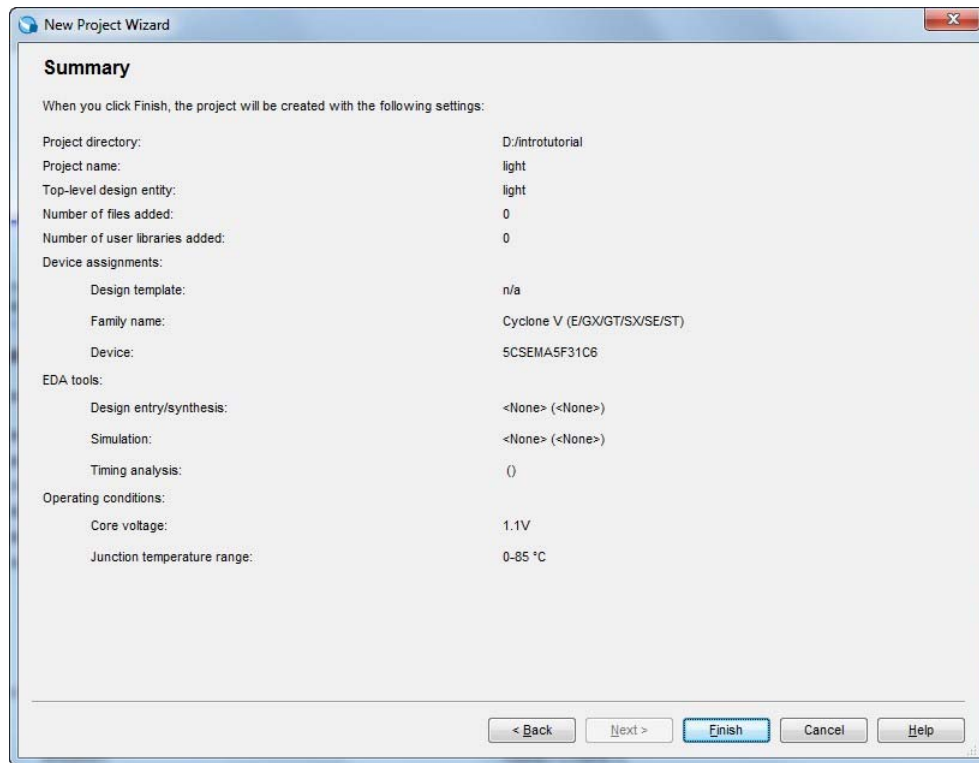


Figure 10. Summary of project settings.

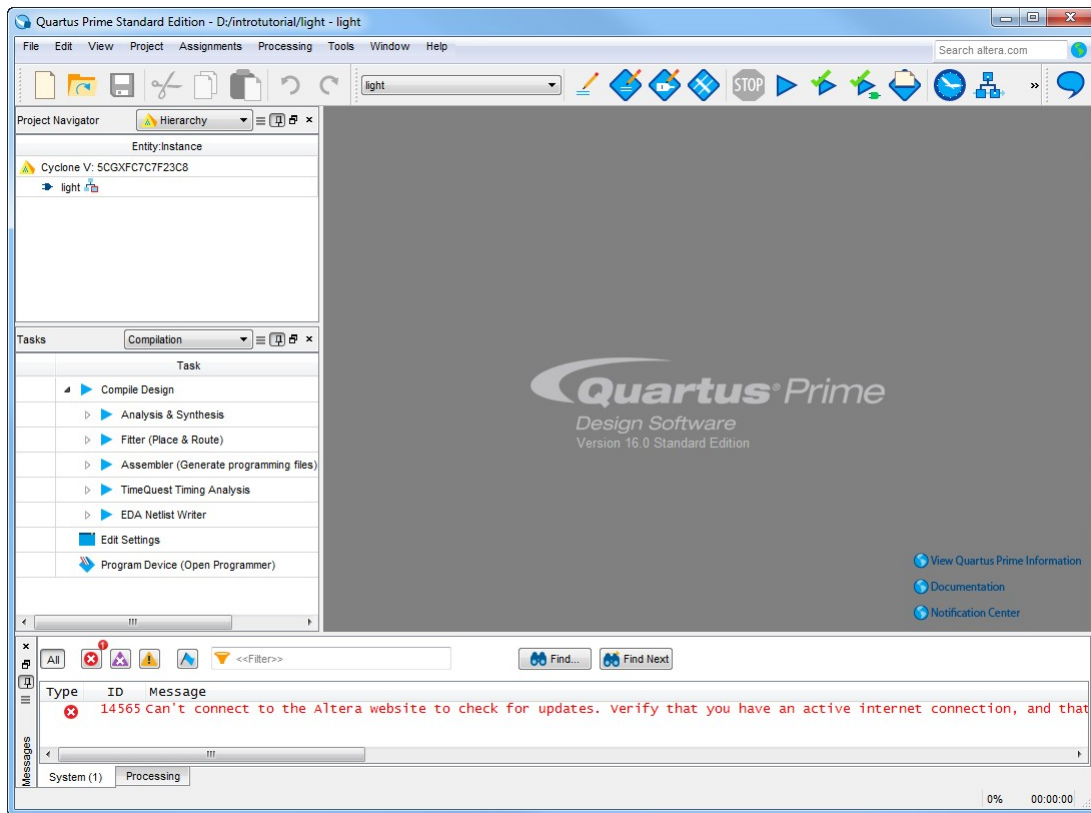


Figure 11. The Quartus Prime display for a created project.

## 5 Design Entry Using the Graphic Editor

As a design example, we will use the two-way light controller circuit shown in Figure 12. 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 implement it using the gates shown.

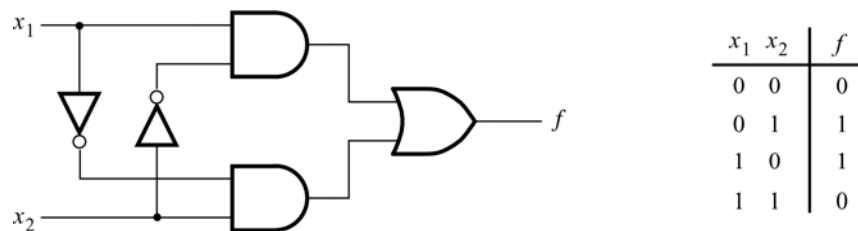


Figure 12. The light controller circuit.

The Quartus Prime Graphic Editor can be used to specify a circuit in the form of a block diagram. Select **File > New** to get the window in Figure 13, choose **Block Diagram/Schematic File**, and click **OK**. This opens the Graphic 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 **Block Diagram/Schematic File (\*.bdf)**. In the box labeled **File name** type *light*, to match the name given in Figure 4, which was specified when the project was created. 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 Graphic Editor window displayed in Figure 15.

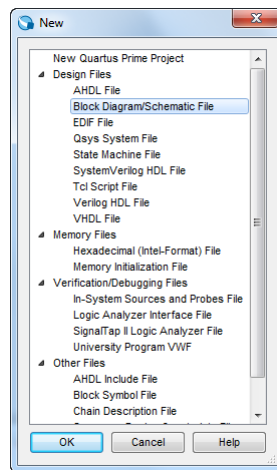


Figure 13. Choose to prepare a block diagram.

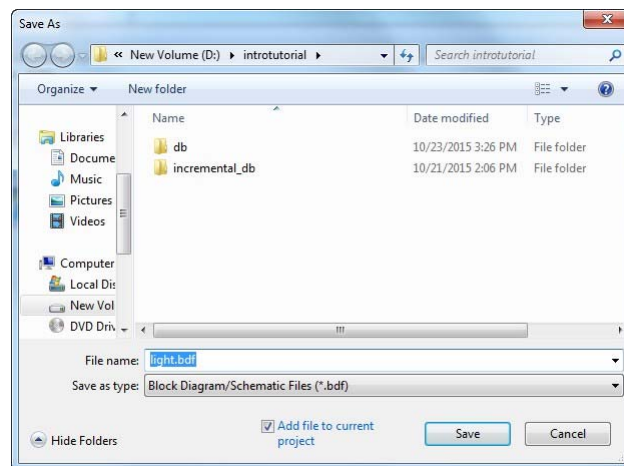


Figure 14. Name the file.

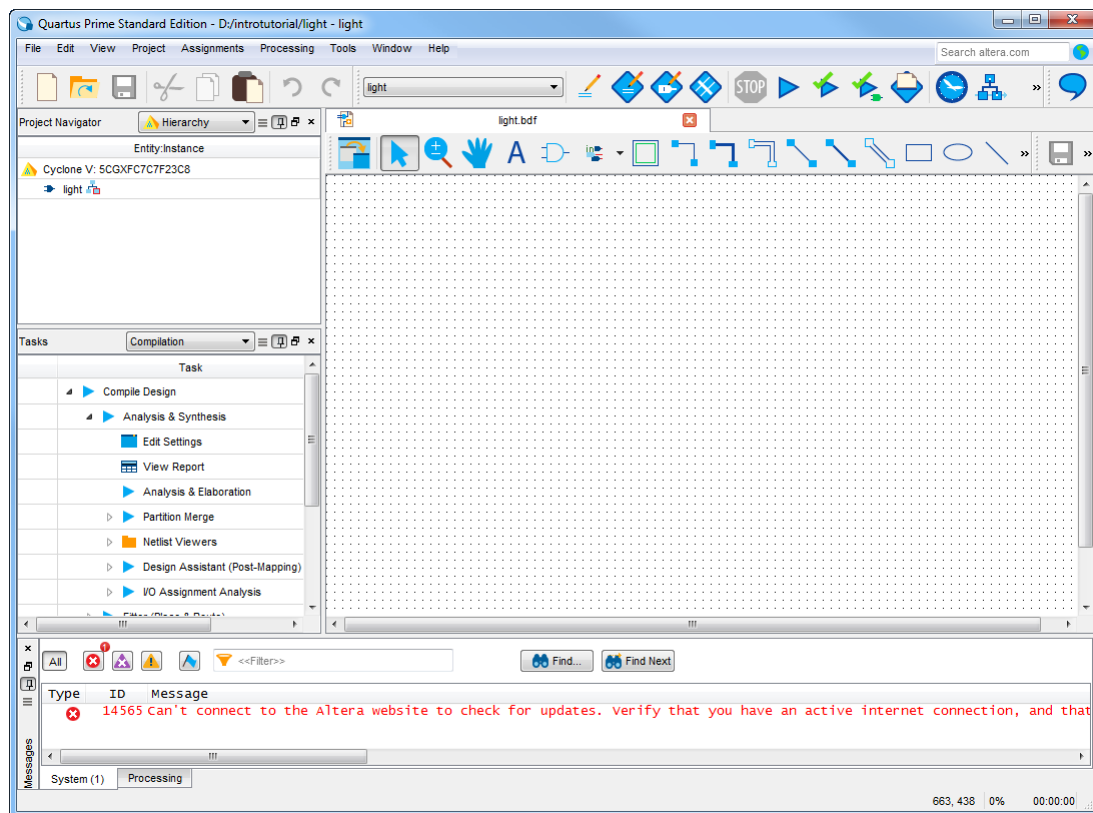





Figure 15. Graphic Editor window.

## 5.1 Importing Logic-Gate Symbols

The Graphic Editor provides a number of libraries which include circuit elements that can be imported into a schematic. Double-click on the blank space in the Graphic Editor window, or click on the  icon in the toolbar that looks like an AND gate. A pop-up box in Figure 16 will appear. Expand the hierarchy in the Libraries box as shown in the figure. First expand *libraries*, then expand the library *primitives*, followed by expanding the library *logic* which comprises the logic gates. Select *and2*, which is a two-input AND gate, and click OK. Now, the AND gate symbol will appear in the Graphic Editor window. Using the mouse, move the symbol to a desirable location and click to place it there. Import a second AND gate by repeating the previous steps. Note that you can enable the Repeat-insert mode in the Symbol window to quickly add multiple copies of a gate. A symbol in the Graphic Editor window can be moved by clicking the  icon in the toolbar that looks like a mouse cursor, then clicking the symbol you want to move and dragging it to a new location with the mouse button pressed. Next, select *or2* from the library and import the OR gate into the diagram. Then, select *not* and import two instances of the NOT gate. Rotate the NOT gates into proper position by right clicking on the desired gate and selecting Rotate by Degrees > Rotate Left 90, or by using the “Rotate left 90” icon . Arrange the gates as shown in Figure 17.

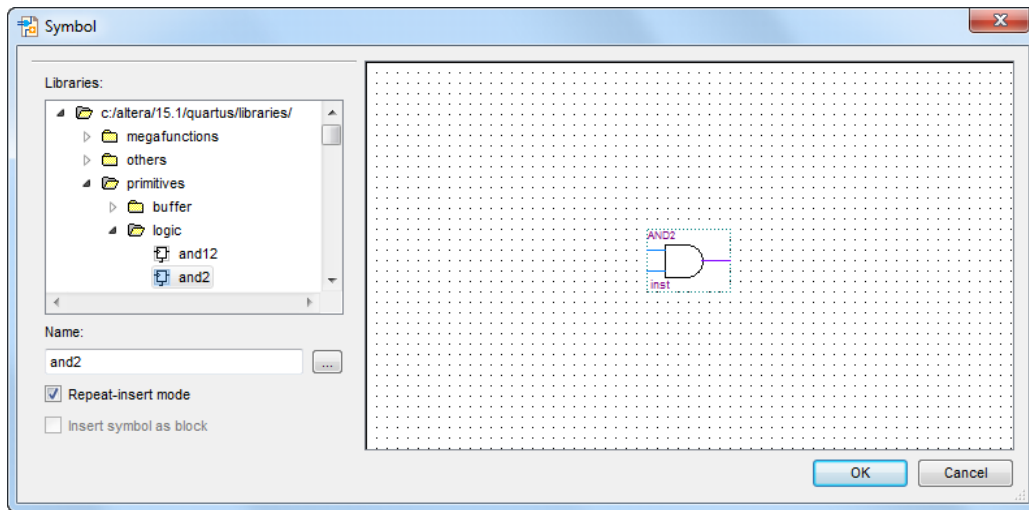


Figure 16. Choose a symbol from the library.

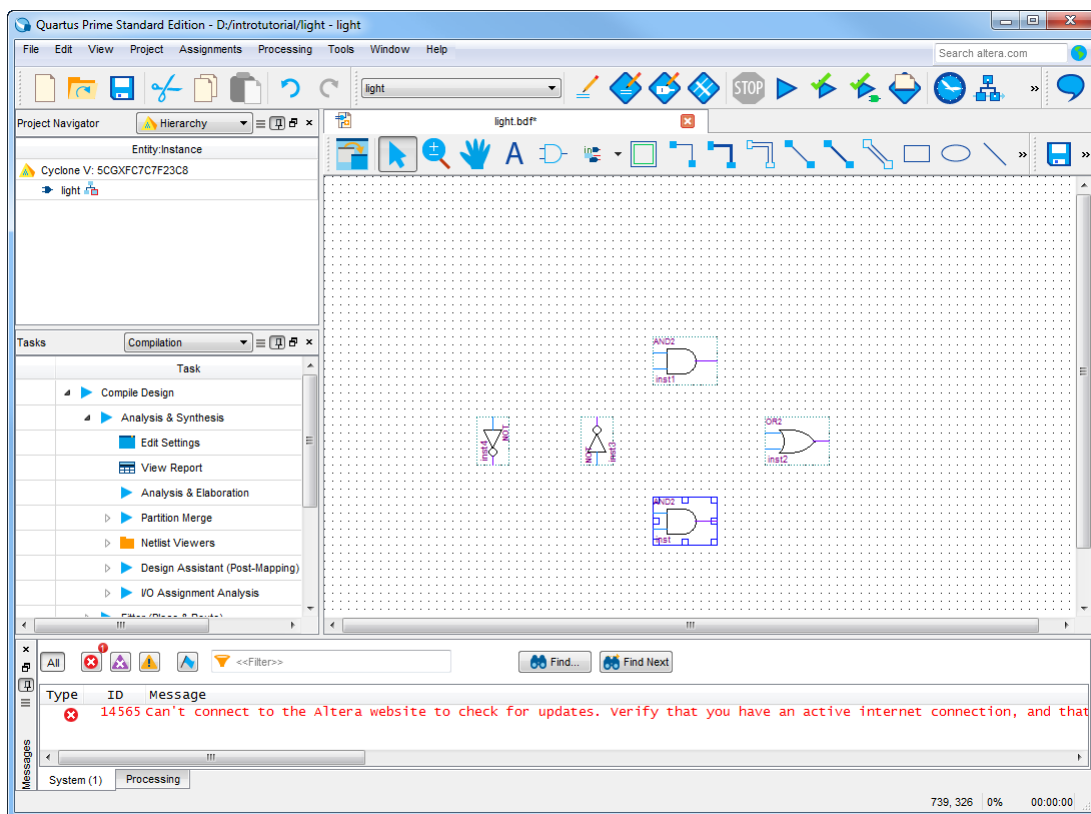


Figure 17. Import the gate symbols into the Graphic Editor window.

## 5.2 Importing Input and Output Symbols

Having entered the logic-gate symbols, it is now necessary to enter the symbols that represent the input and output ports of the circuit. Use the same procedure as for importing the gates, but choose the port symbols from the library *primitives/pin*. Import two instances of the input port and one instance of the output port, to obtain the image in Figure 18.

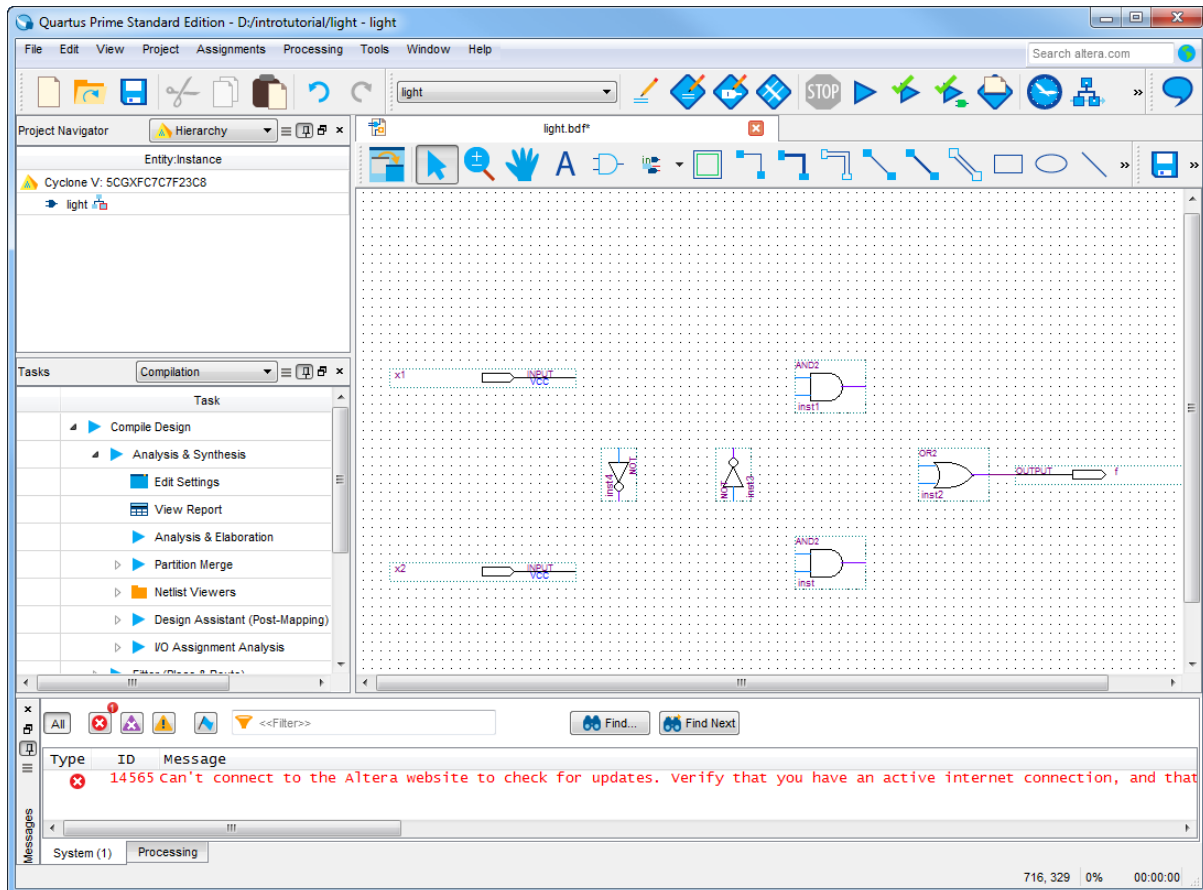


Figure 18. Import the input and output pins.

Assign names to the input and output symbols as follows. Make sure nothing is selected by clicking on an empty spot in the Graphic Editor window. Point to the top input symbol and double-click the mouse. The dialog box in Figure 19 will appear. Type the pin name, *x1*, and click OK. Similarly, assign the name *x2* to the other input and *f* to the output. Alternatively, it is possible to change the name of an element by double-clicking on the name and typing a new one directly.



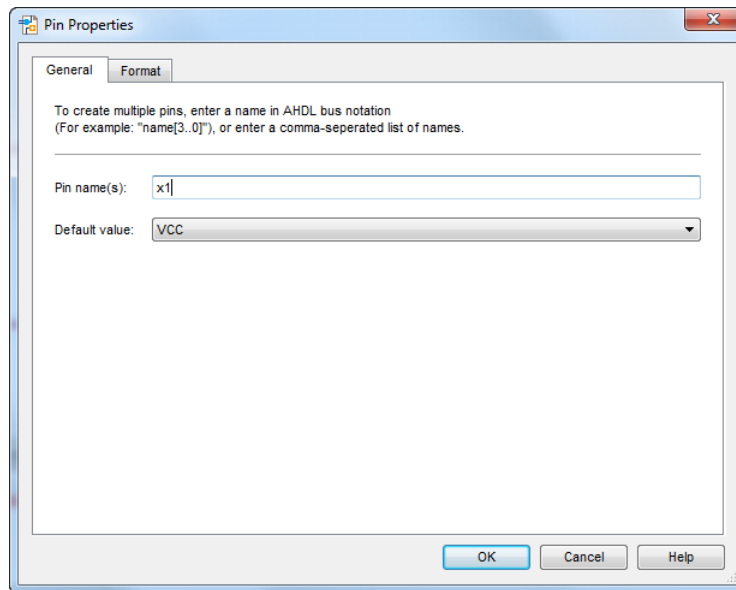




Figure 19. Naming of a pin.

### 5.3 Connecting Nodes with Wires

The symbols in the diagram have to be connected by drawing lines (wires). Click on the icon  in the toolbar to activate the Orthogonal Node Tool. Position the mouse pointer over the right edge of the x1 input pin. Click and hold the mouse button and drag the mouse to the right until the drawn line reaches the pinstub on the top input of the AND gate. Release the mouse button when you see a box appear, which leaves the line connecting the two pinstubs. Next, draw a wire from the input pinstub of the leftmost NOT gate to touch the wire that was drawn above it. Note that a dot will appear indicating a connection between the two wires.

Use the same procedure to draw the remaining wires in the circuit. If a mistake is made, a wire can be selected by clicking on it, and removed by pressing the Delete key on the keyboard. Upon completing the diagram, click on the  icon, to activate the Selection Tool. Now, changes in the appearance of the diagram can be made by selecting a particular symbol or wire and either moving it to a different location or deleting it. The final diagram is shown in Figure 20; save it.

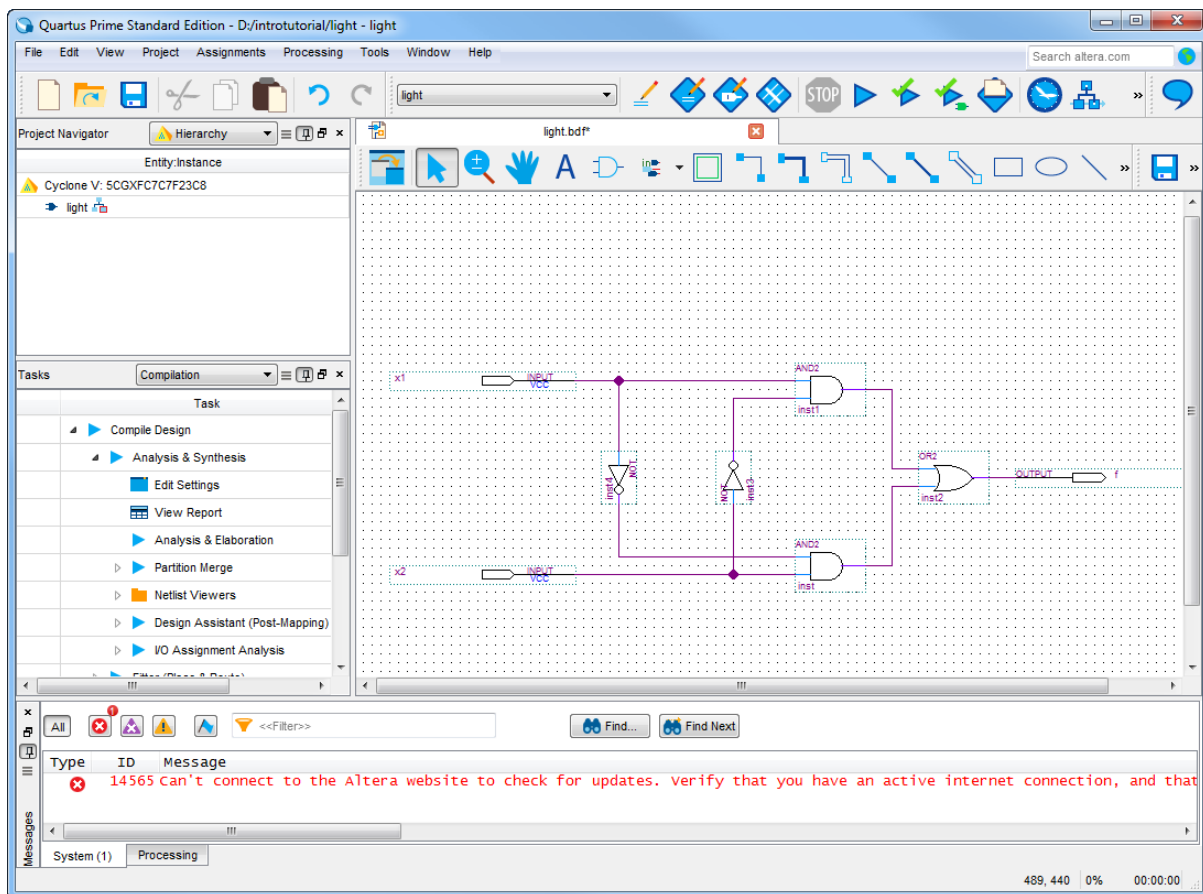




Figure 20. The completed schematic diagram.

## 6 Compiling the Designed Circuit

The entered schematic diagram file, *light.bdf*, is processed by several Quartus Prime tools that analyze the file, 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 blue 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 Prime display. In the message window, at the bottom of the figure, various messages are displayed throughout the compilation process. In case of errors, there will be appropriate messages given.

When the compilation is finished, a compilation report is produced. A tab showing this report is opened automatically, as seen in Figure 21. 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 21 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.

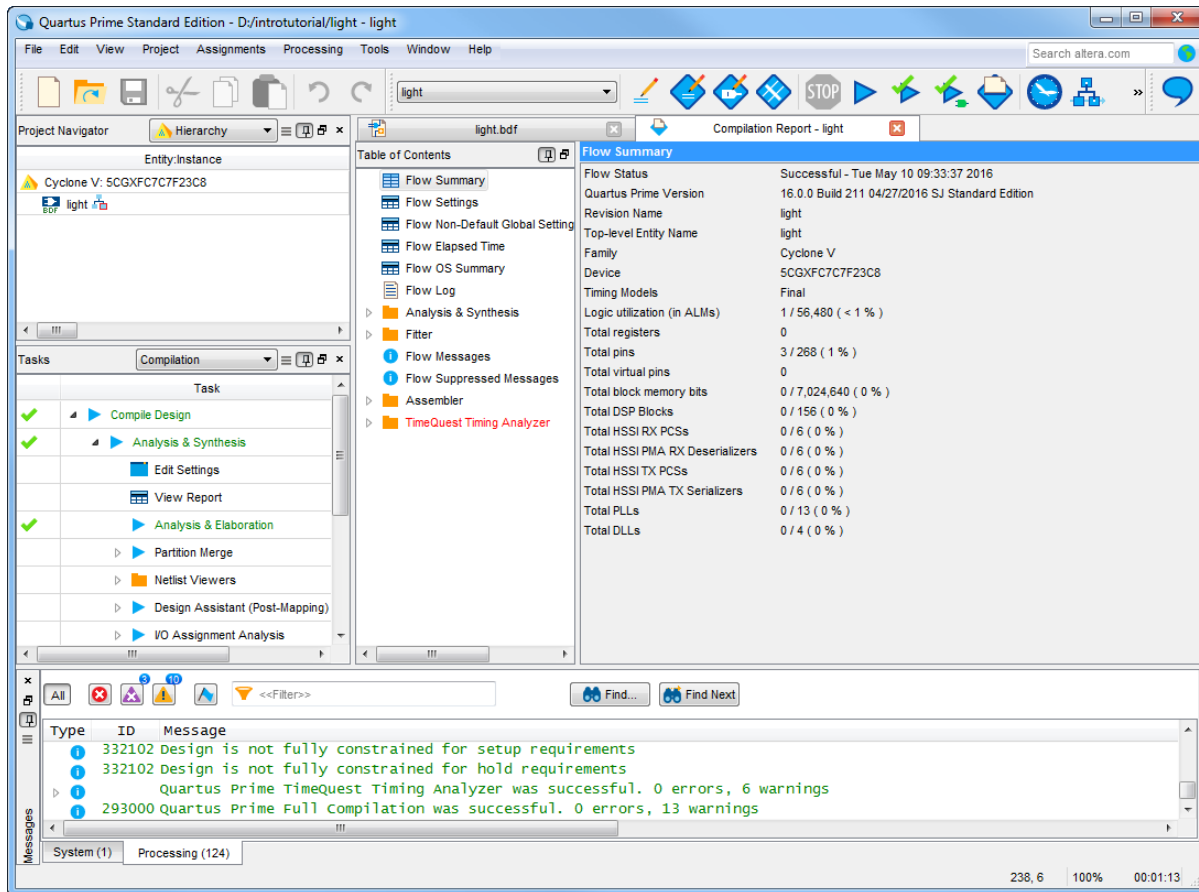




Figure 21. Display after a successful compilation.

## 6.1 Errors

Quartus Prime software displays messages produced during compilation in the Messages window. If the block diagram 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 schematic entry. 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 part of the circuit in the Graphic 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.bdf*. Remove the wire connecting the output of the top AND gate

to the OR gate. To do this, click on the  icon, click the mouse on the wire to be removed (to select it) and press Delete. Compile the erroneous design by clicking on the  icon. A pop-up box will ask if the changes made to the *light.bdf* file should be saved; click **Yes**. After trying to compile the circuit, Quartus Prime software will display error messages in the Messages window, and show that the compilation failed in the Analysis & Synthesis stage of the compilation process. The compilation report summary, given in Figure 22, 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 23. The Compilation Report can be displayed as a separate window as in Figure 23 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, which states that one of the nodes is missing a source. Quartus Prime software responds by displaying the *light.bdf* schematic and highlighting the OR gate which is affected by the error, as shown in Figure 24. Correct the error and recompile the design.

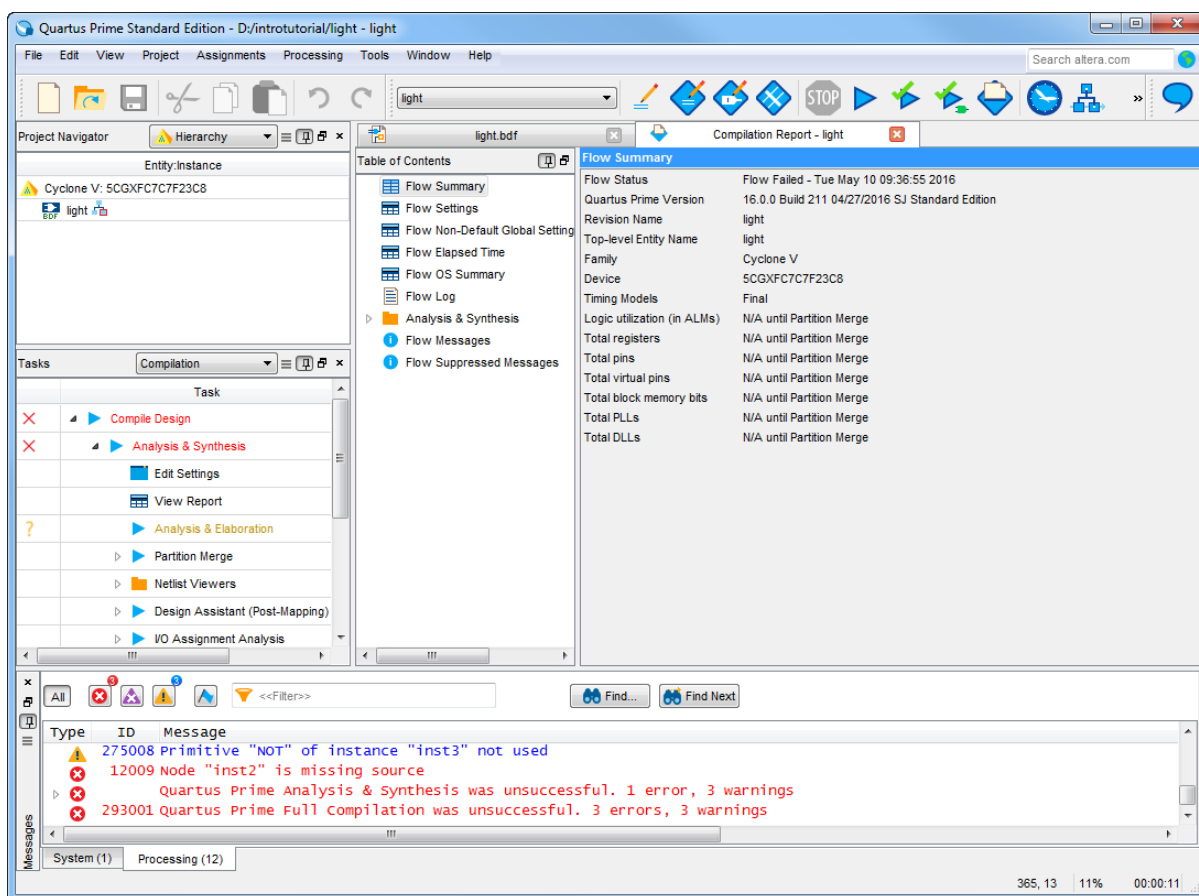


Figure 22. Compilation report for the failed design.

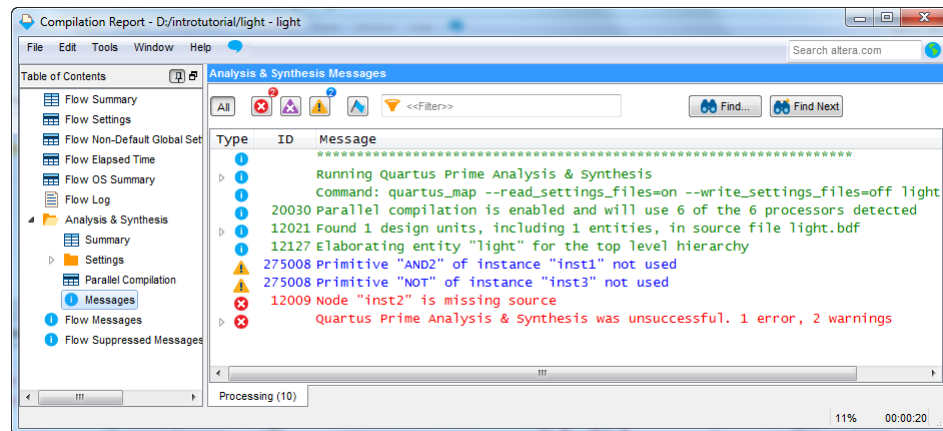


Figure 23. Error messages.

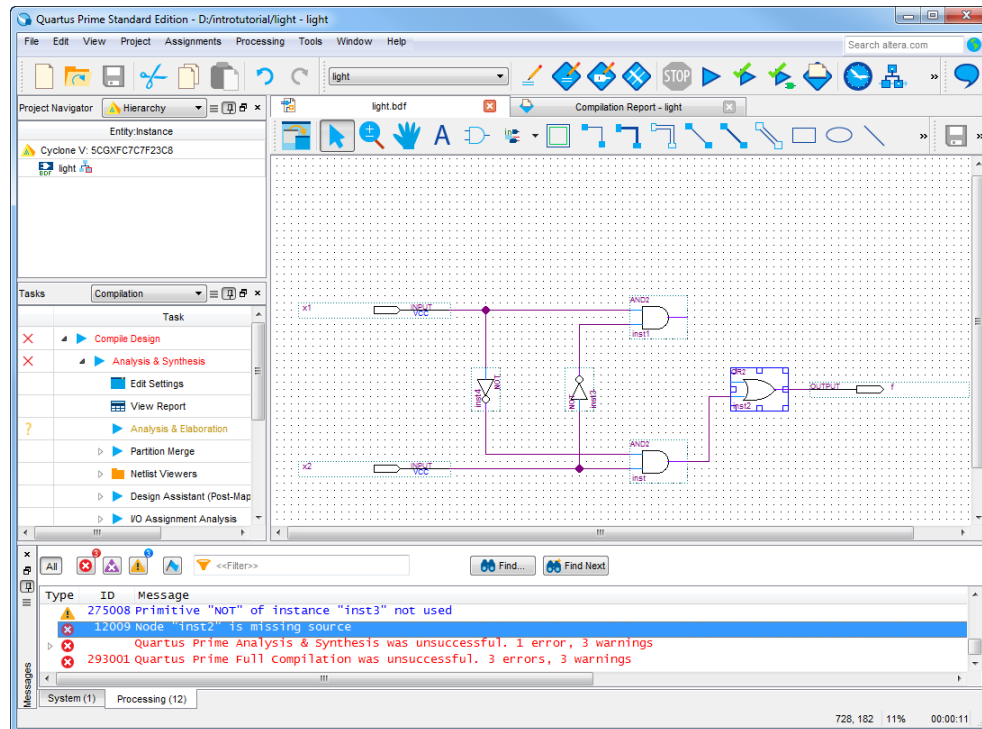


Figure 24. Identifying the location of the error.

## 7 Pin Assignment

During the compilation above, the Quartus Prime 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 a light-emitting diode on your DE-series board. For the DE2-115 we will use a green LED:  $LEDG_0$ . On the DE0-CV and DE1-SoC, we will use  $LEDR_0$ . On the DE0-Nano and DE0-Nano-SoC, we will use  $LED_0$ . The FPGA pin assignment for the LEDs can also be found in Table 2.

Component	DE0-CV	DE0-Nano	DE0-Nano-SoC	DE2-115	DE1-SoC
$SW_0$	PIN_U13	PIN_M1	PIN_L10	PIN_AB28	PIN_AB12
$SW_1$	PIN_V13	PIN_T8	PIN_L9	PIN_AC28	PIN_AC12
$LEDG_0$ , $LED_0$ , or $LEDR_0$	PIN_AA2	PIN_A15	PIN_W15	PIN_E21	PIN_V16

Table 2. DE-Series Pin Assignments

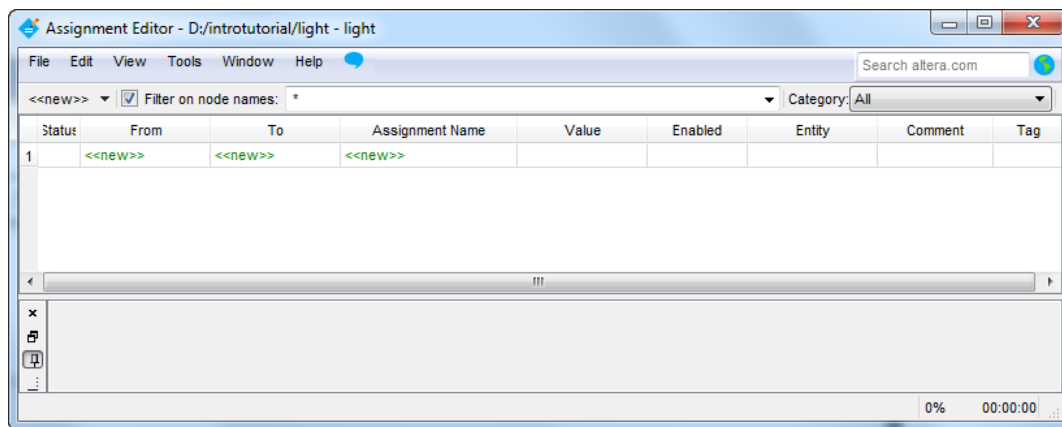





Figure 25. The Assignment Editor window.

Pin assignments are made by using the Assignment Editor. Select **Assignments > Assignment Editor** to reach the window in Figure 25 (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 26. Click on  and  to show or hide 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:  $f$ ,  $x_1$ , and  $x_2$ . Click on  $x_1$  as the first pin to be assigned and click the **>** button; this will enter  $x_1$  in the Selected Nodes box. Click **OK**.  $x_1$  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  $x_1$  entry, in the column labeled **Assignment Name**. Now, the drop-down menu in Figure 27 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 DE1-SoC board is shown in Figure 28. 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.

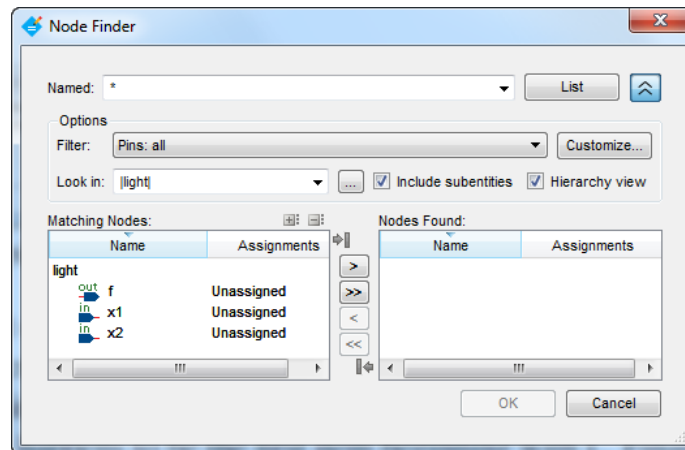


Figure 26. The Node Finder displays the input and output names.

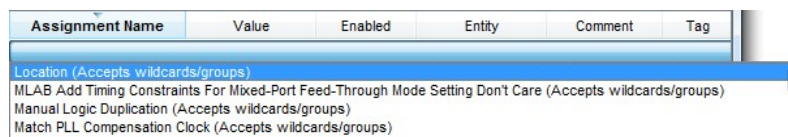


Figure 27. The available assignment names for a DE-series board.

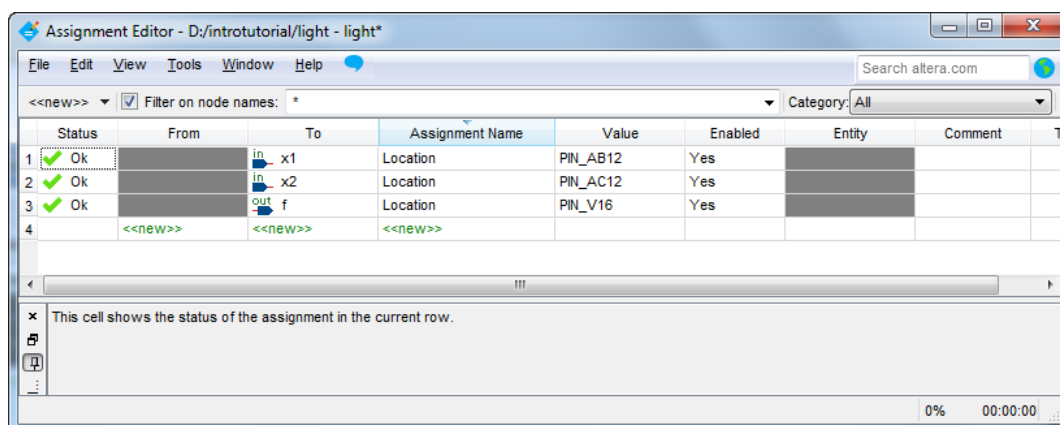


Figure 28. The complete assignment.

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 Prime 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 Settings File (QSF)* format. The format for the file for our simple project (on a DE1-SoC board) is

```
set_location_assignment PIN_AB12 -to x1
set_location_assignment PIN_AC12 -to x2
set_location_assignment PIN_V16 -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 28. Select **Assignments > Export Assignment** which leads to the window in Figure 29. 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.

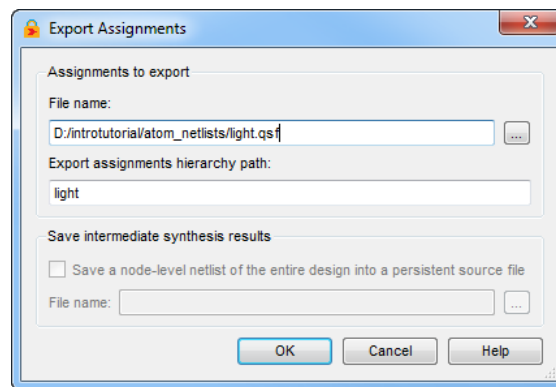


Figure 29. Exporting the pin assignment.

You can import a pin assignment by choosing **Assignments > Import Assignments**. This opens the dialogue in Figure 30 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.



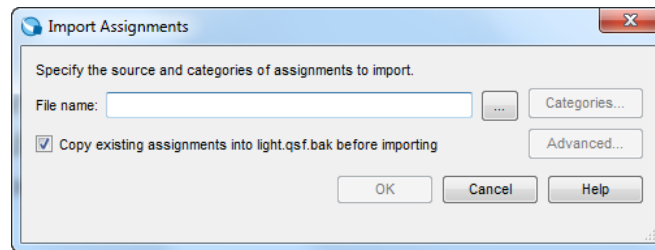


Figure 30. Importing the pin assignment.

For convenience when using large designs, all relevant pin assignments for the DE-series board are given in individual files. For example, the DE1-SoC pin assignments can be found in the *DE1\_SoC.qsf* file, which is available from Intel's FPGA University Program website. This file uses the names found in the *DE1-SoC 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 Block Diagram/Schematic design file; namely, *SW[0]*, *SW[1]* and *LEDG[0]* for *x1*, *x2* and *f*, respectively. Since these signals are specified in the *DE1\_SoC.qsf* file as elements of vectors *SW* and *LEDG*, we must refer to them in the same way in our design file. For example, in the *DE1\_SoC.qsf* file the 10 toggle switches are called *SW[9]* to *SW[0]*. In a design file they can also be referred to as a vector *SW[9..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 Prime'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:

\*NOTE: before starting simulation, you need to link to the ModelSim-Altera simulation tool. In the main Quartus Prime window, select Tools > Options > EDA Tool Options. Click on the three dots [...] to the right of ModelSim-Altera. Navigate to C:\> Software > Modelsim > 10.6c and select win64pe. Click ok and proceed with the tutorial.

1. In the main Quartus Prime 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 31. 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 32. You may wish to resize the window to its maximum size.

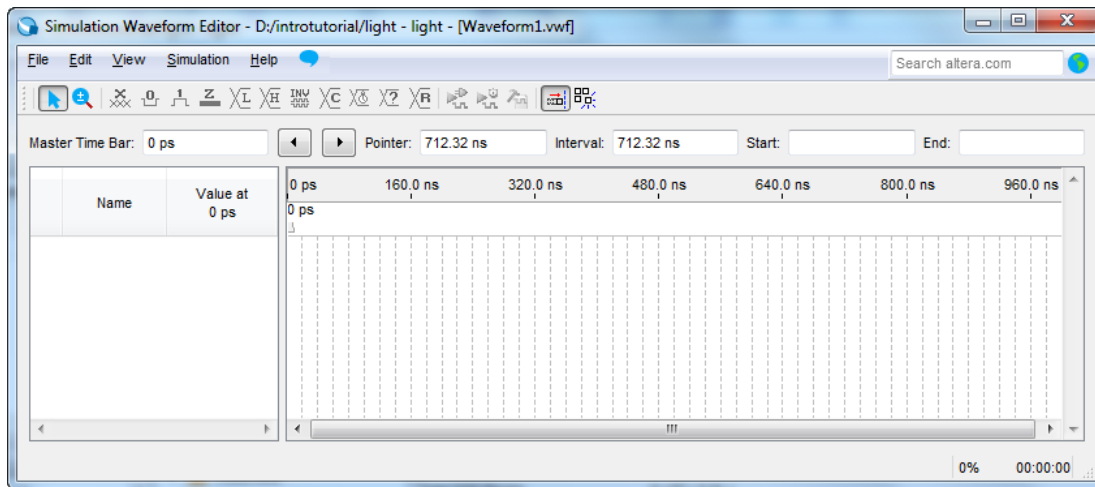


Figure 31. The Waveform Editor window.

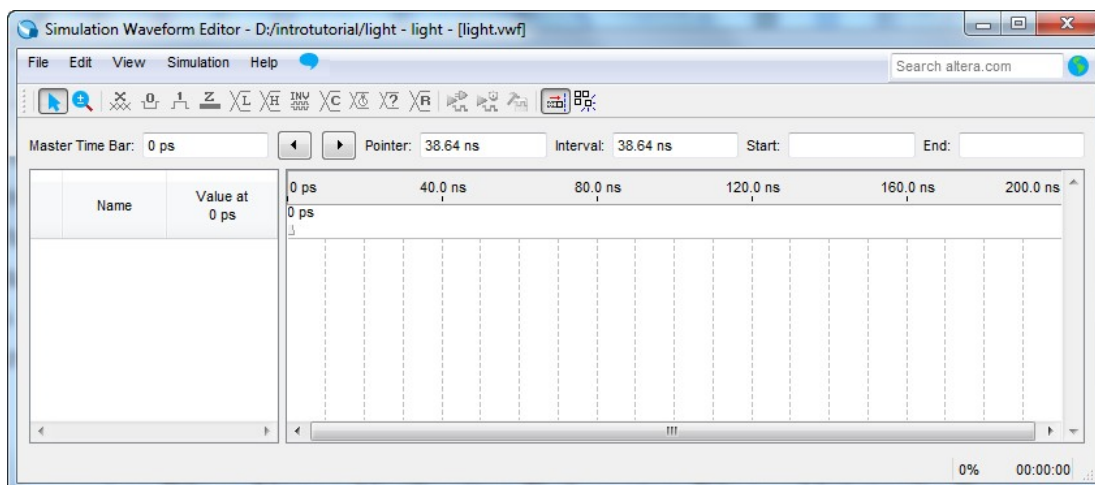


Figure 32. The augmented Waveform Editor window.

- 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 33. 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 34. 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.

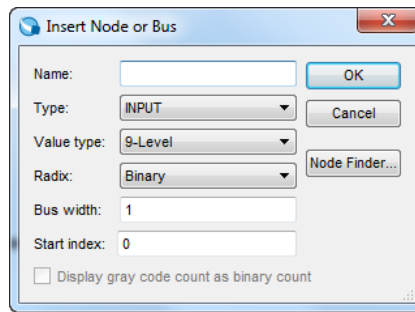


Figure 33. The Insert Node or Bus dialogue.

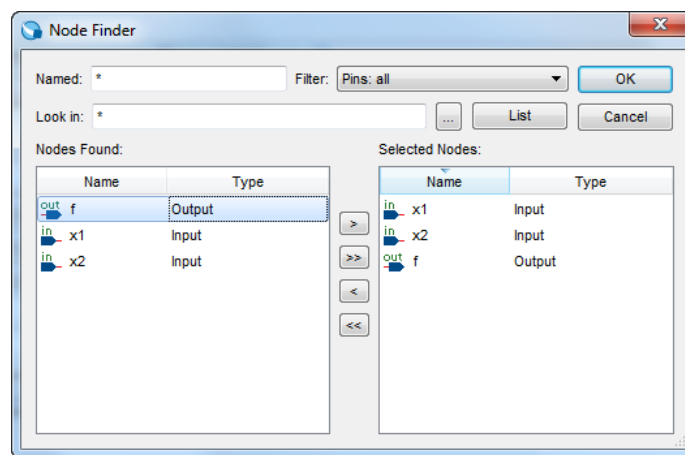


Figure 34. Selecting nodes to insert into the Waveform Editor.

Click on the *x1* signal in the Nodes Found box in Figure 34, 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 33. This leaves a fully displayed Waveform Editor window, as shown in Figure 35. If you did not select the nodes in the same order as displayed in Figure 35, 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.

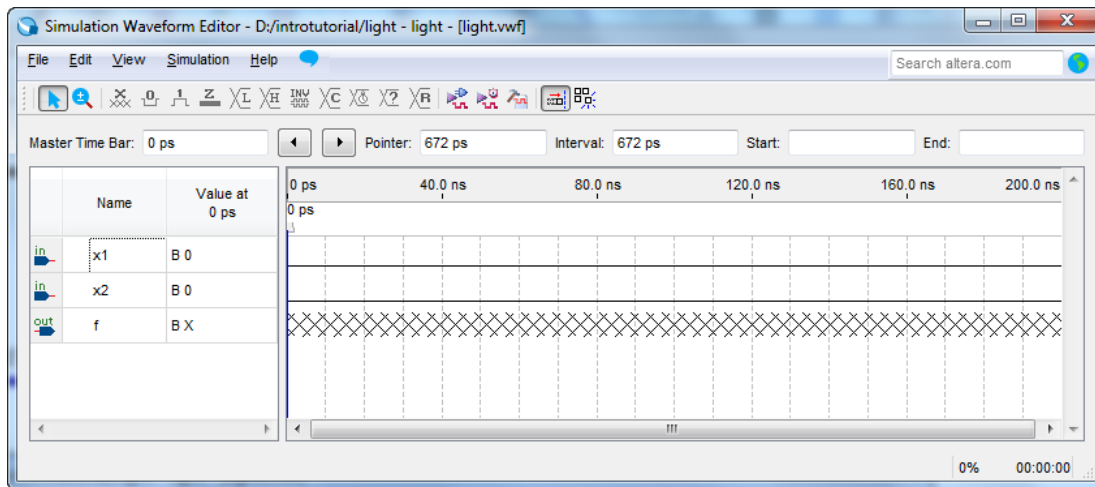



Figure 35. The nodes needed for simulation.

- 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 12. 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 12. This should produce the image in Figure 36. 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.

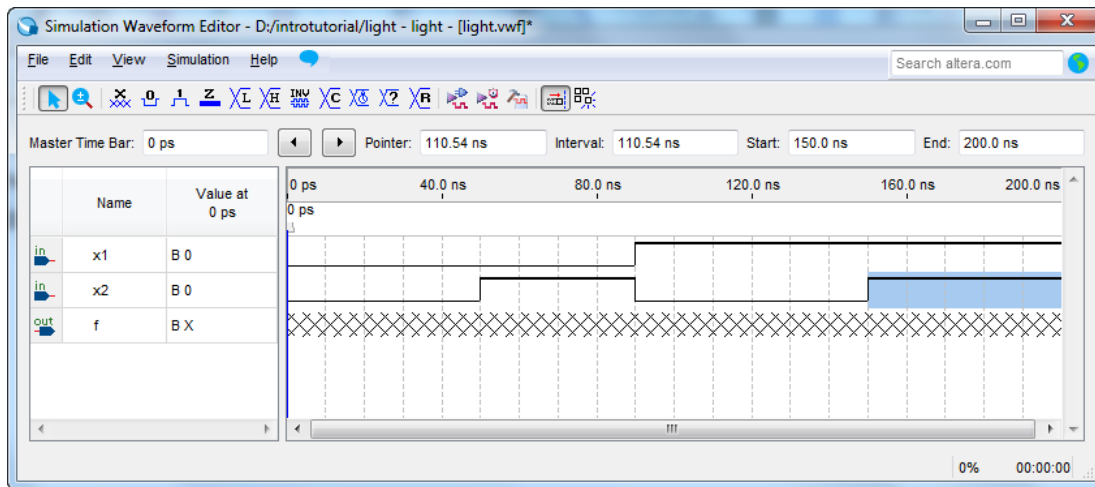




Figure 36. Setting of test values.

## 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 Prime 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  in the Simulation Waveform Editor window. 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 37. Observe that the output *f* is as specified in the truth table of Figure 12.

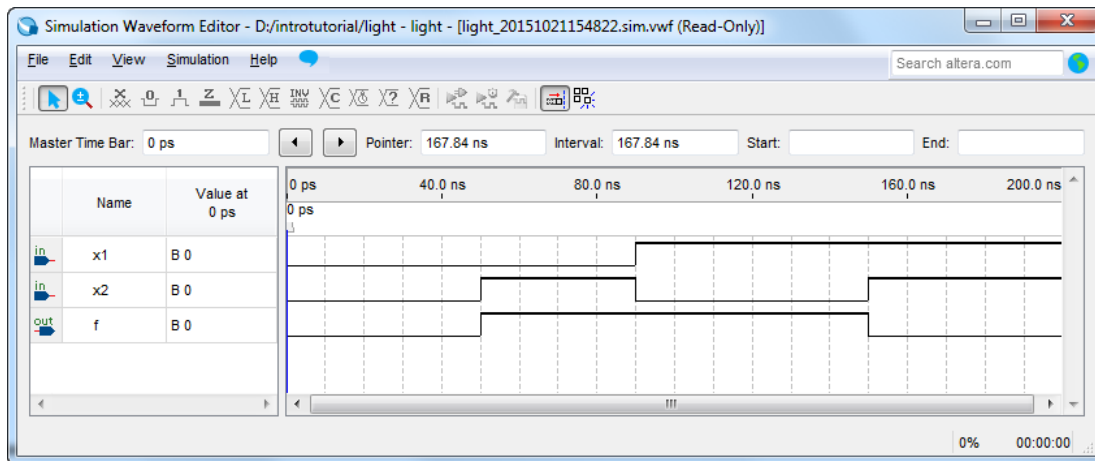




Figure 37. The result of functional simulation.

### 8.1.2 Timing Simulation

**\*NOTE:** because we are using the DE0-CV board, timing simulation is not supported. However, you need to understand what timing simulation is and write about it in your lab documentation. In addition to the following section, information on timing simulation can be found in chapter 1 of the textbook.

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 Prime 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 in the Simulation Waveform Editor window. The simulation should produce the waveforms in Figure 38. 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, *x1* and *x2*, change their values. This delay is due to the propagation delays in the logic element and the wires in the FPGA device.

**Note:** timing simulations are only supported by Cyclone IV and Stratix IV FPGAs. If your DE-series board does not have a Cyclone IV or Stratix IV FPGA, the result of a timing simulation will be identical to the functional simulation.

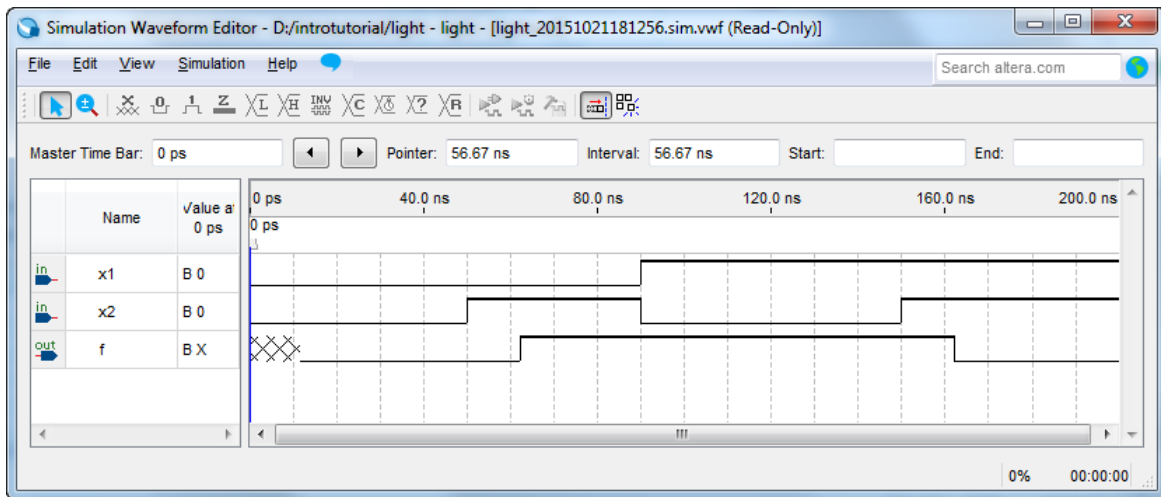


Figure 38. The result of timing simulation.