Quartus II Introduction Using VHDL Design

This tutorial presents an introduction to the Quartus^(R) 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 VHDL design entry method, in which the user specifies the desired circuit in the VHDL hardware description language. Two other versions of this tutorial are also available; one uses the Verilog 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 DE2 Development and Education board connected to a computer that has Quartus II software installed. A reader who does not have access to the DE2 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 8.0; 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 VHDL Design Entry Compiling the Design Pin Assignment Simulating the Designed Circuit Programming and Configuring the FPGA Device Testing the Designed Circuit 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 VHDL 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 VHDL code
- Synthesizing a circuit specified in VHDL 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 DE2 board

1 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.

	Quartus II	
File	Edit View Project	Assignments
D	<u>N</u> ew	Ctrl+N
Ê	Open	Ctrl+O
	⊆lose	Ctrl+F4
A	New Project Wizard	
R	Open Project	Ctrl+J
	Convert MAX+PLUS II F	Project
	Save Projec <u>t</u>	
	Clos <u>e</u> Project	
	Save	Ctrl+S
	Save <u>A</u> s	
	Save Current Report Se	ection As
	<u>File</u> Properties	
	Create / Update	•
	Create / Update Export	•
		∙
	Export	∙
_	Export Convert Programming F	∙
d	Export Convert Programming F Page Setyp	iles Ctrl+P
d	Export Convert Programming F Page Setup Print Prevjew	
d	Export _z Convert Progra <u>m</u> ming F Page Set <u>up</u> Print Pre <u>v</u> iew <u>P</u> rint	

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.

1.1 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. For instance, selecting Help > How to Use Help gives an indication of what type of help is provided.

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.

2 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:

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

D:/introtutorial	project?			
				<u></u>
What is the name of this project?				
light				
What is the name of the top-level des exactly match the entity name in the d		s project? This	name is case se	ensitive and mus
light				

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.

New Project	Vizard: Add Files [page 2 of 5]
	ign files you want to include in the project. Click Add All to add all design files in the ry to the project. Note: you can always add design files to the project later.
<u>F</u> ile name:	Add
File name	Type Library Design entry/sy HDL version Add All
	<u>R</u> emove
	<u>Properties</u>
	Цр
	Down
<	
Specify the p	th names of any non-default libraries. User Libraries
	<back next=""> Finish Cancel</back>

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.

Device family Eamily: Cyclone II			•	Show in 'Available dev Package: Any				
Devices: All			-	Pin <u>c</u> ount:	ĺ	Any		•
Target device				Sp <u>e</u> ed grad	le:	Any		•
Auto device selected	by the Fitter			Show a	idvan	ced de	vices	
Specific device selection	ted in 'Availa:	ble devices	'list	🔲 HardCo		mpatib	le only	
vyailable devices: Name	Core v	LEs	User I/.	Memor	Emt	ed	PLL	^
	Core v	-1				ed	PLL	^
EP2C20F484C8	1.2V	18752	315	239616	52		4	
EP2C20F484I8 EP2C200240C8	1.2V 1.2V	18752 18752	315 142	239616 239616	52 52		4 4	
EP2C35F484C6	1.2V	33216	322	483840	70		4	
EP2C35F484C7	1.2V	33216	322	483840	70		4	
EP2C35F484C8	1.2V	33216	322	483840	70		4	_
EP2C35F484I8	1.2V	33216	322	483840	70		4	
EP2C35F672C6	1.2V	33216	475	483840	70		4	-
CD0008EC7007	1 01/	22216	475	NOCON	70	1	4	>
								-
Companion device								
HardCopy:								-

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 CycloneTM II as the target device family. 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 device called EP2C35F672C6 which is the FPGA used on Altera's DE2 board. Press Next, which opens the window in Figure 8.

w Project V	Vizard: EDA Tool Settings [page 4 of 5]	D
Specify the of	her EDA tools in addition to the Quartus II software used with the pro	iect
Design Entry	y/Synthesis	
Tool name:	<none></none>	•
Format:		-
🗖 Run this	tool automatically to synthesize the current design	
- Simulation -		
Tool name:	<none></none>	•
Format:		-
🔲 Run gat	e-level simulation automatically after compilation	
⊢ Timing Analy	ueie	
Tool name:		•
Format:		-
🗖 Run this	tool automatically after compilation	

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.

New Project Wizard: Summa	rry [page 5 of 5]
When you click Finish, the projec	st will be created with the following settings:
Project directory:	
D:/MyWork/UP Documental	tion/Tutorials/N3-intro-schem/introtutorial/
Project name:	light
Top-level design entity:	light
Number of files added:	0
Number of user libraries added:	0
Device assignments:	
Family name:	Cyclone II
Device:	EP2C35F672C6
EDA tools:	
Design entry/synthesis:	<none></none>
Simulation:	<none></none>
Timing analysis:	<none></none>
Operating conditions:	
Core voltage:	1.2V
Junction temperature range:	0-85 °C
	<back next=""> Finish Cancel</back>

Figure 9. Summary of the project settings.



Figure 10. The Quartus II display for the created project.

3 Design Entry Using VHDL 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 VHDL code in Figure 12. Note that the VHDL entity 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 VHDL entity. The file name must include the extension *vhd*, which indicates a VHDL file. So, we will use the name *light.vhd*.

```
LIBRARY ieee ;

USE ieee.std_logic_1164.all ;

ENTITY light IS

PORT ( x1, x2 : IN STD_LOGIC ;

f : OUT STD_LOGIC ) ;

END light ;

ARCHITECTURE LogicFunction OF light IS

BEGIN

f <= (x1 \text{ AND NOT } x2) \text{ OR (NOT } x1 \text{ AND } x2);

END LogicFunction ;
```

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

3.1 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 VHDL source code file, which we will name *light.vhd*.

Select File > New to get the window in Figure 13, choose VHDL 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 VHDL 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. Maximize the Text Editor window and enter the VHDL code in Figure 12 into it. 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 VHDL code. First, the editor can display different types of VHDL 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 VHDL file.

Save As		
Savejn:	🗀 introtutorial 💌 🔶 📸 🕶	
My Recent Documents Desktop My Documents My Computer	🔁 db	
My Network Places		àave ancel

Figure 14. Name the file.

🅸 light.vhd			
1			
<			>

Figure 15. Text Editor window.

3.1.1 Using VHDL Templates

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

3.2 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 3.1, then the *light.vhd* 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.vhd*, which you created using some other text editor, into the directory *introtutorial*. To add this file to the project, click on the File name: ... button in Figure 16 to get the pop-up window in Figure 17. Select the *light.vhd* 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 VHDL 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.

Select File					
Look in:	introtutorial		•	+ 🗈 💣 🎟]-
My Recent Documents	ight db				
Desktop My Documents					
My Computer					
My Network Places	File name: Files of type:	light Design Files (*.tdf;*.	vhd;*.vhdl;*.v;*.v	▼ /lg; [×] .verilog ▼	Open Cancel

Figure 17. Select the file.

4 Compiling the Designed Circuit

The VHDL code in the file *light.vhd* 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 like that looks like a purple triangle. 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 window showing this report is opened automatically, as seen in Figure 18. The window can be resized, maximized, or 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 O. The report includes a number of sections listed on the left side of its window. 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.

4.1 Errors

Quartus II software displays messages produced during compilation in the Messages window. If the VHDL 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 VHDL 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 VHDL 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 message. 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.vhd*. Remove the semicolon in the statement that defines the function f, illustrating a typographical error that is easily made. Compile the erroneous design file by clicking on the \blacktriangleright icon. A pop-up box will ask if the changes made to the *light.vhd* 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. Expand the Analysis & Synthesis part of the report and then select Messages to have the messages displayed as shown in Figure 20. Double-click on the first error message. Quartus II software responds by opening the *light.vhd* file and highlighting the statement which is affected by the error, as shown in Figure 21. Correct the error and recompile the design.

Compilation Report	Flow Summary	
Legal Notice Flow Summary Flow Settings Flow Non-Default Global Se Flow Icapsed Time Flow Icapsed Time Analysis & Synthesis Summary Summary Messages	Flow Status Quartus II Version Revision Name Top-level Entity Name Family Device Timing Models Met timing requirements Total logic elements Total combinational functions Dedicated logic registers Total registers Total pins Total pins Total memory bits Embedded Multiplier 9-bit elements Total PLLs	Flow Failed - Wed Jun 25 15:46:08 2008 8.0 Build 215 05/23/2008 SJ Full Version light light Cyclone II EP2C35F672C6 Final N/A N/A until Partition Merge N/A until Partition Merge

Figure 19. Compilation report for the failed design.

	Analysis & S	iynthesis Messages
- 😂 🖹 Legal Notice - 🎯 🛅 Flow Summary	Type	Message Info: ******
How Settings How Non-Default Global Settin How Non-Default Global Settin How Lopsed Time Flow Log How Log Analysis & Synthesis Gettings Settings	+ - - - - - - - - - - - - - - - - - - -	Info: Running Quartus II Analysis & Synthesis Info: Command: quartus_mapread_settings_files=onwrite_settings_files=off light -c light Error (10500): VHDL syntax error at light.vhd(9) near text "END"; expecting ";" Info: Found 0 design units, including 0 entities, in source file light.vhd Error: Quartus II Analysis & Synthesis was unsuccessful. 1 error, 0 warnings
- 🚭 🐌 Messages	Processin	g (6) √ Extra In fo 入 In fo (4) 入 Warning 入 Critical Warning 入 Error (2) 入 Flag /
	Message: 0	of 12 🐁 🔮 Location: 🗸 Locate

Figure 20. Error messages.

1	LIBRARY ieee;	
2	USE ieee.std logic 1164.all;	
3	ENTITY light IS	
4	PORT (x1,x2 : IN STD LOGIC;	
5	f : OUT STD LOGIC);	
6	END light;	
7	ARCHITECTURE LogicFunction OF light IS	
8	■ BEGIN f<= (x1 AND NOT x2) OR (NOT x1 AND x2)	
9	END LogicFunction;	

Figure 21. Identifying the location of the error.

5 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 DE2 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 N25 and N26, respectively. We

will connect the output f to the green light-emitting diode labeled $LEDG_0$, which is hardwired to the FPGA pin AE22.

🤣 A.	ssignmen	nt Editor									
-	La Category: Pin V Category: Pin V Category: Pin										
-	X 🔄 🗖 Show assignments for specific nodes:										
⊒+	8										
	Node Filter:							Uncheck All			
	ter:							Delete All			
2											
R∦∏	≚ In	formation: This categ	jory displays all pin assign	nments for the target dev	ice family. Pin assignment	s assign node and entities	s to pins or regions	on the device.			
8	E E	dit: 🗙 🗸 <	<new>></new>								
5		То	Location	I/O Bank	I/O Standard	General Function	Special Function	Reserved			
82 0*	1	< <new>></new>	< <new>></new>								
÷											
٩											
×.											
«»	<										

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. Under Category select Pin. Double-click on the entry <<new>> which is highlighted in blue in the column labeled To. The drop-down menu in Figure 23 will appear. Click on x1 as the first pin to be assigned; this will enter x1 in the displayed table. Follow this by double-clicking on the box to the right of this new x1 entry, in the column labeled Location. Now, the drop-down menu in Figure 24 appears. Scroll down and select PIN_N25. Instead of scrolling down the menu to find the desired pin, you can just type the name of the pin (N25) in the Location box. Use the same procedure to assign input x2 to pin N26 and output f to pin AE22, which results in the image 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.

То	
	-
⊚f	
₽ ×1	
₽ x2	

Figure 23. The drop-down menu displays the input and output names.

Location	I/O Bank		I/O Standar	ď	General Function	Special Fur
PIN_N25	- 5		3.3-V LVTTL		Dedicated Clock	CLK4, LVDS
PIN_M22	I/O Bank 5	Row I/C)	LVDS122p		~
PIN_M23	I/O Bank 5	Row I/C)	LVDS122n		_
PIN_M24	I/O Bank 5	Row I/C)	LVDS125p		
PIN_M25	I/O Bank 5	Row I/C)	LVDS125n		
PIN_N1	I/O Bank 2	Dedicat	ed Clock	CLK1, LVDS	CLKOn, Input	
PIN_N2	I/O Bank 2	Dedicat	ed Clock	CLK0, LVDS0	CLKOp, Input	
PIN_N9	I/O Bank 2	Row I/C)	LVDS31p		
PIN_N18	I/O Bank 5	Row I/C)	LVDS110p		
PIN_N20	I/O Bank 5	Row I/C)	LVDS124p		
PIN_N23	I/O Bank 5	Row I/C)	LVDS126p, [DPCLK7/DQS0R/CQ1R	
PIN_N24	I/O Bank 5	Row I/C)	LVDS126n		
PIN_N25	I/O Bank 5	Dedicat	ed Clock	CLK4, LVD5	CLK2p, Input	
PIN_N26	I/O Bank 5	Dedicat	ed Clock	CLK5, LVDS0	CLK2n, Input	
PIN_P1	I/O Bank 1	Dedicat	ed Clock	CLK3, LVDS	CLK1n, Input	
PIN_P2	I/O Bank 1	Dedicat	ed Clock	CLK2, LVDS	CLK1p, Input	_
PIN_P3	I/O Bank 1	Row I/C)	LVDS26p, DI	PCLK1/DQS1L/CQ1L#	
PIN_P4	I/O Bank 1	Row I/C)	LVDS26n		
PIN_P6	I/O Bank 1	Row I/C)	LVDS22n		
PIN_P7	I/O Bank 1	Row I/C)	LVDS22p		
PIN_P9	I/O Bank 2	Row I/C)	LVDS31n		~

Figure 24. The available pins.

E Category:	Pin			💌 🙀 All 🐧 Timir	ng 🔹 Logic Option				
A Show assignments for specific nodes:									
de F					Uncheck All				
Node Filter:					Delete All				
1	his cell specifies the pin nam	e to which you want to r	nake an assignment> [Double-click to create a nev	w assignment.				
Edit:	<		-	-	-				
Edit:	<pre><<new>></new></pre>	e to which you want to r I/O Bank	I/O Standard	General Function	Special Function				
Edit:	Location		I/O Standard 3.3-V LVTTL	General Function Dedicated Clock	Special Function				
Edit:	Iccation PIN_N25 PIN_N26 PIN_N26		I/O Standard 3.3-V LVTTL 3.3-V LVTTL	General Function Dedicated Clock Dedicated Clock	-				
Edit:	Location		I/O Standard 3.3-V LVTTL	General Function Dedicated Clock	Special Function CLK4, LVDSCLK2p CLK5, LVDSCLK2r				

Figure 25. The complete assignment.

The DE2 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 *comma separated value (CSV)* format, which is a common text file format that contains comma-delimited values. This file format is often used in conjunction with the Microsoft Excel spreadsheet program, but the file can also be created by hand using any plain ASCII text editor. The format for the file for our simple project is

To, Location
x1, PIN_N25
x2, PIN_N26
f, PIN_AE22

By adding lines to the file, any number of pin assignments can be created. Such *csv* 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. Now, select File > Export which leads to the window in Figure 26. Here, the file *light.csv* is available for export. Click on Export. If you now look in the directory *introtutorial*, you will see that the file *light.csv* has been created.

Export					
Save in:	introtutorial		•	+ 🗈 💣 🔳]-
à	db				
My Recent Documents					
Desktop					
My Documents					
My Computer					
My Network	File name:	light		-	Export
Places	Save as type:	Comma Separated V	alue File (* csv)		Cancel

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 *csv* 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.

Import Assignments	X
Specify the source and categories of assignments to import. Click LogicLock Import to select LogicLock Import File(s). Assignment source File name:	ort File Assignments
C Use LogicLock Import File Assignments LogicLock Import File Assignments	Advanced
Copy existing assignments into light.qsf.bak before importing	Cancel

Figure 27. Importing the pin assignment.

For convenience when using large designs, all relevant pin assignments for the DE2 board are given in the file called $DE2_pin_assignments.csv$ in the directory $DE2_tutorials \ design_files$, which is included on the CD-ROM that accompanies the DE2 board and can also be found on Altera's DE2 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 VHDL design file; namely, SW[0], SW[1] and LEDG[0] for x1, x2 and f, respectively. Since these signals are specified in the $DE2_pin_assignments.csv$ file as elements of vectors SW and LEDG, we must refer to them in the same way in the VHDL design file. For example, in the $DE2_pin_assignments.csv$ file the 18 toggle switches are called SW[17] to SW[0]; since VHDL uses parentheses rather than square brackets, these switches are referred to as SW(17) to SW(0). They can also be referred to as an array SW(17 downto 0).

6 Simulating the Designed Circuit

Before implementing the designed circuit in the FPGA chip on the DE2 board, it is prudent to simulate it to ascertain its correctness. Quartus II software includes a simulation tool that 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 Quartus II Waveform Editor to draw the test vectors, as follows:

- 1. Open the Waveform Editor window by selecting File > New, which gives the window shown in Figure 28. Choose Vector Waveform File and click OK.
- 2. The Waveform Editor window is depicted in Figure 29. 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 > 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 30. You may wish to resize the window to its maximum size.



Figure 28. Need to prepare a new file.

D 1	ight.vwf								
Mast	er Time Bar:	15.95 ns	 Image: Pointer: 	4.7 ns	Interval:	-11.25 ns	Start:	End:	
	Name	Value at 15.95 ns	0 ps		10.0 ns		45.05	20.0 ns	
	Name	15.95 ns					15.95 ns 🚽		
<		>							>

Figure 29. The Waveform Editor window.

laster	Time Bar:	15.95 ns	Image: Pointer:	9.68 ns	Interval:	-6.27 ns	Start:	0 ps	End:	200.0 ns
	Name	Value at 15.95 ns	0 ps 15.95 ns	40.0 ns	80.0	ns	120 _, 0 ns		160,0 ns	200.0 r
		5								

Figure 30. 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 31. It is possible to type the name of a signal (pin) into the Name box, but it is easier to click on the button labeled Node Finder to open the window in Figure 32. 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.

Insert Node	or Bus	
Name:		ОК
Туре:	INPUT 💌	Cancel
Value type:	9-Level	Node Finder
Radix:	ASCII	
Bus width:	1	
Start index:	0	
🔲 Display gr	ay code count as binary count	

Figure 31. The Insert Node or Bus dialogue.

N	lode Finder									×
	Named:	▼ Filte	er: Pins: all			•	Customize	Lis	± 👝	ОК
	Look in: Ilight					▼	Include subentities	Sto	• Š	Cancel
	Nodes Found:				Selected Not	les:				
	Name	Assignments	Туре		Name		Assig	gnments	Туре	
	€	PIN_AE22	Output	>	🗩 light x1		PIN_	N25	Input	
	₩x1	PIN_N25	Input	>>	🗩 light x2		PIN_	N26	Input	
	₽ x2	PIN_N26	Input	<	💿 light f		PIN_	AE22	Output	
	<		>	<<	<		п —		>	
										/

Figure 32. Selecting nodes to insert into the Waveform Editor.

Click on the x1 signal in the Nodes Found box in Figure 32, 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 31. This leaves a fully displayed Waveform Editor window, as shown in Figure 33. If you did not select the nodes in the same order as displayed in Figure 33, it is possible to rearrange them. To move a waveform up or down in the Waveform Editor window, click on the node name (in the Name column) and release the mouse button. The waveform is now highlighted to show the selection. Click again on the waveform and drag it up or down in the Waveform Editor.

🖸 lig	ht.vwf							
Master	Time Bar:	15.95 ns	Pointer:	10.06 ns	Interval: -5.89 ns	Start: 0 ps	End:	200.0 ns
	Name	Value at 15.95 ns	0 ps 15.95 ns	40.0 ns	80.0 ns	120 _, 0 ns	160,0 ns	200.0 ns
D 0	x1	AO						
⊡ 1	x2	A O						
-⊙ 2	f	AX		*******		*****	********	******
<		>						

Figure 33. The nodes needed for simulation.

4. 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 View > 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 \Bbbk in the toolbar, or the Waveform Editing Tool, which is activated by the icon \Re .

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 name 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), don't care (DC), 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 Edit menu can also be opened by right-clicking on a waveform name.

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 34. 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.

Master	Time Bar:	0 ps	1	Pointer:	166.25 n	is Interv	al: 166	6.25 ns	Start:		End:	
		Value at	0 ps	20.0 ns	40.0 ns	60.0 ns	80.0 ns	100 _, 0 ns	120,0 ns	140,0 ns	160,0 ns	180,0 ns
	Name	0 ps	0 ps									
▶0	x1	AO										
▶1	x2	A O										
2	f	AX		******	******	*******	******	******	******	******	******	*******

Figure 34. Setting of test values.

6.1 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. This takes much less time, because the simulation can be performed simply by using the logic expressions that define the circuit.

6.1.1 Functional Simulation

To perform the functional simulation, select Assignments > Settings to open the Settings window. On the left side of this window click on Simulator Settings to display the window in Figure 35, choose Functional as the simulation mode, and click OK. The Quartus II simulator takes the inputs and generates the outputs defined in the *light.vwf* file. Before running the functional simulation it is necessary to create the required netlist, which is done by selecting Processing > Generate Functional Simulation Netlist. A simulation run is started by Processing > Start Simulation, or by using the icon \mathbb{R} . At the end of the simulation, Quartus II software indicates its successful completion and displays a Simulation Report illustrated in Figure 36. If your report window does not show the entire simulation time range, click on the report window to select it and choose View > Fit in Window. Observe that the output *f* is as specified in the truth table of Figure 11.

Settings - light	×
Category:	
- Libraries	Simulator Settings
- Device	
🚊 Operating Settings and Conditions	Select simulation options.
Voltage	
Temperature	Simulation mode: Functional
Compilation Process Settings	
- Early Timing Estimate	Simulation input: light.vwf Add Multiple Files
Incremental Compilation	
EDA Tool Settings	Simulation period
- Design Entry/Synthesis	• Bun simulation until all vector stimuli are used
- Simulation	
- Timing Analysis	🖸 End simulation at: 📔 🔤 💌
- Formal Verification	
- Physical Synthesis	Glitch filtering options: Auto
Board-Level	
Analysis & Synthesis Settings	More Settings
VHDL Input	
Verilog HDL Input	
- Default Parameters	
Synthesis Netlist Optimizations	
Fitter Settings	
Physical Synthesis Optimization	
Timing Analysis Settings	
TimeQuest Timing Analyzer	
🖻 Classic Timing Analyzer Setting	
Classic Timing Analyzer Re	Design of the second
Assembler	Description:
Design Assistant	Specifies the type of simulation to perform for the current Simulation focus.
SignalTap II Logic Analyzer	
Logic Analyzer Interface	
Simulator Settings PowerPlay Power Analyzer Setting	
FowerFlay Fower Analyzer Setting	
	OK Cancel

Figure 35. Specifying the simulation mode.

		Waveforms ode: Function									
ß	Master	Time Bar:	0 ps		Pointer:	64.33 ns	Interval:	64.33 ns	Start:	End:	
A ⊛		Name	Value at Ops	0 ps 0 ps 1		40.0 ns	80.0 (ns	120 _, 0 ns	160 _i 0 ns	200.0 ns
<u> </u>	▶0	×1	A O								
Ē	⊡ 1	x2	A 0								
#4	@2	f	A O								

Figure 36. The result of functional simulation.

6.1.2 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. Select Assignments > Settings > Simulator Settings to get to the window in Figure 35, choose Timing as the simulation mode, and click OK. Run the simulator, which should produce the waveforms in Figure 37. Observe that there is a delay of about 6 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.

Simu	ilation m	node: Timing									
ß	Master	Time Bar:	0 ps		Image: Image	109.45 ns	Interval:	109.45 ns	Start:	End:	
A ⊛ ≘		Name	Value at Ops	0 ps 0 ps		40.0 ns	80.0	ns	120 _, 0 ns	160 _, 0 ns	200.0
٩.	₽0	×1	AO								
h	1	х2	AO								
14) 1	2	f	AO								

Figure 37. The result of timing simulation.

7 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 DE2 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 DE2 Board* 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 DE2 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 DE2 board. The RUN position selects the JTAG mode, while the PROG position selects the AS mode.

7.1 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 38. 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 39.

🌲 Hardware S	Setup USB-Blaster [USB	-0]	Mode: JTAG	Progress	0%			
Enable real-ti	ime ISP to allow backgroun	d programming (for MAX II de	vices)					
Mu Start	File	Device	Checksum	Usercode	Program/ Configure	Verify	Blank- Check	Examine
Stop	light.sof	EP2C35F672	002F8284	FFFFFFF				
Auto Detec	t							
🔇 Delete								
Add File								
🖉 Change File								

Figure 38. The Programmer window.

Observe that the configuration file *light.sof* is listed in the window in Figure 38. 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 40.

ardware Settings JTAG Setting Select a programming hardware hardware setup applies only to th	setup to use whe		ces. This programming
Currently selected hardware: Available hardware items:	USB-Blaster (L	JSB-0]	
Hardware USB-Blaster	Local	Port USB-0	Add Hardware

Figure 39. The Hardware Setup window.

🖺 light.cdf*						
🔔 Hardware Setup	USB-Blaster [USB-	0]	Mode: JTAG		Progress:	0%
🔲 Enable real-time I	SP to allow background	programming (for MAX II devi	ces)			
🏓 Start	File	Device	Checksum	Usercode	Program/ Configure Vi	erify Blank- Check Examine
🛍 Stop	light.sof	EP2C35F672	002F8284	FFFFFFF		
Auto Detect						
X Delete						
🚵 Add File						
👺 Change File	<					>

Figure 40. The updated Programmer window.

Now, press Start in the window in Figure 40. 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.

7.2 Active Serial Mode Programming

In this case, the configuration data has to be loaded into the configuration device on the DE2 board, which is identified by the name EPCS16. To specify the required configuration device select Assignments > Device, which leads to the window in Figure 41. Click on the Device and Pin Options button to reach the window in Figure 42. Now, click on the Configuration tab to obtain the window in Figure 43. In the Configuration device box (which may be set to Auto) choose EPCS16 and click OK. Upon returning to the window in Figure 41, click

OK. Recompile the designed circuit.

ategory: General	Device							
Files Libraries Device Operating Settings and Conditions Compilation Process Settings EDA Tool Settings EDA Tool Settings Fitter Settings Timing Analysis Settings Timing Analysis Settings Timing Analysis Settings Design Assistant SignalTap II Logic Analyzer SignalTap II Logic Analyzer Simulator Settings PowerPlay Power Analyzer Settings	Select the family and device you want to target for compilation. Device family Earnily: Cyclone II Devices: All Target device Auto device selected by the Fitter © Specific device selected in 'Available devices' list Oliver: n/a					Show in 'Available devices' list Package: Any Pin gount: Any Speed grade: Any Show advanced devices HardCopy compatible only Device and Pin Options		
	A <u>v</u> ailable devices:	Core v	LEs	User I/	Memor	Embed	PLL 🗸	
	EP2C20Q240C8 EP2C35F484C6 EP2C35F484C7 EP2C35F484C8 EP2C35F484C8 EP2C35F484C8 EP2C35F672C6 EP2C35F672C6 EP2C35F672C7	1.2V 1.2V 1.2V 1.2V 1.2V 1.2V 1.2V 1.2V	18752 33216 33216 33216 33216 33216 33216 33216	142 322 322 322 322 322 475 475	239616 483840 483840 483840 483840 483840 483840 483840	52 70 70 70 70 70 70 70	4 4 4 4 4 4 4 4 2	
	Migration compatibility Migration Devices 0 migration devices select		Companion HardCopy: 🔽 Limit DS		o HardCopy (device reso	v rces	

Figure 41. The Device Settings window.

Device and Pin Options
Pin Placement Error Detection CRC Capacitive Loading Board Trace Model General Configuration Programming Files Unused Pins Dual-Purpose Pins Voltage
Specify general device options. These options are not dependent on the configuration scheme.
Options:
Auto-restart configuration after error
☐ Release clears before tristates ☐ Enable user:supplied start-up clock (CLKUSR) ☐ Enable device-wide reset (DEV_CLRn) ☐ Enable device-wide output enable (DEV_OE) ☐ Enable INIT_DONE output
T Auto usercode
JTAG user code (32-bit hexadecimal): FFFFFFF
In-system programming clamp state:
Delay entry to user mode:
Description:
Directs the device to restart the configuration process automatically if a data error is encountered. If this option is turned off, you must externally direct the device to restart the configuration process if an error occurs.
Reset
OK Cancel

Figure 42. The Options window.

evice ar	nd Pin Options
Pin Plac	ement Error Detection CRC Capacitive Loading Board Trace Mode
General	Configuration Programming Files Unused Pins Dual-Purpose Pins Voltag
HardCop Configur Configur	the device configuration scheme and the configuration device. Note: For ay II, these settings apply to the FPGA prototype device.
🔽 Gen	erate compressed bitstreams EPCS64
Descript	
Specifie the targ	es the configuration device that you want to use as the means of configuring 🦟 let device.
,	Reset
	OK Cancel

Figure 43. 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 38. 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 44 will appear, asking if you want to clear all devices. Click Yes. Now, the Programmer window shown in Figure 45 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 46 will appear. Select the file *light.pof* in the directory *introtutorial* 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 EPCS16 configuration device. 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 47.

Quarti	ıs II
1	Some devices in current device list cannot be added to selected programming mode Active Serial Programming. Do you want to clear all devices in current device list and switch to selected mode?
	Yes No

Figure 44. Clear the previously selected devices.

	ietup USB-Blaster [U	ound programming (for MAX II	devices)		tive Serial Progra		 Progres 		0%	
Start	File	Device	Checksum	Usercode	Program/ Configure	Verify	Blank- Check	Examine	Security Bit	Eras
₩ Stop										
& Auto Detec	t									
(Delete										
Add File	1									
Change File										
Save File										

Figure 45. The Programmer window with Active Serial Programming selected.

Select Program	ming File				×
Look in:	introtutorial		•	+ 🗈 💣	<u>⊞∎</u> -
	🛅 db 💼 light.pof				
My Recent Documents					
Desktop					
>					
My Documents					
My Computer					
N					
My Network Places	File name: Files of type:	light POF Files (*.pof)		- -	Open Cancel
	riles of type:	POP Files (".por)		_	Cancer

Figure 46. Choose the configuration file.

💾 Quartus II - D:	:/introtutorial/ligh	t - light - [light.cd	f*]								_	
<u>File E</u> dit P <u>r</u> ocessin	ng <u>T</u> ools <u>W</u> indow											
Ardware Setup USB-Blaster [USB-0]					Mode: Active Serial Programming Progress:							
Enable real-time ISP to allow background programming (for MAX II devices)												
🟓 Start	File	Device	Checksum	Usercode	Program/ Configure	Verify	Blank- Check E	kamine	Security Bit	Erase	ISP CLAMP	
🖬 Stop	light.pof	EPCS16	1C78FBFF	00000000								
🔑 Auto Detect												
X Delete												
Add File												
👺 Change File												
🔛 Save File												
Add Device												
For Help, press F1	_										NUM	

Figure 47. The updated Programmer window.

Flip the RUN/PROG switch on the DE2 board to the PROG position. Press Start in the window in Figure 47. An LED on the board will light up when the configuration data has been downloaded successfully. Also, the Progress box in Figure 47 will indicate when the configuration and programming process is completed, as shown in Figure 48.

	- <mark>D:/introtutorial/lig</mark> ssing <u>T</u> ools <u>W</u> indow	ght - light - [light.cd	ŋ									
🔔 Hardware Se	etup USB-Blaster [US	B-0] nd programming (for MAX	II devices)		Mod	e: Active	Serial Proj	gramming 💌	Progress	:	100 %	
🏓 Start	File	Device	Checksum	Usercode	Program/ Configure	Verify	Blank- Check	Examine	Security Bit	Erase	ISP CLAMP	
🖬 Stop	light.pof	EPCS16	1C78FBFF	00000000								
📲 Auto Detect												
imes Delete												
🏙 Add File												
Change File.												
Save File												
Madd Device.												
The Up for Help, press F1											NUM	

Figure 48. The Programmer window upon completion of programming.

8 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 VHDL design file, compile the circuit, and program the board as explained above.

Copyright ©2008 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.