

# Quartus® II Introduction for Verilog Users

This tutorial presents an introduction to the Quartus® II software. 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 simple circuit in an Altera® FPGA device.

The Quartus® II system includes full support for all of the popular methods of entering a description of the desired circuit into a CAD system. This tutorial makes use of the Verilog design entry method, in which the user specifies the desired circuit in the Verilog hardware description language. Another version of this tutorial is available that uses VHDL hardware description language.

The screen captures in the tutorial were obtained using Quartus® II version 9.0; if other versions of the software are used, some of the images may be slightly different.

## **Contents:**

- Getting Started
- Starting a New Project
- Design Entry Using Verilog Code
- Compiling the Verilog Code
- Using the RTL Viewer
- Specifying Timing Constraints
- Quartus® II Windows

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.

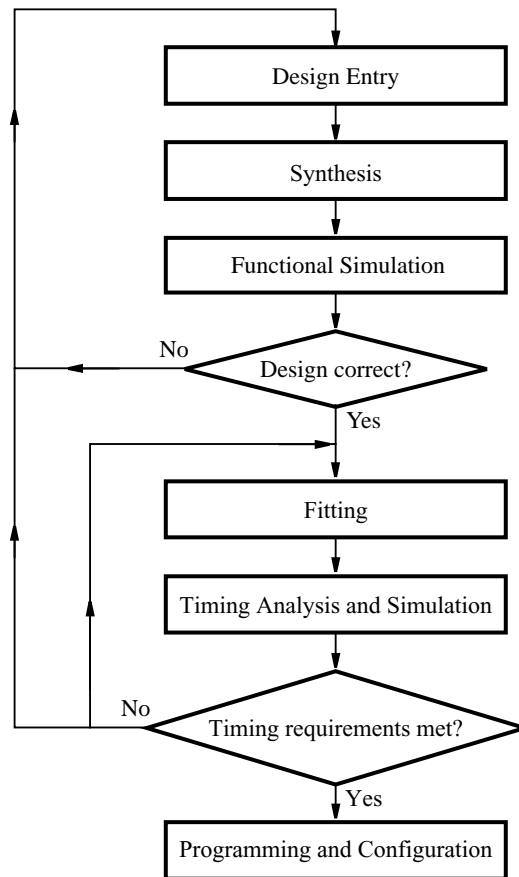


Figure 1: Typical CAD flow.

It involves the following basic steps:

- **Design Entry** – the desired circuit is specified either by using a hardware description language, such as Verilog or VHDL, or by means of a schematic diagram
- **Synthesis** – the CAD Synthesis tool synthesizes the circuit into a netlist that gives the logic elements (LEs) needed to realize the circuit and the connections between the LEs
- **Functional Simulation** – the synthesized circuit is tested to verify its functional correctness; the 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 using the Verilog hardware description language. It makes use of the graphical user interface to invoke the Quartus® II commands. During this tutorial, the reader will learn about:

- Creating a project
- Synthesizing a circuit from Verilog code using the Quartus® II Integrated Synthesis tool
- Fitting a synthesized circuit into an Altera® FPGA
- Examining the report on the results of fitting and timing analysis
- Examining the synthesized circuit in the form of a schematic diagram generated by the RTL Viewer tool
- Making simple timing assignments in the Quartus® II software

# 1 Getting Started

Each logic circuit, or subcircuit, being designed with the 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. As part of the installation of the Quartus® II software, a few sample projects are placed into a directory called `altera\90\qdesigns\vhdl_verilog_tutorial`. To hold the design files for this tutorial, we will use a directory called `quartus_tutorial`. The running example for this tutorial is a simple adder/subtractor circuit, which is defined in the Verilog hardware description language.

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 the Quartus® II software, which the user selects with the computer mouse. Most of the commands provided by the 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 the 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.

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 list of available toolbars, select Tools > Customize > Toolbars. Once a toolbar is opened, it can be moved using the mouse. To see the command associated with an icon, position the mouse over the icon and a tooltip will appear that displays the command name.

It is possible to modify the appearance of the display in Figure 2 in many ways. Section 7 shows how to move, resize, close, and open windows within the main Quartus® II display.

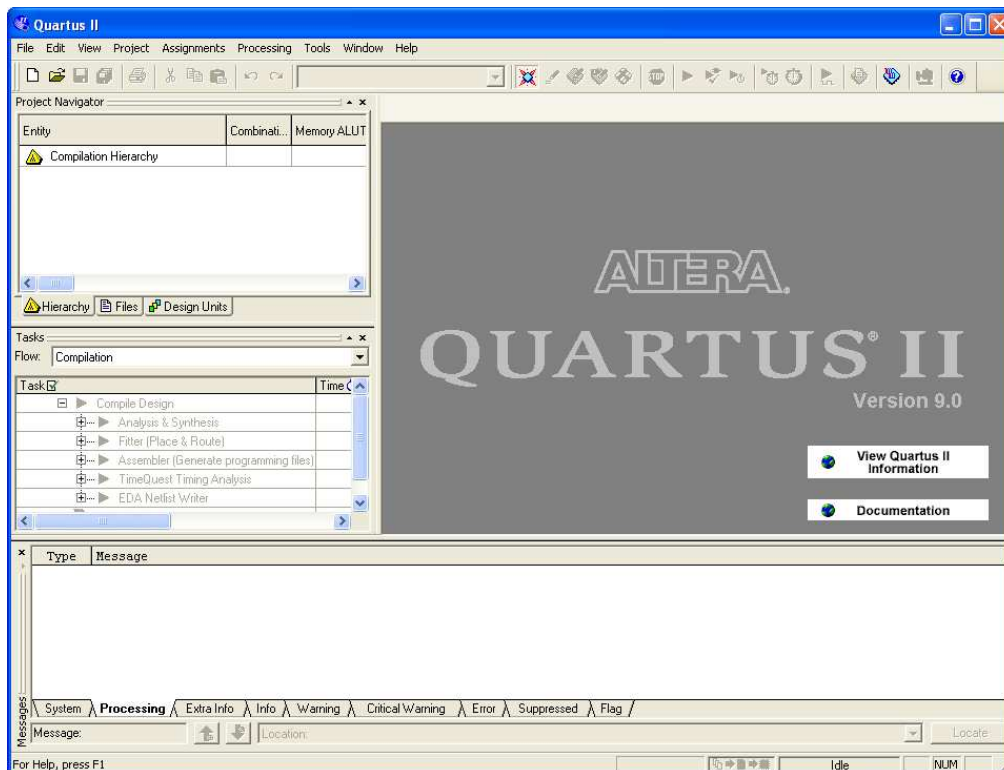


Figure 2: The main Quartus® II display.

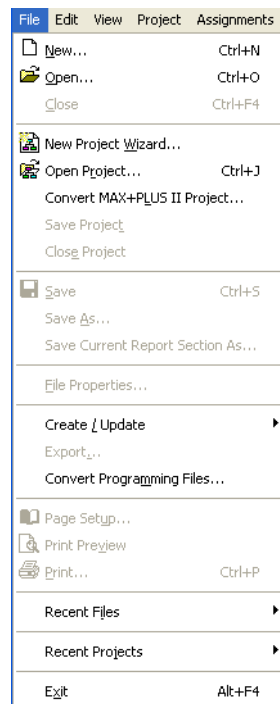


Figure 3: An example of the File menu.

## 1.1 Quartus® II Online Help

The 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 menu in the Help window. 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 about 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*. The Quartus® II software makes the designer's task easy by providing support in the form of a *wizard*.

1. Select **File > New Project Wizard** to reach a window that indicates the capability of this wizard, as shown in Figure 4.

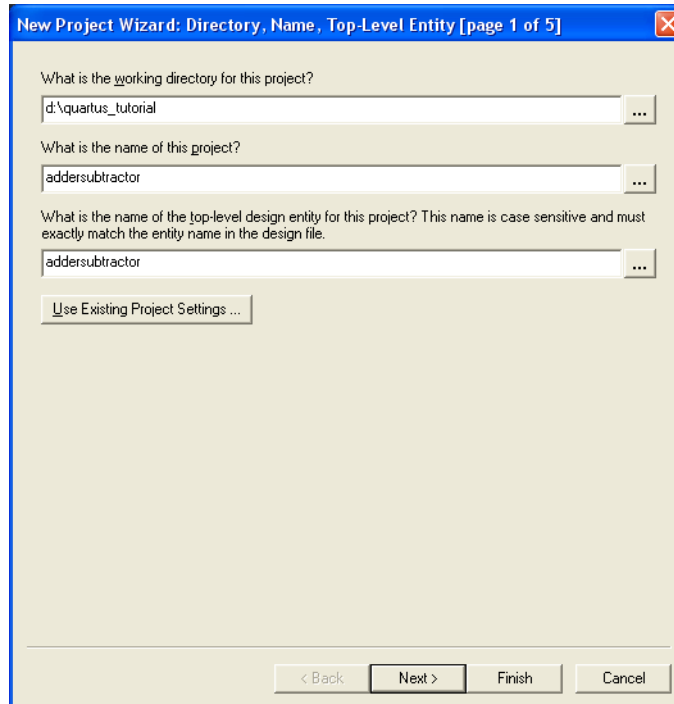


Figure 4: Creation of a new project.

2. Set the working directory to be *quartus\_tutorial*; of course, you can use a directory name of your choice. 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 *addersubtractor* 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 *quartus\_tutorial*, the 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.

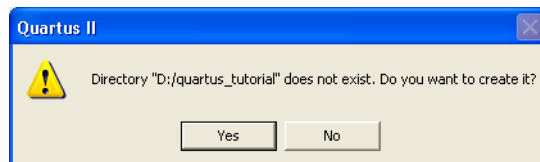


Figure 5: The Quartus® II software can create a new directory for the project.

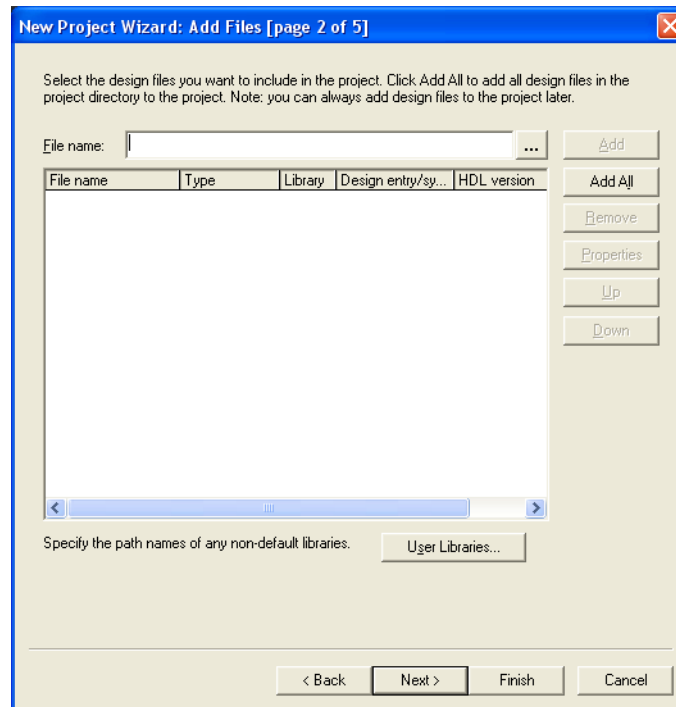


Figure 6: The wizard can include user-specified design files.

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

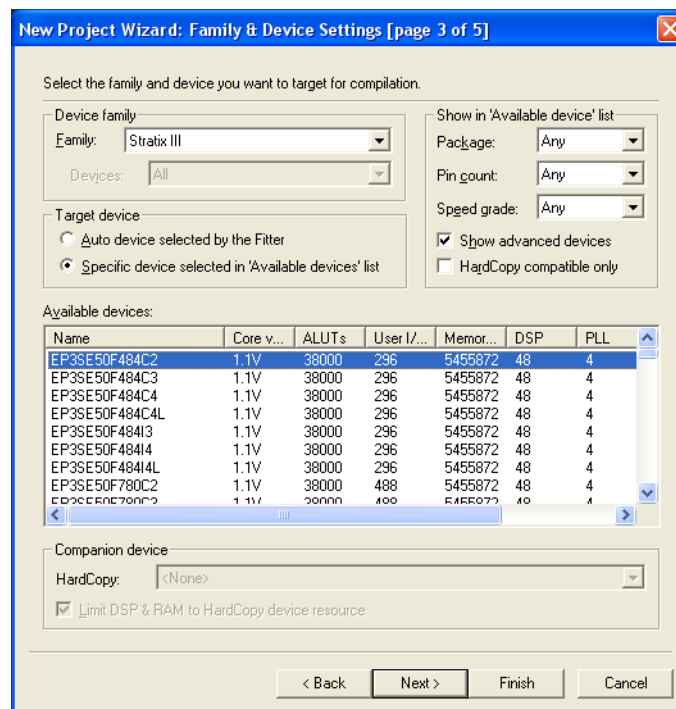


Figure 7: Choose the device family and a specific device.

4. In this window, we can specify the type of device in which the designed circuit will be implemented. Choose

the Stratix III® menu item as the target device family. We can let the 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 EP3SE50F484C2. Press **Next**, which opens the window in Figure 8.

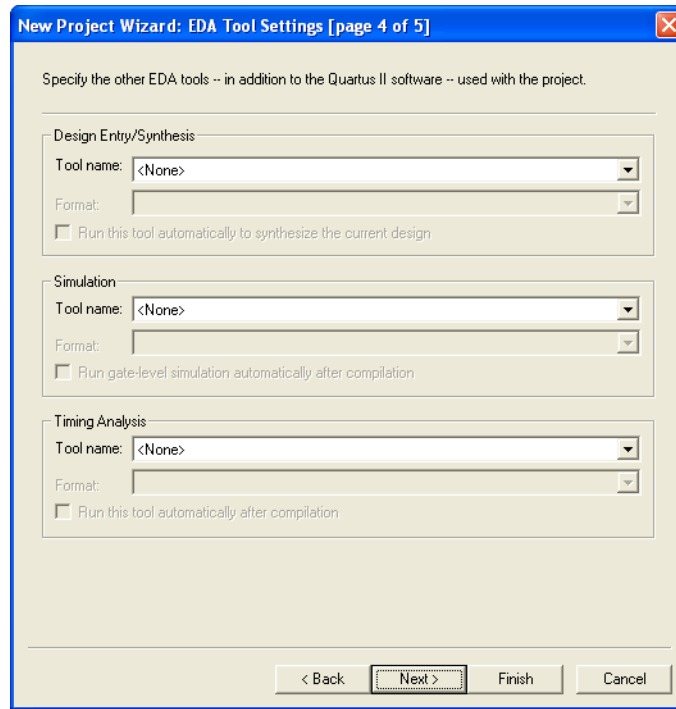


Figure 8: Other EDA tools can be specified.

5. In this window we 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 the Quartus® II messages that refer to third-party tools, which are the tools developed and marketed by companies other than Altera®; other tutorials show how such tools may be used. Since we will rely solely on the Quartus® II tools, we will not choose any other tools. Press **Next**. Now, a summary of the chosen settings appears in the screen shown in Figure 9. Press **Finish**, which returns to the main Quartus® II display. Note that *addersubtractor* is now specified as the current project, as indicated in the title bar at the top of the display. The screen should look similar to that of Figure 10.



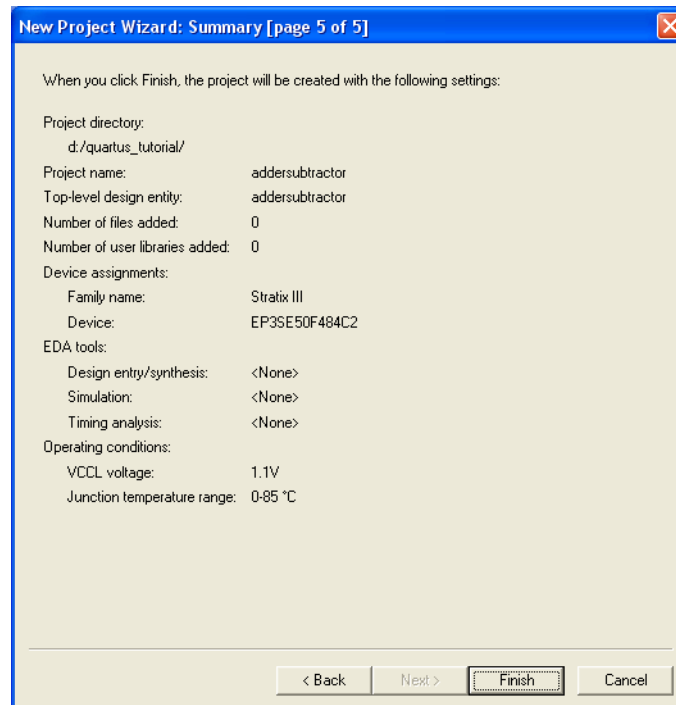


Figure 9: Summary of the project settings.

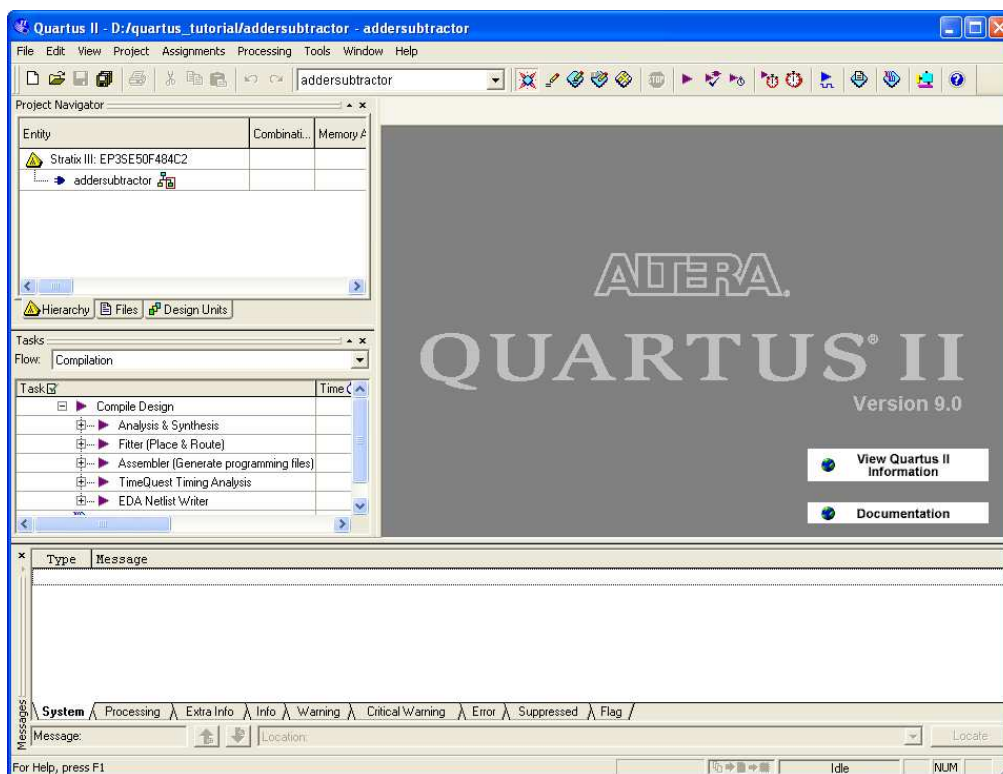


Figure 10: The Quartus® II display for the created project.

### 3 Design Entry Using Verilog Code

As a design example, we will use the adder/subtractor circuit shown in Figure 11. The circuit can add, subtract, and accumulate  $n$ -bit numbers using the 2's complement number representation. The two primary inputs are numbers  $A = a_{n-1}a_{n-2} \cdots a_0$  and  $B = b_{n-1}b_{n-2} \cdots b_0$ , and the primary output is  $Z = z_{n-1}z_{n-2} \cdots z_0$ . Another input is the *AddSub* control signal which causes  $Z = A + B$  to be performed when *AddSub* = 0 and  $Z = A - B$  when *AddSub* = 1. A second control input, *Sel*, is used to select the accumulator mode of operation. If *Sel* = 0, the operation  $Z = A \pm B$  is performed, but if *Sel* = 1, then  $B$  is added to or subtracted from the current value of  $Z$ . If the addition or subtraction operations result in arithmetic overflow, an output signal, *Overflow*, is asserted.

To make it easier to deal with asynchronous input signals, we will load them into flip-flops on a positive edge of the clock. Thus, inputs  $A$  and  $B$  will be loaded into registers *Areg* and *Breg*, while *Sel* and *AddSub* will be loaded into flip-flops *SelR* and *AddSubR*, respectively. The adder/subtractor circuit places the result into register *Zreg*.

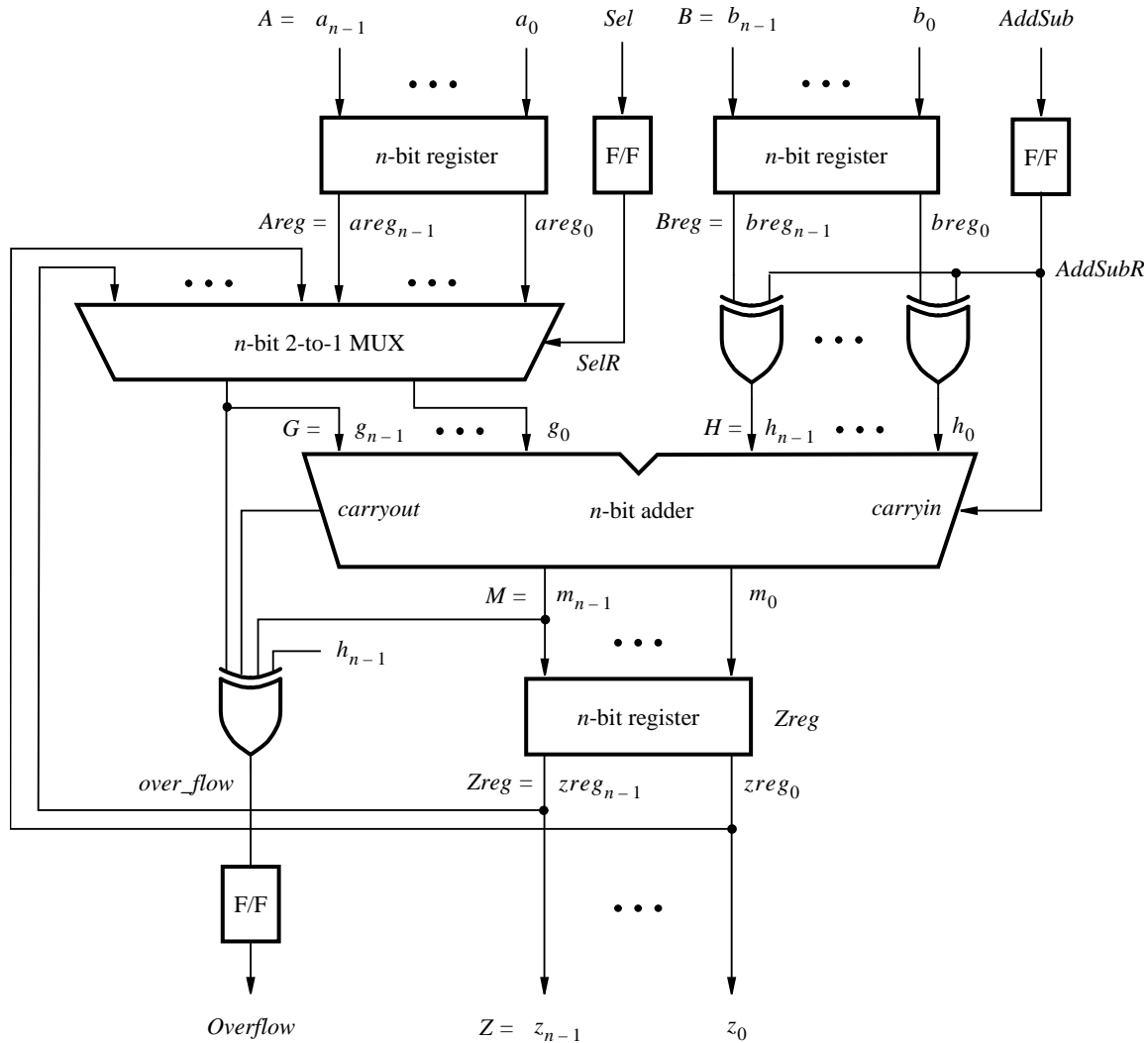


Figure 11: The adder/subtractor circuit.

The required circuit is described by the Verilog code in Figure 12. For our example, we will use a 16-bit circuit as specified by  $n = 16$ .

```
// Top-level module
module addersubtractor (A, B, Clock, Reset, Sel, AddSub, Z, Overflow);
    parameter n = 16;
    input  [n-1:0] A, B;
    input  Clock, Reset, Sel, AddSub;
    output [n-1:0] Z;
    output Overflow;
    reg SelR, AddSubR, Overflow;
    reg [n-1:0] Areg, Breg, Zreg;
    wire [n-1:0] G, H, M, Z;
    wire carryout, over_flow;

    // Define combinational logic circuit
    assign H = Breg ^ {n{AddSubR}};
    mux2to1 multiplexer (Areg, Z, SelR, G);
    defparam multiplexer.k = n;
    adder nbit_adder (AddSubR, G, H, M, carryout);
    defparam nbit_adder.k = n;
    assign over_flow = carryout ^ G[n-1] ^ H[n-1] ^ M[n-1];
    assign Z = Zreg;

    // Define flip-flops and registers
    always @(posedge Reset or posedge Clock)
        if (Reset == 1)
            begin
                Areg <= 0; Breg <= 0; Zreg <= 0;
                SelR <= 0; AddSubR <= 0; Overflow <= 0;
            end
        else
            begin
                Areg <= A; Breg <= B; Zreg <= M;
                SelR <= Sel; AddSubR <= AddSub; Overflow <= over_flow;
            end
    end
endmodule

// k-bit 2-to-1 multiplexer
module mux2to1 (V, W, Sel, F);
    parameter k = 8;
    input  [k-1:0] V, W;
    input  Sel;
    output [k-1:0] F;
    reg [k-1:0] F;

    always @(V or W or Sel)
        if (Sel == 0) F = V;
        else F = W;
endmodule
```

... continued in Part *b*

Figure 12: Verilog code for the circuit in Figure 11 (Part *a*)

```
// k-bit adder
module adderk (carryin, X, Y, S, carryout);
    parameter k = 8;
    input  [k-1:0] X, Y;
    input  carryin;
    output [k-1:0] S;
    output carryout;
    reg    [k-1:0] S;
    reg    carryout;

    always @(X or Y or carryin)
        {carryout, S} = X + Y + carryin;
endmodule
```

Figure 12: Verilog code for the circuit in Figure 11 (Part b).

Note that the top Verilog module is called *addersubtractor* to match the name given in Figure 4, which was specified when the project was created. This code can be typed into a file by using any text editor that stores ASCII files, or by using the Quartus® II text editing facilities. While the file can be given any name, it is a common designers' practice to use the same name as the name of the top-level Verilog module. The file name must include the extension *v*, which indicates a Verilog file. So, we will use the name *addersubtractor.v*. For convenience, we have provided the required file in the directory *altera\90\qdesigns\vhdl\_verilog\_tutorial*. Copy this file into the project directory *quartus\_tutorial*.

### 3.1 Using the Quartus® II Text Editor

This section demonstrates how to use the Quartus® II Text Editor. You can skip this section if you prefer to use another text editor to create the *addersubtractor.v* file, or if you have chosen to copy the file from the *altera\90\qdesigns\vhdl\_verilog\_tutorial* directory.

1. Select File > New to get the window in Figure 13, choose Verilog HDL File, and click OK. This opens the Text Editor window.

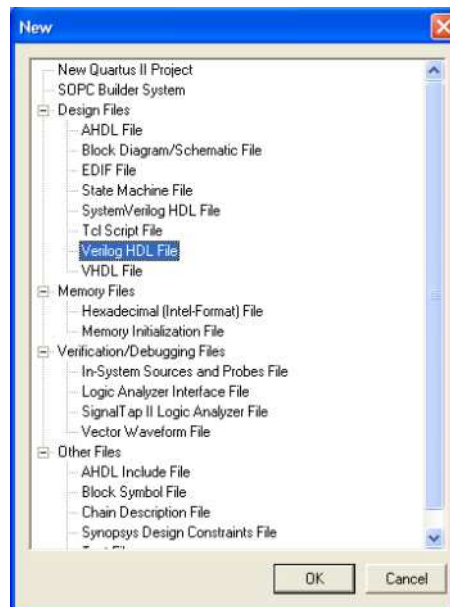


Figure 13: Choose to prepare a Verilog file.

2. 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 shown in Figure 14. In the field labeled **Save as type** choose **Verilog HDL File**. In the field labeled **File name** type *addersubtractor*. Put a checkmark in the box **Add file to current project**. Click **Save**, which puts the file into the directory *quartus\_tutorial* and leads to the Text Editor window shown in Figure 15.

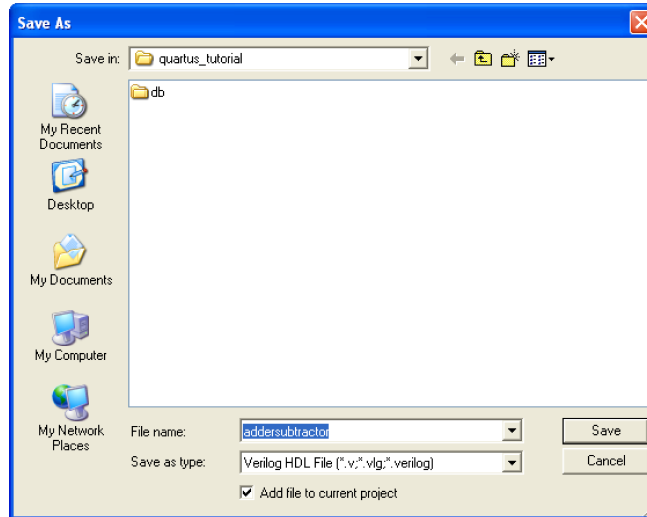


Figure 14: Name the file.

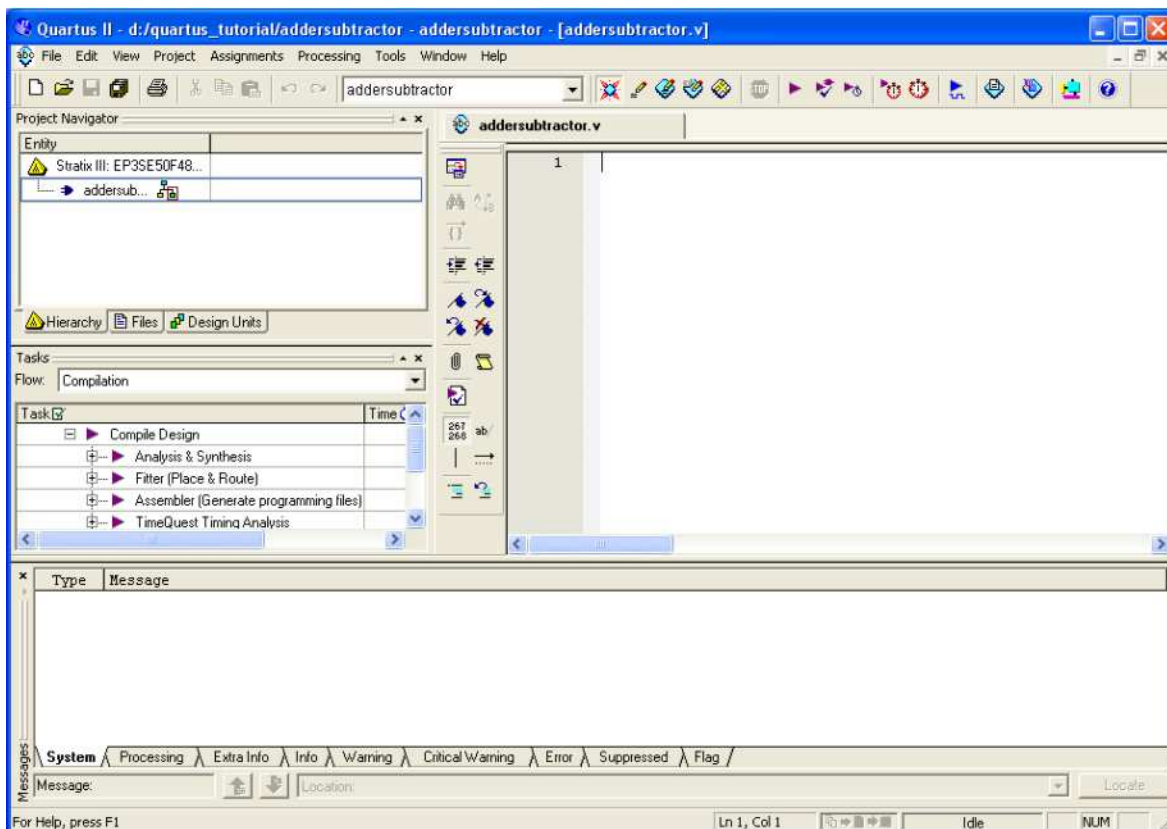


Figure 15: The Quartus® II display after saving the file.

3. Enter the Verilog code in Figure 12 into the Text Editor Window, which is located on the right side of the screen. Save the file by going to File > Save, or by typing the shortcut Ctrl-s.

Most of the commands available in the Text Editor are self-explanatory. Text is entered at the *insertion point*, which is indicated by a thin vertical line. The insertion point can be moved either by using the keyboard arrow keys or by using the mouse. Two features of the Text Editor are especially convenient for typing Verilog code. First, the editor can display different types of Verilog statements in different colors, which is the default choice. Second, the editor can automatically indent the text on a new line so that it matches the previous line. Such options can be controlled by the settings in Tools > Options > Text Editor, as shown in Figure 16.

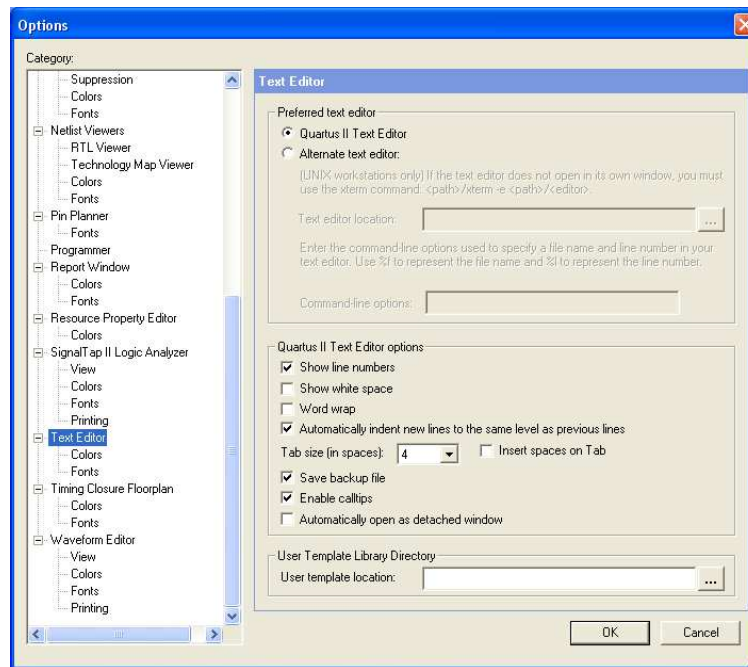


Figure 16: Text Editor Options.

### 3.1.1 Using Verilog Templates

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

## 3.2 Adding Design Files to a Project

As we indicated when discussing Figure 6, you can tell the 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 *addersubtractor* project, select Assignments > Settings > Files, which leads to a window similar to the window in Figure 17. An alternative way of making this selection is to go to 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 *addersubtractor.v* file is already a part of the project and will be listed in the window in Figure 17. Otherwise, the file must be added to the project.

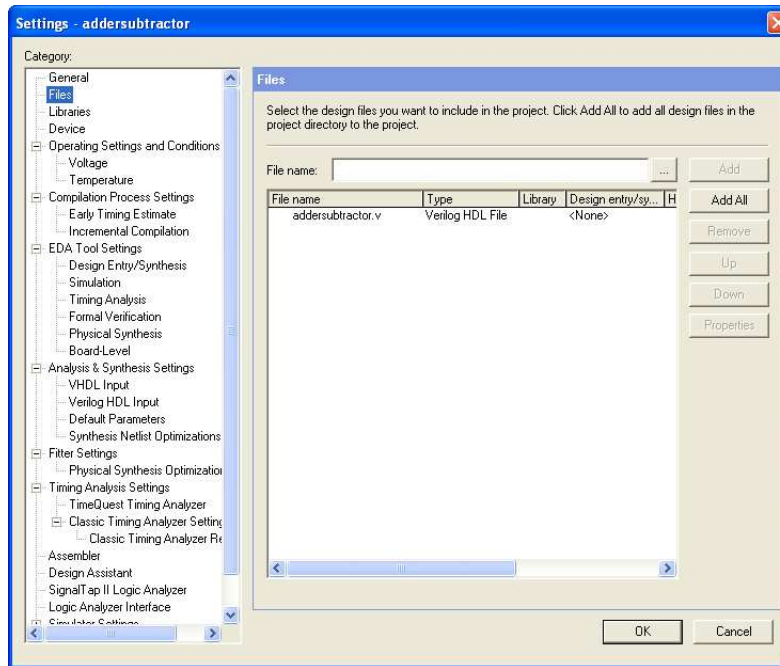


Figure 17: Settings window.

1. If not already done, place a copy of the file *addersubtractor.v* into the directory *quartus\_tutorial* by getting it from the directory *altera\90\qdesigns\vhdl\_verilog\_tutorial*, or by using a file that you created using another text editor.
2. To add this file to the project, click on the ... button beside the File name field in Figure 17 to get the pop-up window in Figure 18.

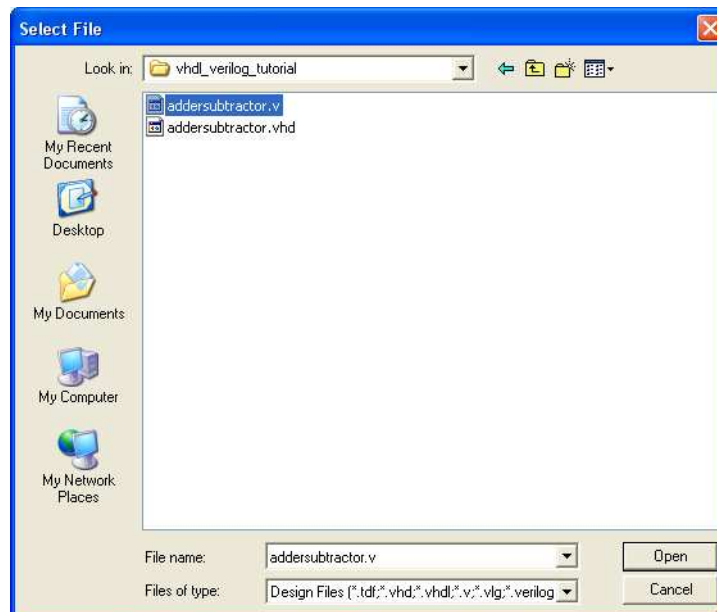


Figure 18: Select the file.


3. Select the *addersubtractor.v* file and click **Open**. The selected file is now indicated in the File name field of Figure 17. Click **Add** and then **OK** to include the *addersubtractor.v* file in the project.

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



## 4 Compiling the Verilog Code

The Verilog code is processed by several Quartus® II tools that analyze the code and generate an implementation of it for the target chip. These tools are controlled by the application program called the *Compiler*.

1. Run the Compiler by selecting **Processing > Start Compilation**, or by using the toolbar icon . As the compilation moves through various stages, its progress is reported in the Tasks window on the left side. This window also provides a comprehensive interface to edit, start, and monitor different stages of the compilation. Successful (or unsuccessful) compilation is indicated in a pop-up box. Acknowledge it by clicking OK. This leads to the Quartus® II display in Figure 19, in which we have expanded the Entity hierarchy in the top left corner to show all modules in the *addersubtractor* design. In the message window, located at the bottom of the display, various messages are shown. In case of errors, there will be appropriate messages given.

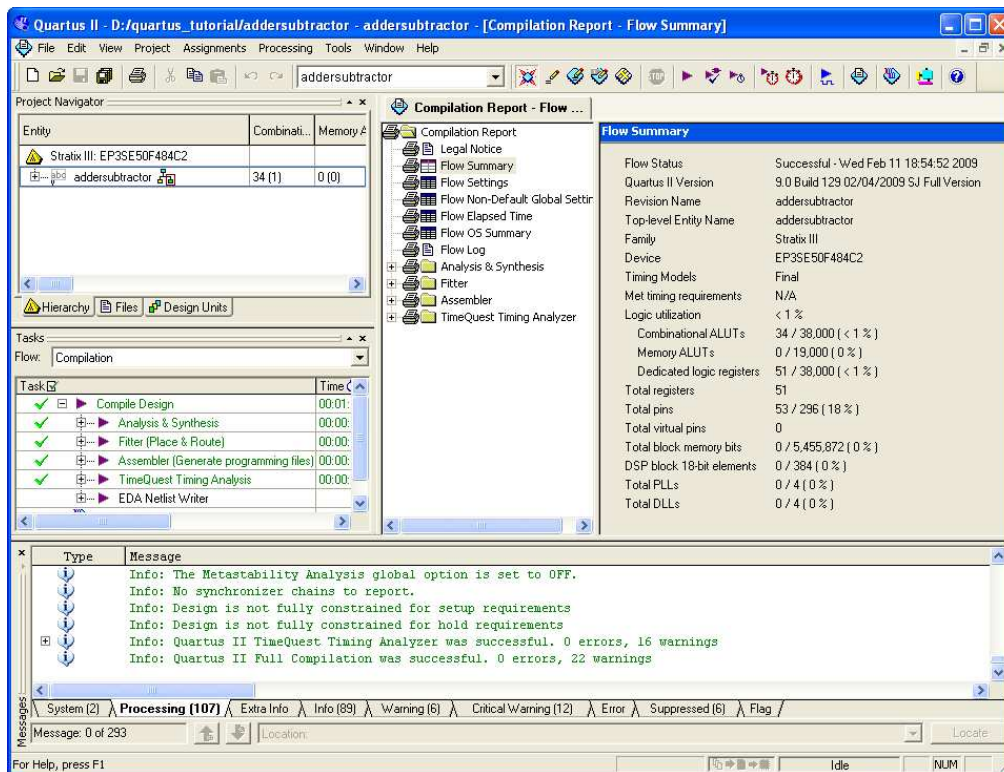



Figure 19: Display after a successful compilation.

2. When the compilation is finished, a compilation report is produced. A window showing this report, displayed in Figure 20, is opened automatically. 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  in the toolbar. The report includes a number of sections listed on the left side of its window. Figure 20 shows the Compiler Flow Summary section, which indicates that only a miniscule amount of chip resources are needed to implement this tiny circuit on the selected FPGA chip.

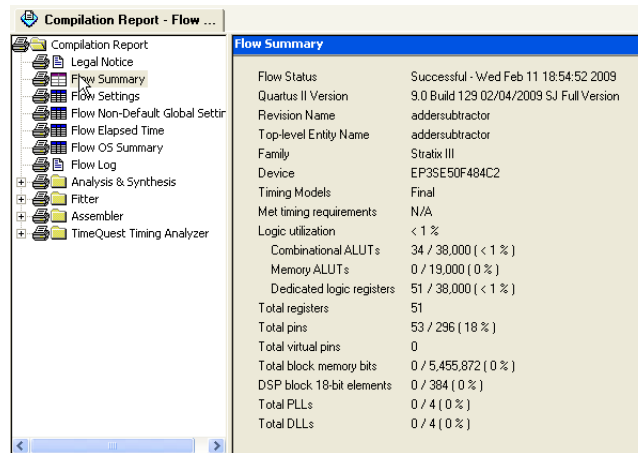


Figure 20: Compilation report.

The Compilation Report provides a lot of information that may be of interest to the designer, such as the speed of the implemented circuit. A good measure of the speed is the maximum frequency at which the circuit can be clocked, referred to as  $f_{max}$ . This measure depends on the longest delay along any path between two registers clocked by the same clock. The Quartus® II software performs a timing analysis to determine the expected performance of the circuit. It evaluates several parameters, which are listed in the TimeQuest Timing Analyzer section of the Compilation Report.

3. Click on the small + symbol next to TimeQuest Timing Analyzer to expand this section of the report, as shown in Figure 21. Notice there are multiple models included, which describe the performance of the circuit under different operating conditions. Expand the report for Slow 1100mV 85C Model by clicking on the small + symbol next to it and click on the item Fmax Summary to display the table in Figure 21. The table shows that the maximum frequency for our circuit implemented on the specified chip is 407.5 MHz. You may get a different value of  $f_{max}$ , dependent on the specific version of the Quartus® II software installed on your computer.

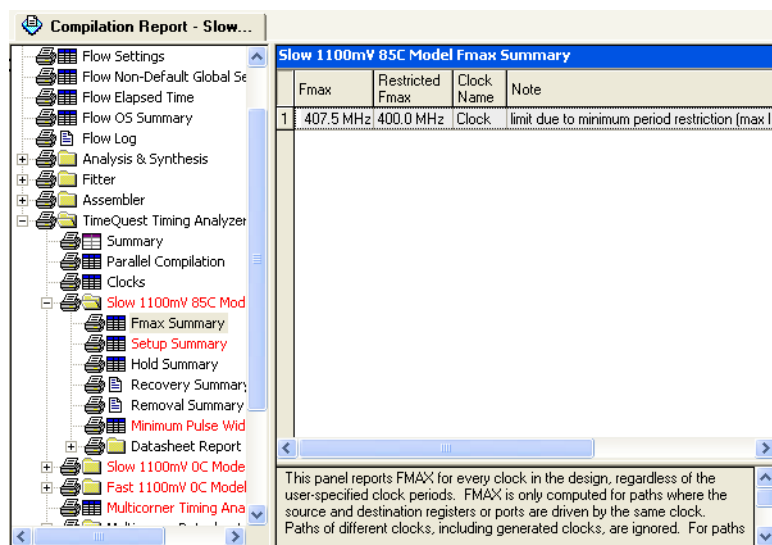
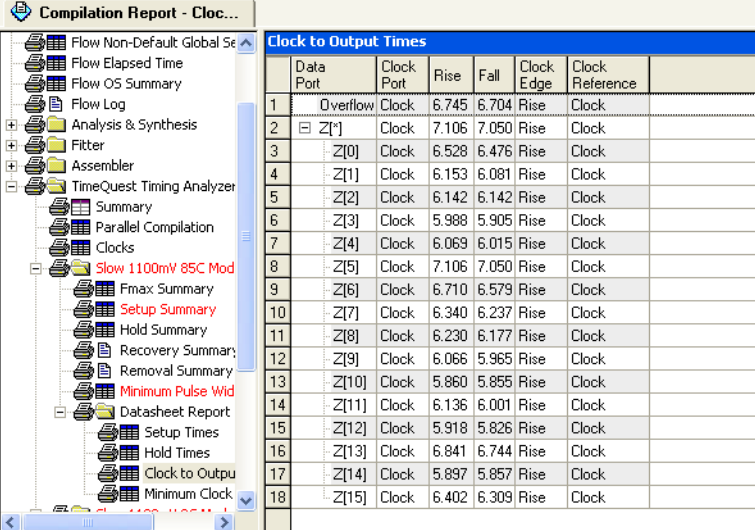


Figure 21: Fmax Summary of TimeQuest Timing Analysis.



4. While  $f_{max}$  is a function of the longest propagation delay between two registers in the circuit, it does not indicate the delays with which output signals appear at the pins of the chip. Time elapsed from an active

edge of the clock signal at the clock source until a corresponding output signal is produced (from a flip-flop) at an output pin is denoted as the *Clock to Output Time* at that pin. To see this parameter, expand Datasheet Report under the Slow 1100mV 85C Model heading and select Clock to Output Times to obtain the display in Figure 22. For each output signal, the delays for rise edge and fall edge are listed. The clock signal and its active edge are also shown in the table. Two other parameters listed in the Datasheet Report are *Setup Time* and *Hold Time*. The *Setup Time* measures the length of time for which data that feeds a register must be present at an input pin before the clock signal is asserted at the clock pin. The *Hold Time* measures the minimum length of time for which data that feeds a register must be retained at an input pin after the clock signal is asserted at the clock pin.



	Data Port	Clock Port	Rise	Fall	Clock Edge	Clock Reference
1	Overflow	Clock	6.745	6.704	Rise	Clock
2	Z[*]	Clock	7.106	7.050	Rise	Clock
3	Z[0]	Clock	6.528	6.476	Rise	Clock
4	Z[1]	Clock	6.153	6.081	Rise	Clock
5	Z[2]	Clock	6.142	6.142	Rise	Clock
6	Z[3]	Clock	5.988	5.905	Rise	Clock
7	Z[4]	Clock	6.069	6.015	Rise	Clock
8	Z[5]	Clock	7.106	7.050	Rise	Clock
9	Z[6]	Clock	6.710	6.579	Rise	Clock
10	Z[7]	Clock	6.340	6.237	Rise	Clock
11	Z[8]	Clock	6.230	6.177	Rise	Clock
12	Z[9]	Clock	6.066	5.965	Rise	Clock
13	Z[10]	Clock	5.860	5.855	Rise	Clock
14	Z[11]	Clock	6.136	6.001	Rise	Clock
15	Z[12]	Clock	5.918	5.826	Rise	Clock
16	Z[13]	Clock	6.841	6.744	Rise	Clock
17	Z[14]	Clock	5.897	5.857	Rise	Clock
18	Z[15]	Clock	6.402	6.309	Rise	Clock

Figure 22: The *Clock to Output Time* delays.

5. An indication of where the circuit is implemented on the chip is available by selecting Tools > Chip Planner(Floor Plan and Chip Editor), or by clicking on the icon . This opens the Chip Planner display, as shown in Figure 23. This display highlights the location of the logic elements used to implement the circuit. To make the image appear as shown in Figure 23 you may have to select View > Fit in Window (shortcut Ctrl-w), and it can be expanded to fill the screen by clicking the Full Screen icon  in the left hand toolbar.

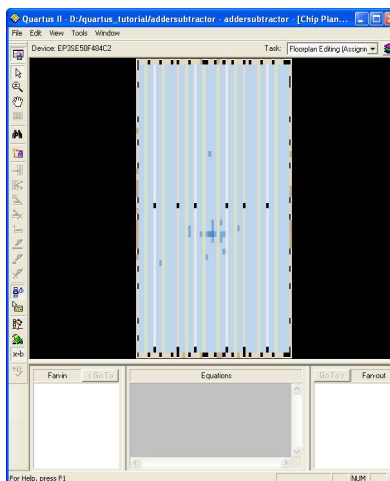



Figure 23: View of the floorplan.

- A Zoom Tool, activated by the icon  in the left hand toolbar, can be used to enlarge parts of the image even more. You can click and drag a box over an area of the chip to quickly zoom into that part of the chip. Figure 24 shows a zoomed-in view of the floorplan that highlights the implemented circuit. By positioning the cursor on any logic element, the designer can see what part of the circuit is implemented in this resource. The chip planner tool has several icons that can be used to view aspects such as fan-in and fan-out of nodes, connecting paths between nodes, and so on. For more information on using this tool, refer to Help by selecting Help > Contents > Achieving Timing Closure > Working With Assignments in the Chip Planner from the main Quartus® II display.

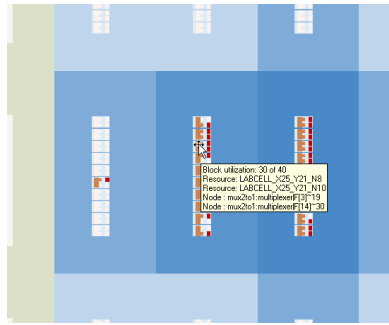


Figure 24: A portion of the expanded view.

## 4.1 Errors

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

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

- To see the effect of an error, open the file *addersubtractor.v*. Line 21 has the statement

```
assign H = Breg ^ {n{AddSubR}};
```

Remove the semicolon in this statement, illustrating a typographical error that is easily made. Compile the erroneous design file. The Quartus® II software displays a pop-up box indicating that the compilation was not successful. Acknowledge it by clicking OK. The compilation report summary, given in Figure 25, now confirms the failed result.

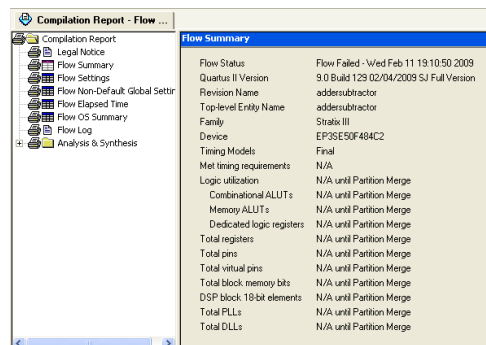


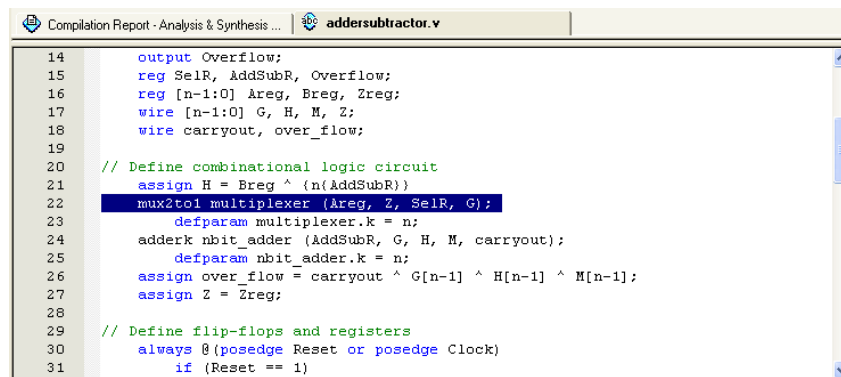
Figure 25: Compilation report for the failed design.

2. In this window, Click on Analysis & Synthesis > Messages to have all messages displayed as shown in Figure 26.

Type	Message
Info	*****
Info	Running Quartus II Analysis & Synthesis
Info	Command: quartus map --lower_priority --read_settings_files-on --write_settings_files-off addsubtractor -c addsubtractor
Error	Error (10170): Verilog HDL syntax error at addsubtractor.v(22) near text "mux2tol"; expecting ";", or ";
Error	Error (10112): Ignored design unit "addsubtractor" at addsubtractor.v(9) due to previous errors
Error	Error (10112): Ignored design unit "mux2tol" at addsubtractor.v(44) due to previous errors
Error	Error (10112): Ignored design unit "adderk" at addsubtractor.v(59) due to previous errors
Info	Found 0 design units, including 0 entities, in source file addsubtractor.v
Error	Error: Quartus II Analysis & Synthesis was unsuccessful. 4 errors, 0 warnings
Error	Error: Quartus II Full Compilation was unsuccessful. 6 errors, 0 warnings

Figure 26: Error messages.

3. Double-click on the first error message, which states that there is a Verilog syntax error. The Quartus® II software responds by opening the *addsubtractor.v* file and highlighting the statement affected by the error, as shown in Figure 27. Correct the error and recompile the design.



```

14     output Overflow;
15     reg SelR, AddSubR, Overflow;
16     reg [n-1:0] Areg, Breg, Zreg;
17     wire [n-1:0] G, H, M, Z;
18     wire carryout, over_flow;
19
20     // Define combinational logic circuit
21     assign H = Breg ^ {n{AddSubR}}
22     mux2tol multiplexer (Areg, Z, SelR, G);
23     defparam multiplexer.k = n;
24     adderk nbit_adder (AddSubR, G, H, M, carryout);
25     defparam nbit_adder.k = n;
26     assign over_flow = carryout ^ G[n-1] ^ H[n-1] ^ M[n-1];
27     assign Z = Zreg;
28
29     // Define flip-flops and registers
30     always @(posedge Reset or posedge Clock)
31         if (Reset == 1)

```

Figure 27: Identifying the location of the error.

## 5 Using the RTL Viewer

The Quartus® II software includes a tool that can display a schematic diagram of the designed circuit. The display is at the Register Transfer Level of detail, and the tool is called the *RTL Viewer*.

1. Click Tools > Netlist Viewers > RTL Viewer, to reach the window shown in Figure 28.

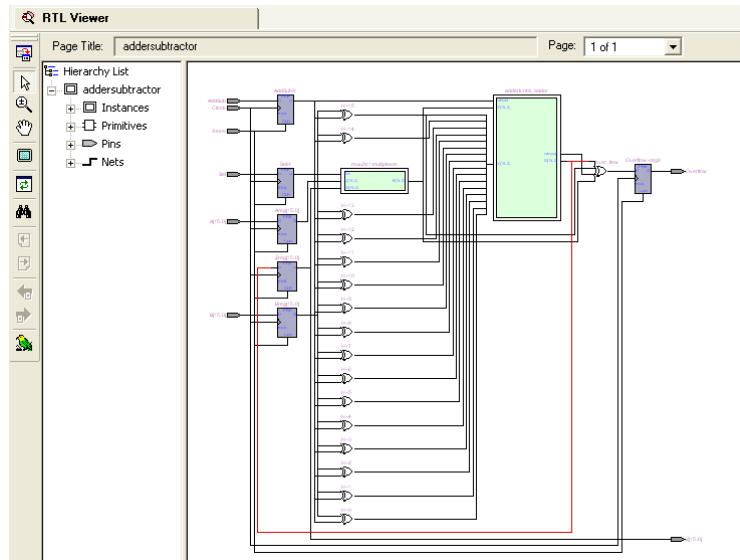


Figure 28: The *addersubtractor* circuit displayed by the RTL Viewer.

The displayed image shows the structure of the entire *addersubtractor* circuit. The inputs to the circuit, shown on the left side, are registered. The two subcircuits, defined by the *mux2to1* and *adderk* modules, are drawn as shaded boxes and their respective names appear above the boxes. The remainder of the circuit are the XOR gates used to complement the *B* vector when subtraction is performed, and the circuitry needed to generate the *Overflow* signal.

2. Use the Zoom Tool, located in the left hand toolbar, to enlarge the image and view the upper-left portion of the circuit, as illustrated in Figure 29. Note that individual flip-flops are used for the *AddSub* and *Sel* signals. Sixteen-bit vectors *A* and *B* are denoted by heavy lines connected to the registers, *Areg* and *Breg*, which are indicated as heavily outlined flip-flop symbols. The *Zreg* register is drawn in the same manner as *Areg* and *Breg*.

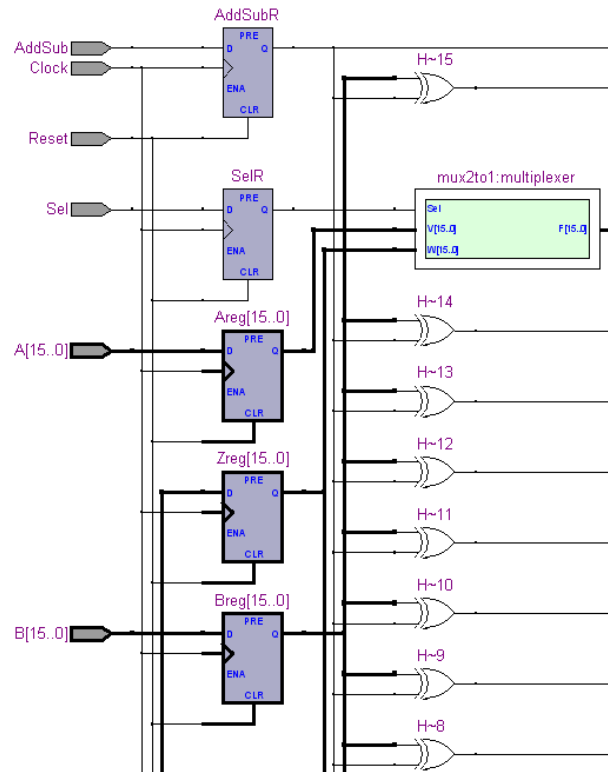


Figure 29: An enlarged view of the circuit.

- Details of subcircuits can be seen by clicking on the box that represents a subcircuit. Select the Selection Tool from the left hand toolbar (above the Zoom Tool), and double-click on the *mux2to1* box to obtain the image in Figure 30. It shows the multiplexers used to choose either the *Areg* or *Z* vector as one of the inputs to the adder, under control of the *Sel* signal. Observe that the multiplexer data inputs are labeled as specified in the Verilog code for the *mux2to1* module in part *b* of Figure 12, namely as *V* and *W* rather than *Areg* and *Z*.

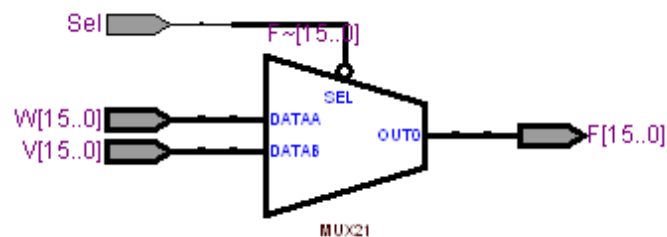



Figure 30: The multiplexer subcircuit.

The RTL viewer is a useful tool. It can be used effectively to facilitate the development of Verilog code for a circuit that is being designed. It provides a pictorial feedback to the designer, which gives an indication of the structure of the circuit that the code will produce. Viewing the pictures makes it easy to spot missing elements, wrong connections, and other typical errors that one makes early in the design process.



## 6 Specifying Timing Constraints

The Quartus® II software allows the user to specify timing constraints for the designed circuit.

1. Open TimeQuest Timing Analyzer by selecting Tools > TimeQuest Timing Analyzer, or by clicking the icon . Figure 31 shows the interface of TimeQuest Timing Analyzer. It is a very powerful tool for the designer to create, manage, and analyze timing constraints, and to quickly perform timing verification for their design. The compilation in Section 4 produced the  $f_{max}$  of 407.5 MHz, which translates to a minimum period of 2.4540ns. Suppose that we need a circuit that can operate at a clock frequency of 450 MHz. We can use the TimeQuest Timing Analyzer tool to create a new SDC (Synopsys Design Constraints) file containing the clock constraint.

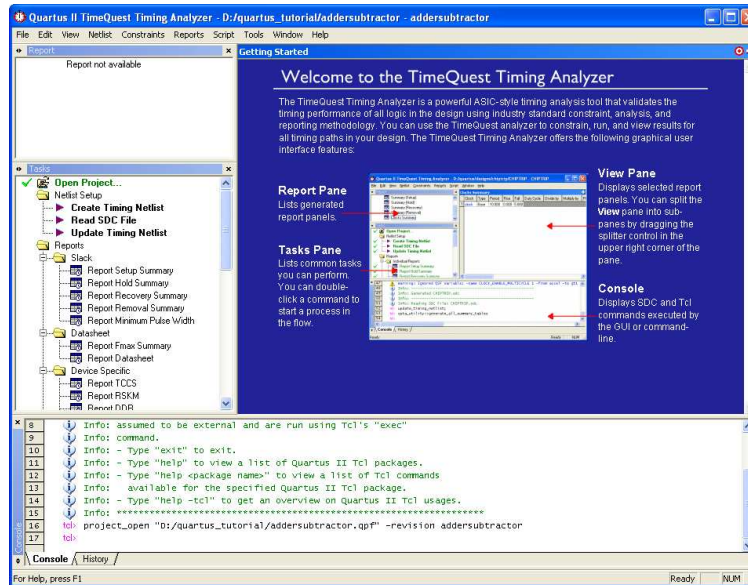


Figure 31: TimeQuest Timing Analyzer.

In the TimeQuest Analyzer window, double click **Create Timing Netlist** under the **Tasks** heading to generate the timing netlist from database created after compilation. Then select **Constraints > Create Clock** from the menu to reach the dialog box in Figure 32. Specify a name for the clock you want to constrain. This name can be used to refer to this clock when creating other timing constraints. For this example, we will name the clock "CLOCK". As we want the circuit to operate at a clock frequency of 450 MHz, set the period to 2.22 ns in the **Period** field. To specify the actual clock this constraint is applied to, click on the ... button beside the **Targets** field in the **Create Clock** window to get the pop-up window in Figure 33.

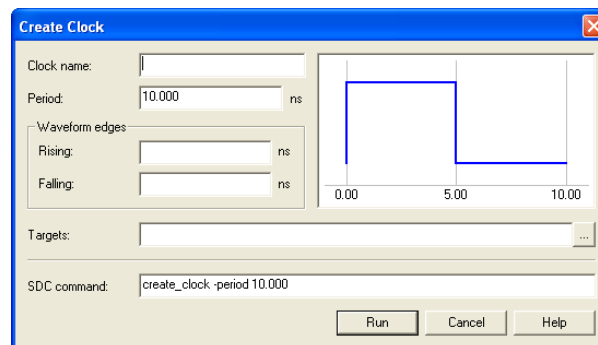


Figure 32: Create clock constraints.



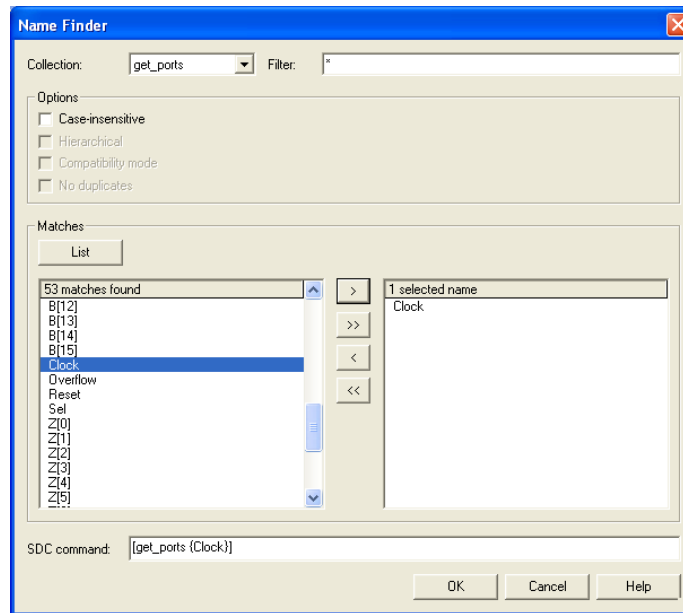


Figure 33: Choose target clock.

Click on the list button in the middle left of the window to get a list of all the ports. Select **Clock** from the list of matches on the left and click on the **>** button to reach the display in Figure 33. This tells the TimeQuest Timing Analyzer to constrain the **Clock** port to the period you have specified. Notice in a larger design with many ports, we can enter the name of the port in the **Filter** field located in the top right of the window to reduce the number of matches in our search. Click **OK** to return to the dialog box in Figure 34.

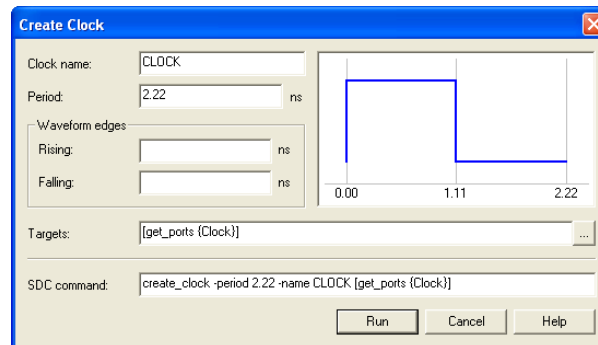


Figure 34: Complete clock constraint.

In the SDC command field at the bottom of the dialog box in Figure 34, we can see the actual command that would be written into the SDC file. It creates a simple clock named **CLOCK** with a 2.22 ns period and associates this constraint with the port **Clock**. Click **Run** to return to the window in Figure 31. Select **Constraints > write SDC File** and save the file as *addersubtractor.out.sdc* in the project directory *quartus\_tutorial*. Close the TimeQuest Timing Analyzer window to return to the Quartus® II main display.

- From the main Quartus® II window, click **Assignments > Timing Analysis Settings** and select "Use TimeQuest Timing Analyzer during compilation". Expand the Timing Analysis Settings category and then click on TimeQuest Timing Analyzer to reach the window in Figure 35. Here, we can add the SDC file we have created to the project. Click on the ... button beside the SDC filename field and select *addersubtractor.out.sdc*, which is the SDC file we created using TimeQuest Timing Analyzer. Click on **Open** to return to the window in Figure 35 and click **Add** to associate the SDC file with this project. Click **OK** to return to the main Quartus® II window.

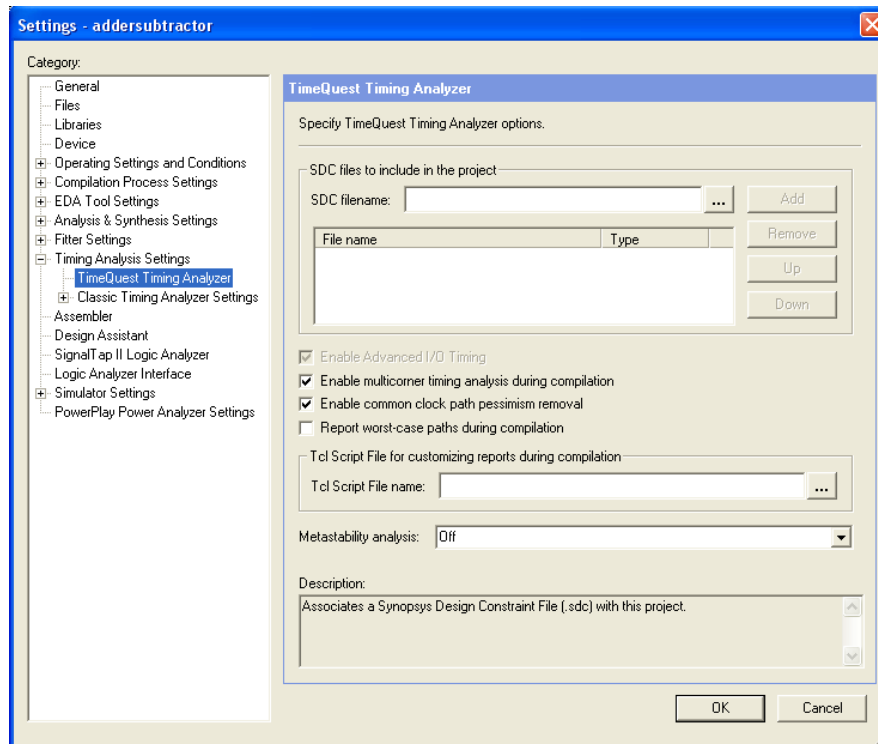


Figure 35: Adding SDC file to project.

The Quartus® II Compiler includes a Fitter executable that places the designed circuit into the available logic elements on the chip and generates the necessary wiring connections to realize the circuit. This is a complex process that can take a long time, particularly if the circuit is large and an ambitious value of  $f_{max}$  is specified. The time can be reduced if a lower value of  $f_{max}$  is acceptable. The user can indicate the level of the Fitter's effort.

3. From the main Quartus® II window, click **Assignments > Settings** and then select the category **Fitter Settings** which opens the window in Figure 36. Three different levels of effort can be given. Choose the Auto Fit option, which instructs the Fitter to stop as soon as it finds an adequate implementation. The Fast Fit option reduces the compilation time, but it may produce a lower  $f_{max}$ . The third option, Standard Fit, forces the Fitter to produce the best implementation it can find; at this effort level the Fitter will exceed the user's timing requirements as much as it can, which often results in longer compilation time. Click OK, and recompile the circuit.

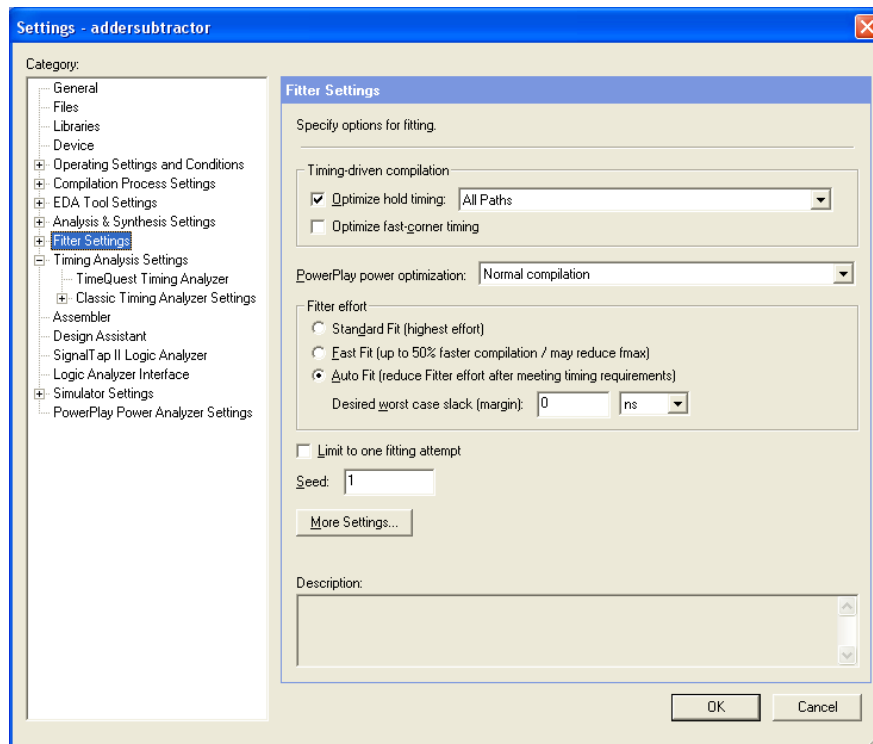


Figure 36: Fitter settings.

- The new timing results are shown in Figure 37. The new  $f_{max}$  is 502.77 MHz, which meets the specified requirement.

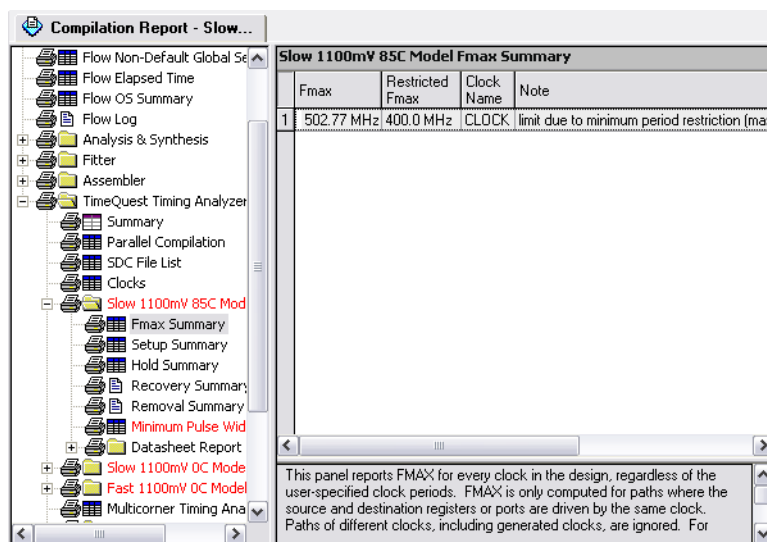


Figure 37: New timing results.

The  $f_{max}$  of a circuit is limited by the path with the longest delay. To view this critical path of this circuit, open the TimeQuest Timing Analyzer. Under the Tasks heading, select Reports > Custom Reports > Report Timing and double-click on it. This creates the dialog box in Figure 38. In the drop down boxes labeled From clock and To clock, select CLOCK, which is the name we specified for the clock in the circuit when we were creating the SDC file. Use a value of 10 for the Report number of paths field, located in

the Paths section of the window, to reach the display in Figure 38. Click on the Report Timing button to start the analysis. After the analysis is complete, a timing report similar to Figure 39 will be displayed.

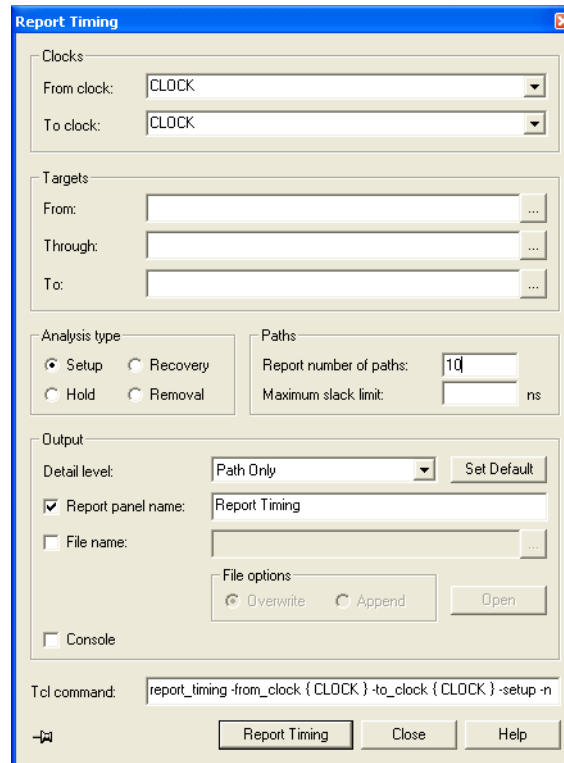


Figure 38: Create timing report.

The Report Timing window shows the ten paths with the longest delays in our circuit. We see that the critical path begins at *SelR* and ends at *Overflow*. The first column in the top part of the window shows the *slack* for each path, which is the amount of delay that could still be added to a given path without violating the specified timing constraint. The bottom part of the window shows the actual elements in the selected path and the incremental delay for each stage in the path.

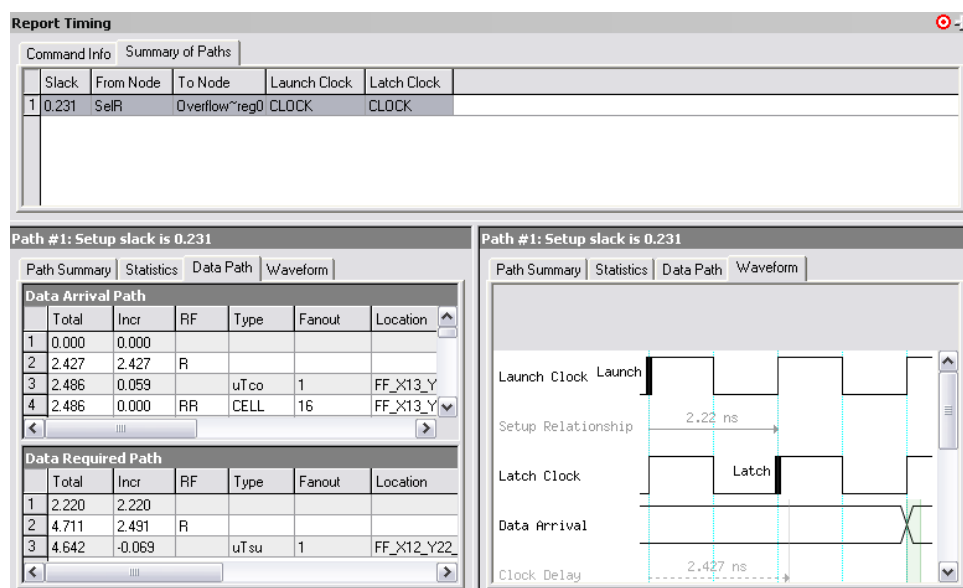


Figure 39: Critical path.

## 7 Quartus® II Windows

The Quartus® II display contains several utility windows which can be positioned in various places on the screen, changed in size, or closed. In Figure 19, which is reproduced in Figure 40, there are five windows.

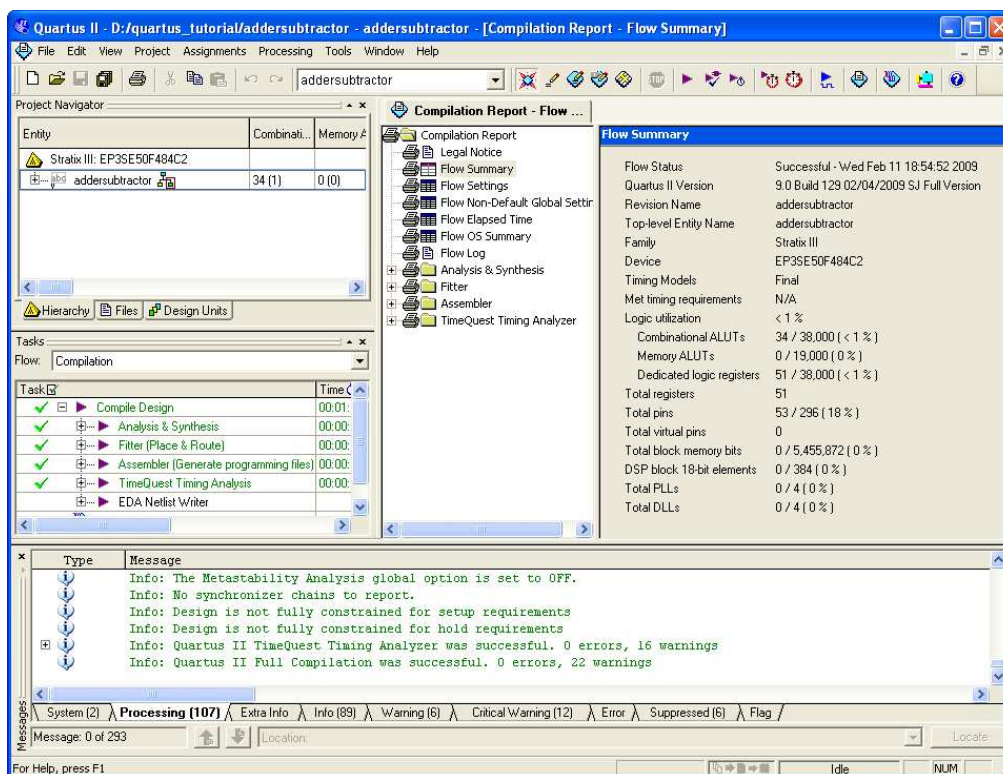


Figure 40: The main Quartus® II display.

The Project Navigator window is shown near the top left of the figure. Under the heading Entity, it depicts a tree-like structure of the designed circuit using the names of the modules in the Verilog code of Figure 12.

1. To see the usefulness of this window, open the previously compiled project *quartus\_tutorial\addersubtractor.qpf* to get a window similar to Figure 40.
2. Double-click on the name *adderk* in the hierarchy under the Entity heading. The Quartus® II software will open the file *addersubtractor.v* and highlight the Verilog module that specifies the adder subcircuit.
3. Right-click on the same name and choose **Locate > Locate in Chip Planner(Floor Plan & Chip Editor)** from the pop-up menu that appears. This causes the Quartus® II software to display the floorplan, as in Figure 23, and highlight the part that implements the adder subcircuit.

The Tasks window is located below the Project Navigator window in the Quartus® II main window. As you have already observed, this window displays the compilation progress. It can also be used to edit and start different stages of the compilation. Double-clicking on a compilation stage from the Tasks window causes that stage of the compilation to be re-run.

At the bottom of the Quartus® II main window is the Message window, which displays user messages produced during the compilation process.

The large area in the middle-right of the Quartus® II window is used for various purposes. As we have seen, it is used by Report Viewers and the Text Editor.

A utility window can be moved by dragging its title bar, resized by dragging the window border, or closed by clicking on the X in the top-right corner of that window. A particular utility window can be opened by selecting it from the **View > Utility Windows** menu.

Copyright ©2009 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.