Xilinx® ISE WebPACK™ Schematic Capture Tutorial

Revision: February 27, 2010

215 E Main Suite D | Pullman, WA 99163
(509) 334 6306 Voice and Fax

Overview

This tutorial provides instruction for using the Xilinx ISE WebPACK toolset for basic development on Digilent system boards. This tutorial covers the following steps:

- Creating a Xilinx ISE project
- Using schematic capture to create logic circuits and symbol elements
- Creating a User Constraints File (UCF)
- Synthesizing, implementing, and generating a Programming file

More detailed tutorials for the Xilinx ISE tools can be found at http://www.xilinx.com/support/techsup/tutorials/.

Getting Started

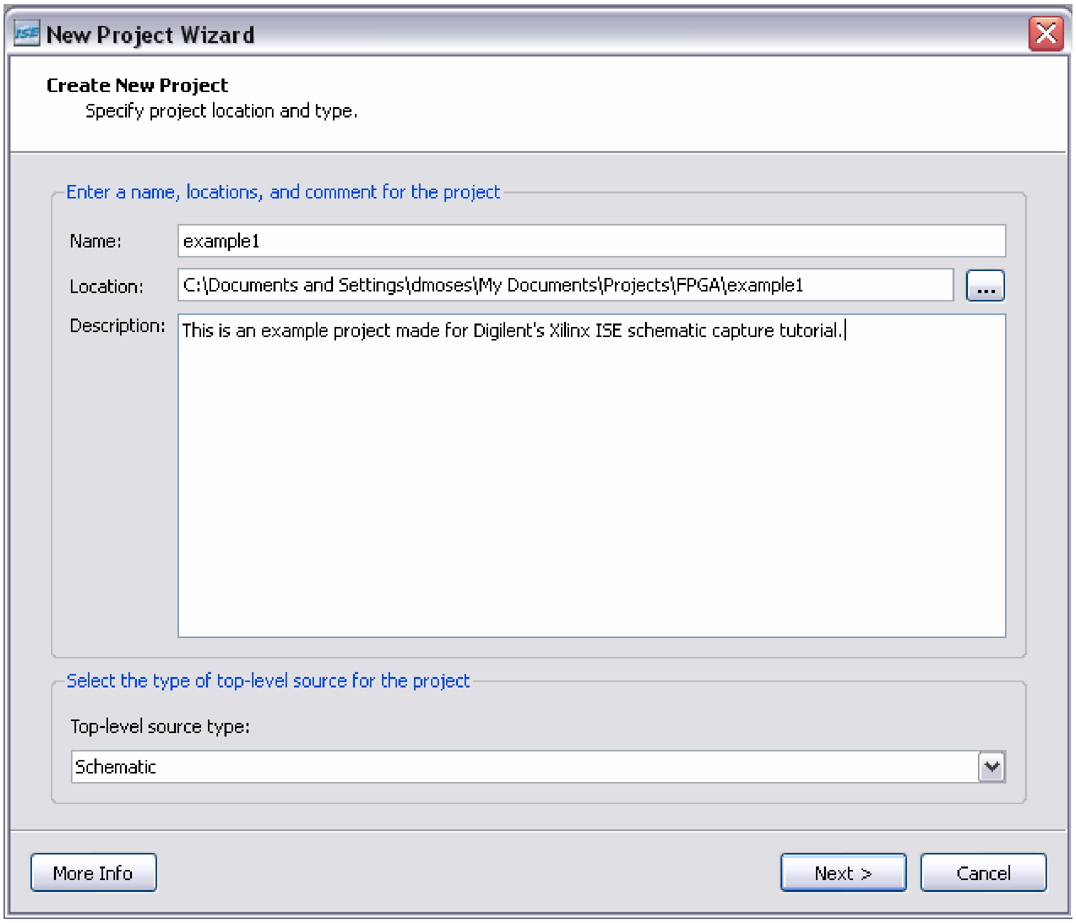
First, install Xilinx ISE WebPACK on your PC or laptop. This tutorial is based on version 11.1. It is available as a free download from www.xilinx.com.

This tutorial uses settings for the Nexys2 500k board, which can be purchased from www.digilentinc.com. The settings for other Digilent system boards can be found there as well.

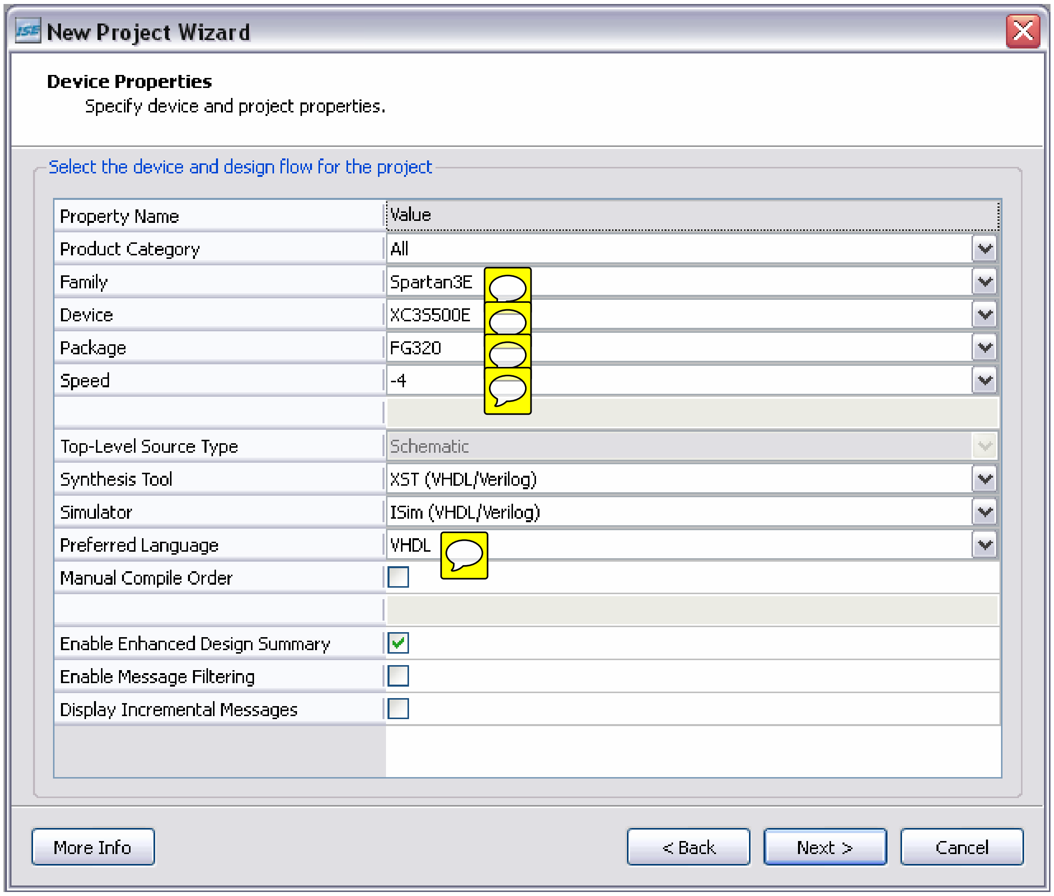
Starting a New Project

To create a new project, open Project Navigator either from the Desktop shortcut icon or by selecting Start > Programs > Xilinx ISE Design Suite 11 > ISE > Project Navigator. In Project Navigator, select the New Project option from the Getting Started menu (or by selecting Select File > New Project).

This brings up a Dialog box where you can enter the desired project name and project location. You should choose a meaningful name for easy reference. In this tutorial, we call this project "example1" and save it in a local directory. You can place comments for your project in the Description text box. We use Schematic for our top-level source type in this tutorial.


New Project Wizard
Create New Project
Specify project location and type.
Enter a name, locations, and comment for the project
Name:
example1
Location:
C:\Documents and Settings\dmoses\My Documents\Projects\FPGA\example1
Description:
This is an example project made for Digilent's Xilinx ISE schematic capture tutorial.
Select the type of top-level source for the project
Top-level source type:
Schematic
More Info
Next >
Cancel

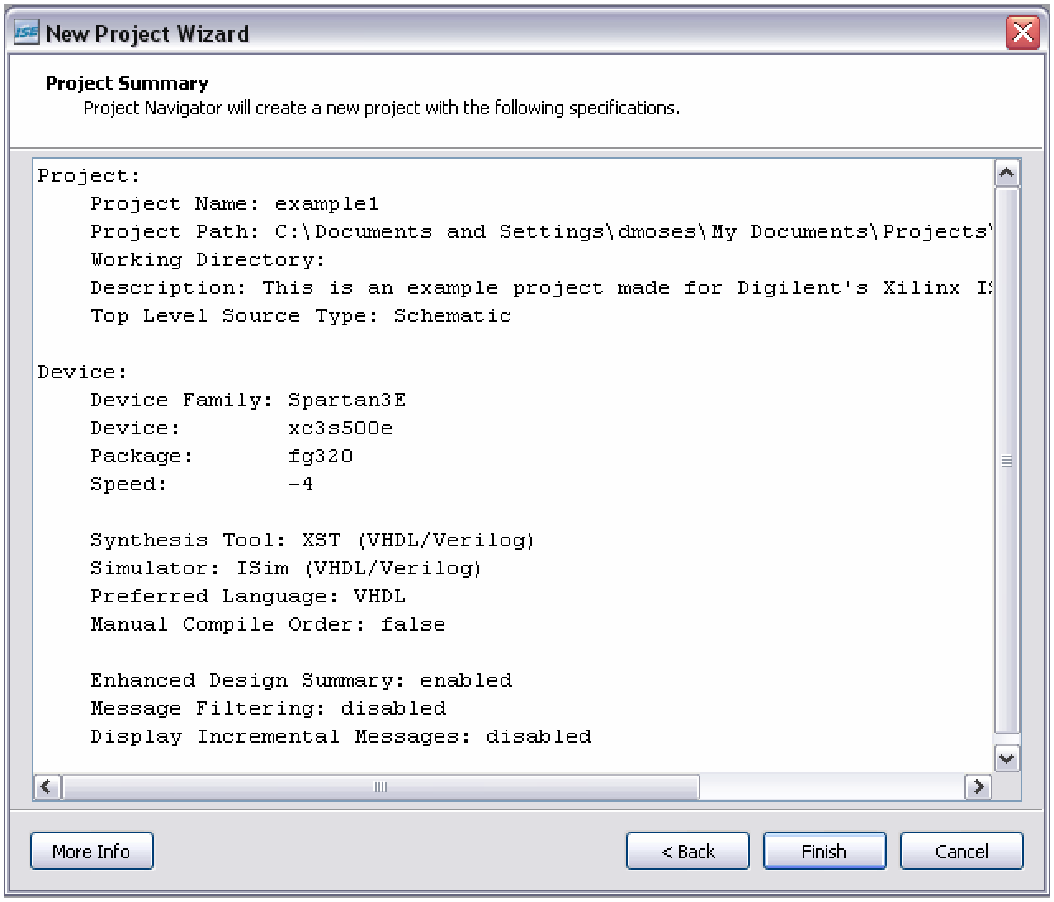
The next step is to select the proper Family, Device, and Package for your project. This depends on the chip you are targeting for the project. The appropriate settings for a project suited for the Nexys2 500k board are as follows:



Once the appropriate settings have been entered, click Next.

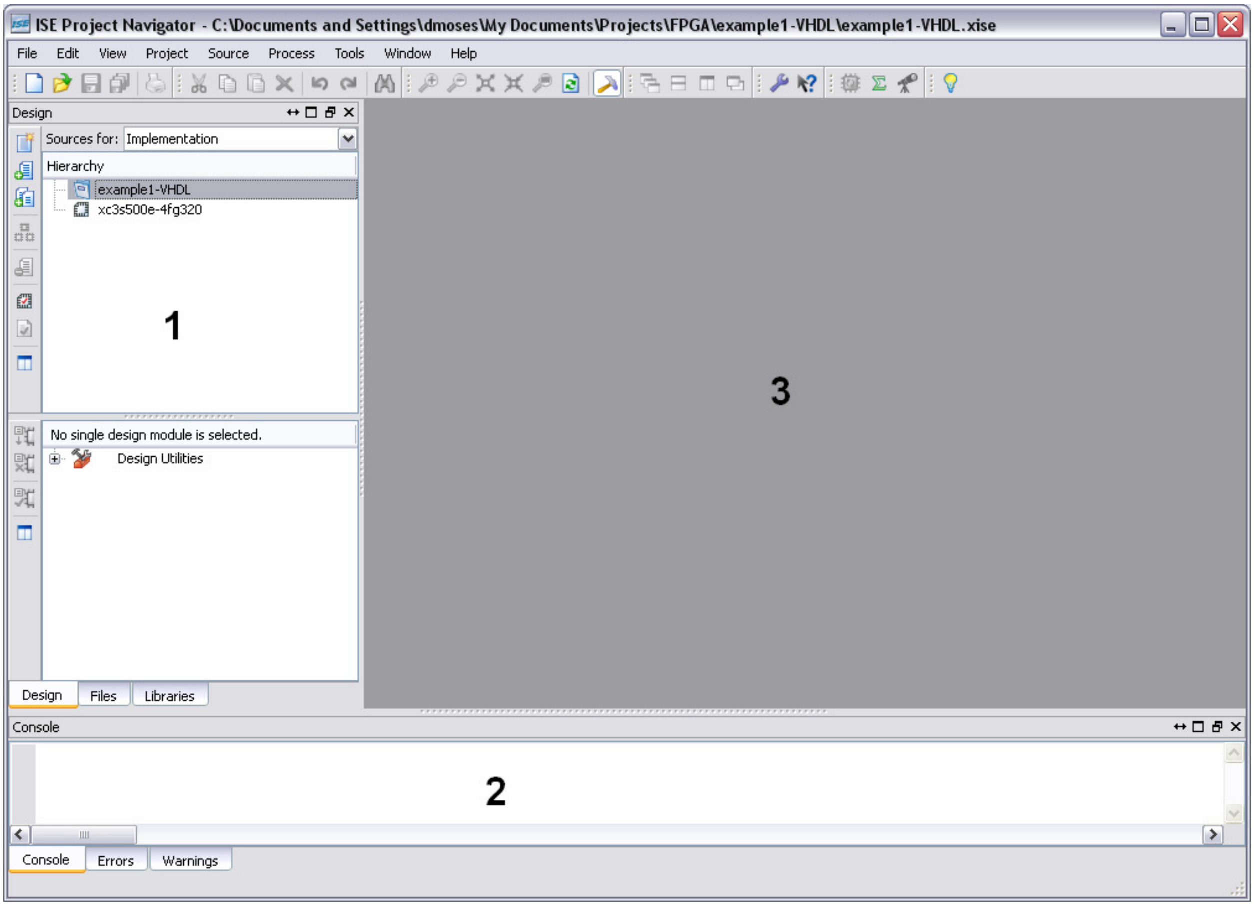
The next two dialog boxes give you the option of adding new or existing source files to your project. Since we will fulfill these steps later, click Next without adding any source files.

Before the new project is created, the New Project Wizard gives you a project summary consisting of the selected specifications you have chosen for the project. Make sure all settings are correct before clicking Finish to end the New Project Wizard. Any modifications to these settings can be made by clicking the Back button.



Project Navigator Overview

Once the new project has been created, ISE opens the project in Project Navigator. Click the Design tab to show the Design panel and click the Console tab to show the Consol panel.



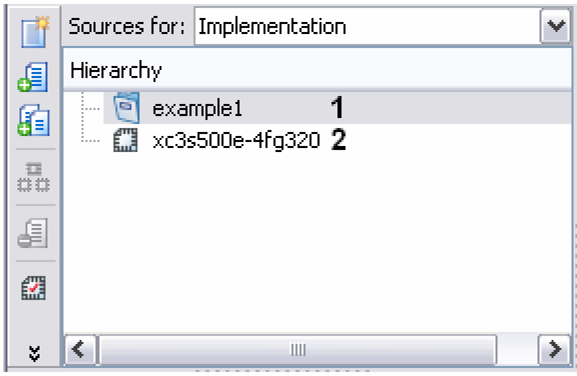
The Design panel (1) contains two windows: a Sources window that displays all source files associated with the current design and a Process window that displays all available processes that can be run on a selected source file.

The Console panel (2) displays status messages including error and warning messages.

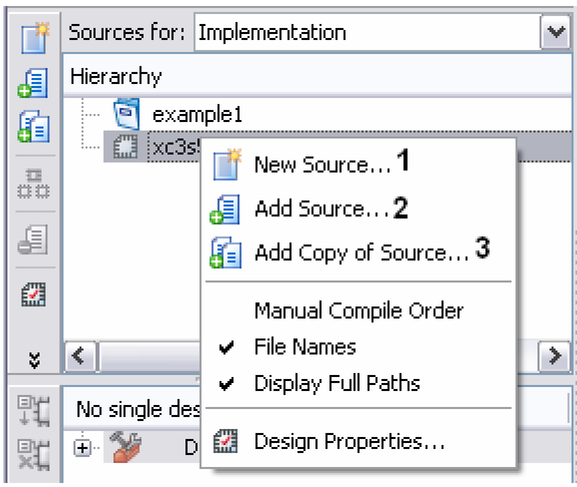
The HDL editor window (3) displays source code or the schematic from files selected in the Design panel.

Adding New Source Files

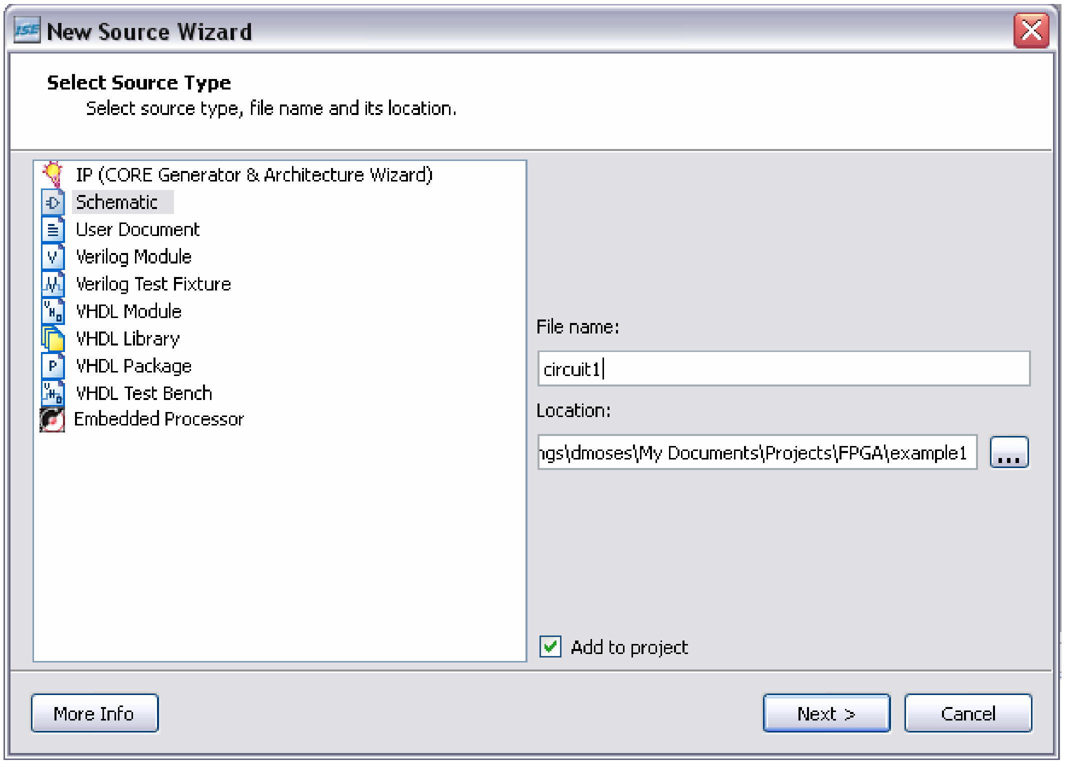
Once the new project is created, two sources are listed under sources in the Design panel: the Project file name and the Device targeted for design.



You can add a new or existing source file to the project. To do this, right-click the target device and select one of the three options for adding source files.

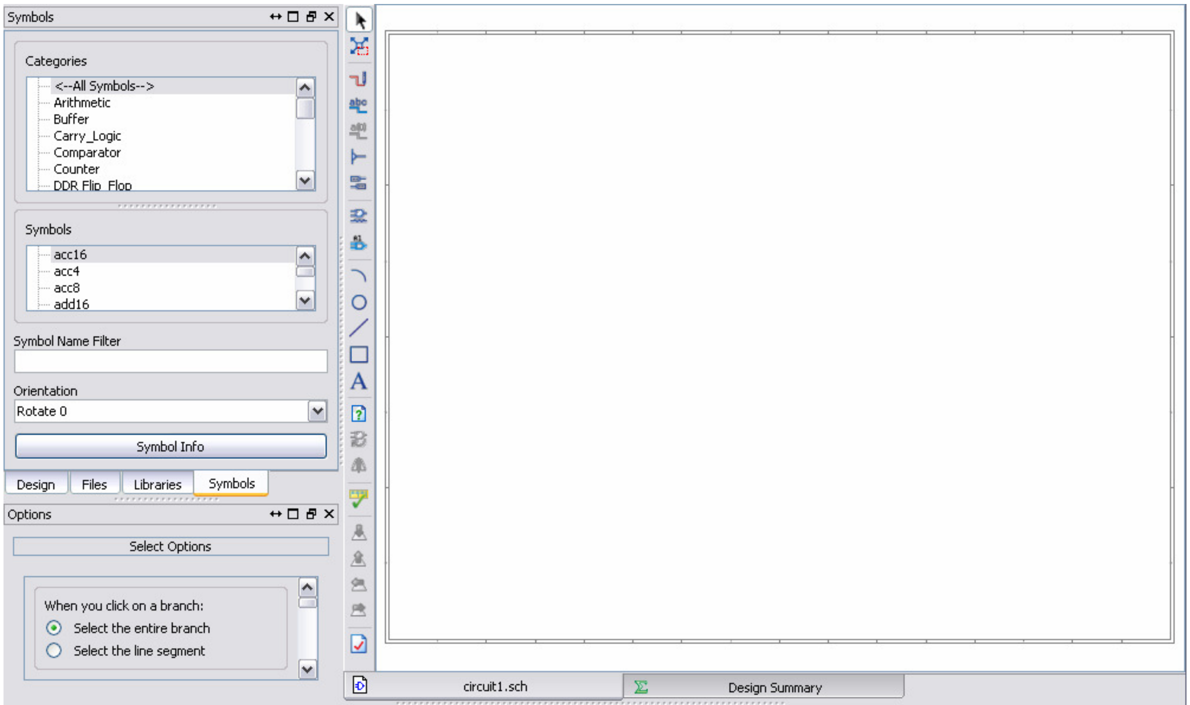


In this tutorial, we create a new source file, so select New Source from the list. This starts the New Source Wizard, which prompts you for the Source type and file name. Select Schematic and give it a meaningful name (we name it circuit1).



Schematic Editor Window

Once you have created the new schematic file you can see it in the Sources panel. Double-click it to open the file in the schematic capture window as follows:



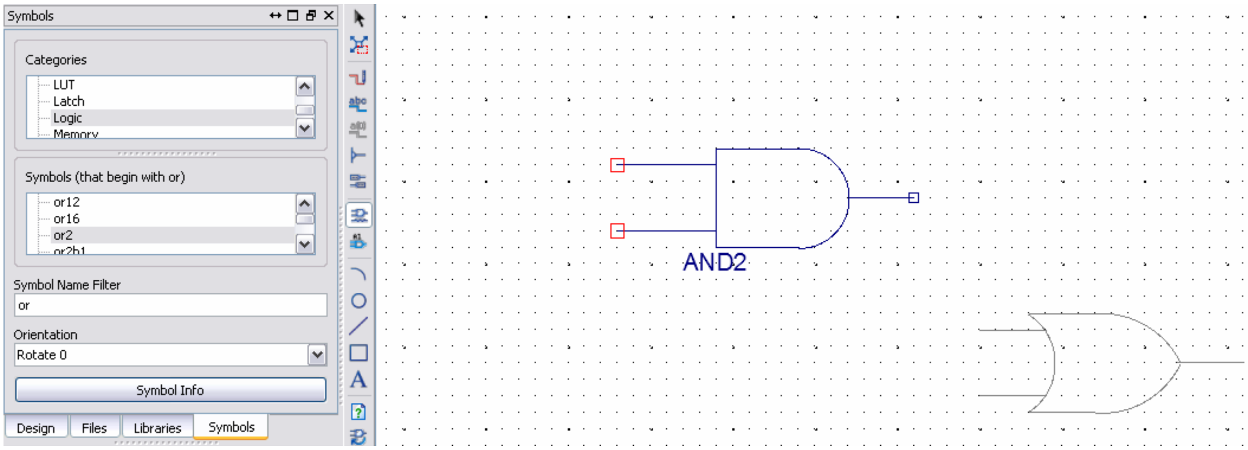
You can add symbols and shapes representing logic gates or logic circuits and then add lines representing wires to connect those shapes.

On the screen, there are two list boxes labeled Symbols and Categories.

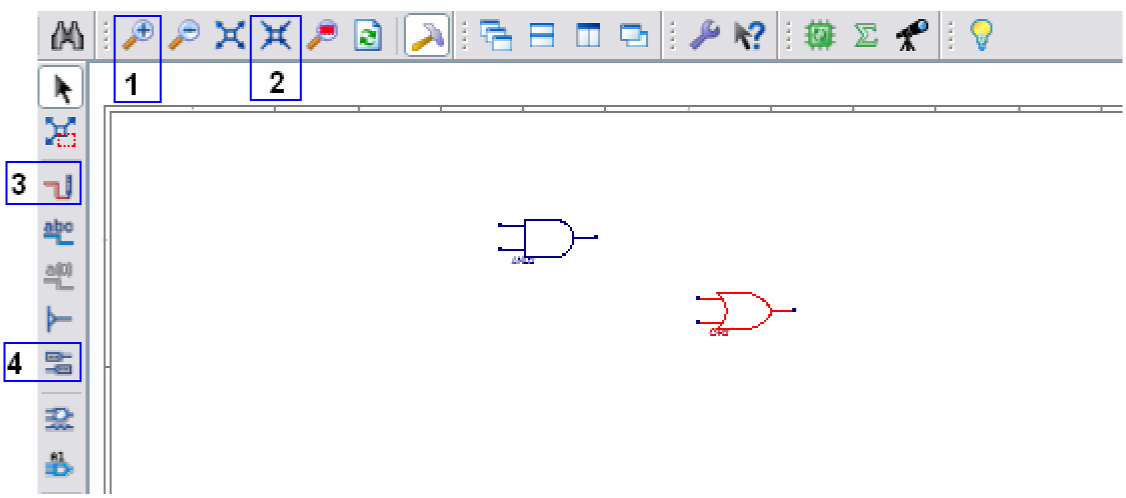
The Symbols list shows all of the symbols in selected category in the Categories list. For example, the “<--All symbols-->” category displays all symbols in the current library in the Symbols list.

In this tutorial, we use a simple combinational logic example, and then show how it can be used as a macro symbol in another schematic module. We start with the basic logic equation: Y<= (A·B) + C.

To add a simple logic gate to the circuit, left-click to select a symbol from the Symbols list, drag the cursor to the pallet where you want the symbol to be placed, and left-click again to drop the gate in the schematic. You can also find a symbol by typing a name into the symbol name filter.



The following figure contains descriptions for the tools necessary for creating a basic digital circuit using schematic capture:



1. The Magnifying Glass icon indiscriminately zooms to the center of the schematic.
2. The Zoom box icon is used to draw a box using the mouse to magnify a specific area of the schematic.
3. The Wire-Add tool icon places cursor in wire-add mode.
4. The Add I/O marker tool icon places cursor into add I/O marker mode.

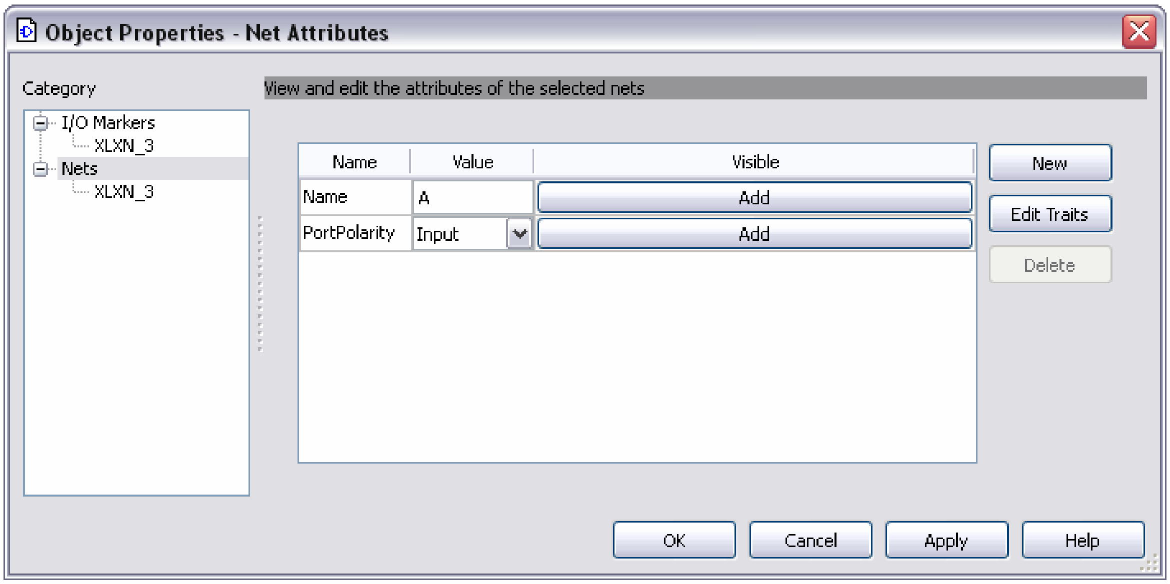
Now we connect the gates with wires. To do this, left-click on the wire-add tool button to change to wire-add mode. Drag the cursor to a component pin where four red boxes appear (they indicate that a left-click will add one end of the wire to the pin). Using this method you can add wires between pins of gates.

To move a gate, you must go back to select mode by either selecting the cursor in the tool bar or pressing the Esc button on the keyboard.

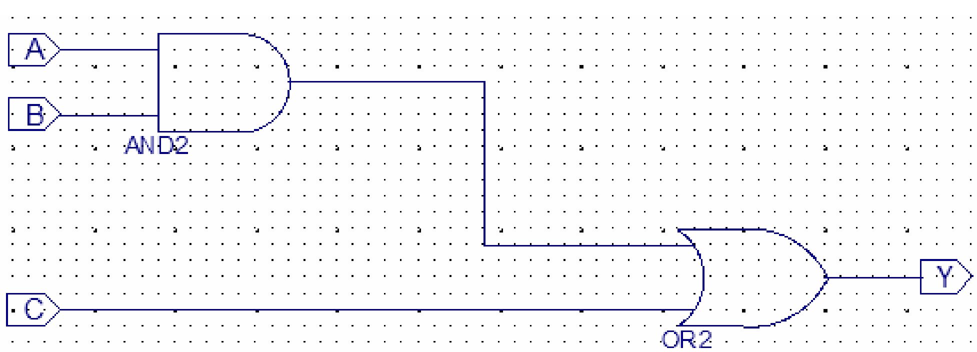
To terminate one end of a wire without connecting it to a gate, double-click where you want the wire to end in space. This is used for general circuit inputs and outputs.

Adding top-level I/O markers to your circuit tells the synthesizer and simulator tools which ports to regard as overall inputs and outputs. To add these to your circuit, left-click on the Add I/O marker tool icon to place the cursor into add I/O marker mode. Left-click on the end of a wire to add an I/O marker and then repeat until a marker is placed on all inputs and outputs.

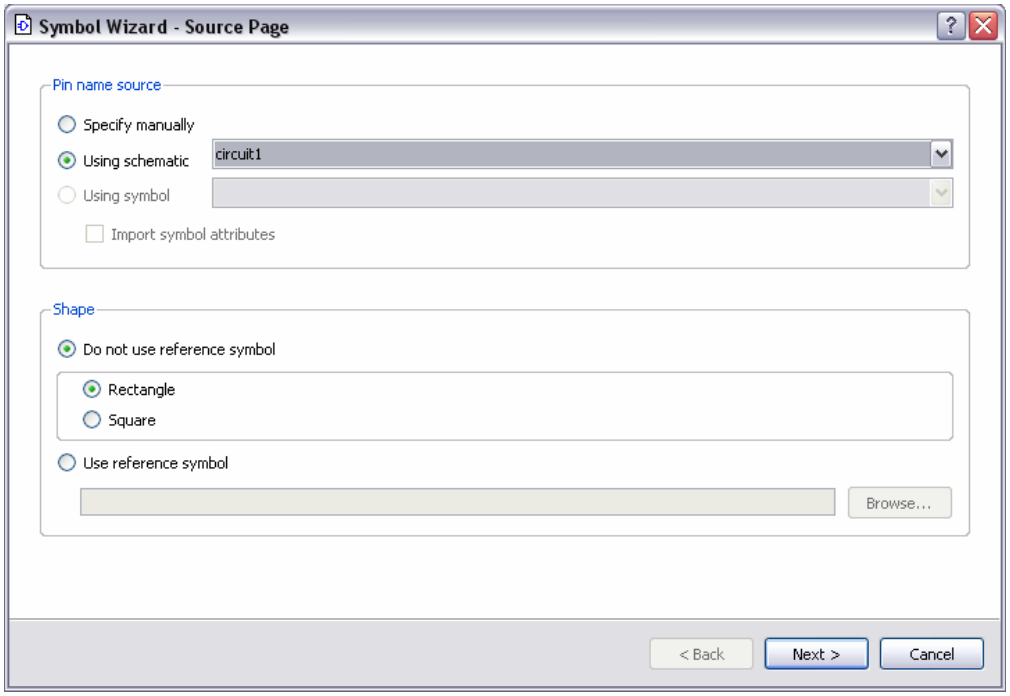
Go back to regular cursor mode and double left-click on an I/O marker. When the I/O marker's object properties dialog box appears, select the Nets category and enter a meaningful value for the Name field of the I/O marker. The finished dialog box should look similar to the following:



Repeat this process for all I/O. The finished circuit should look similar to the following:



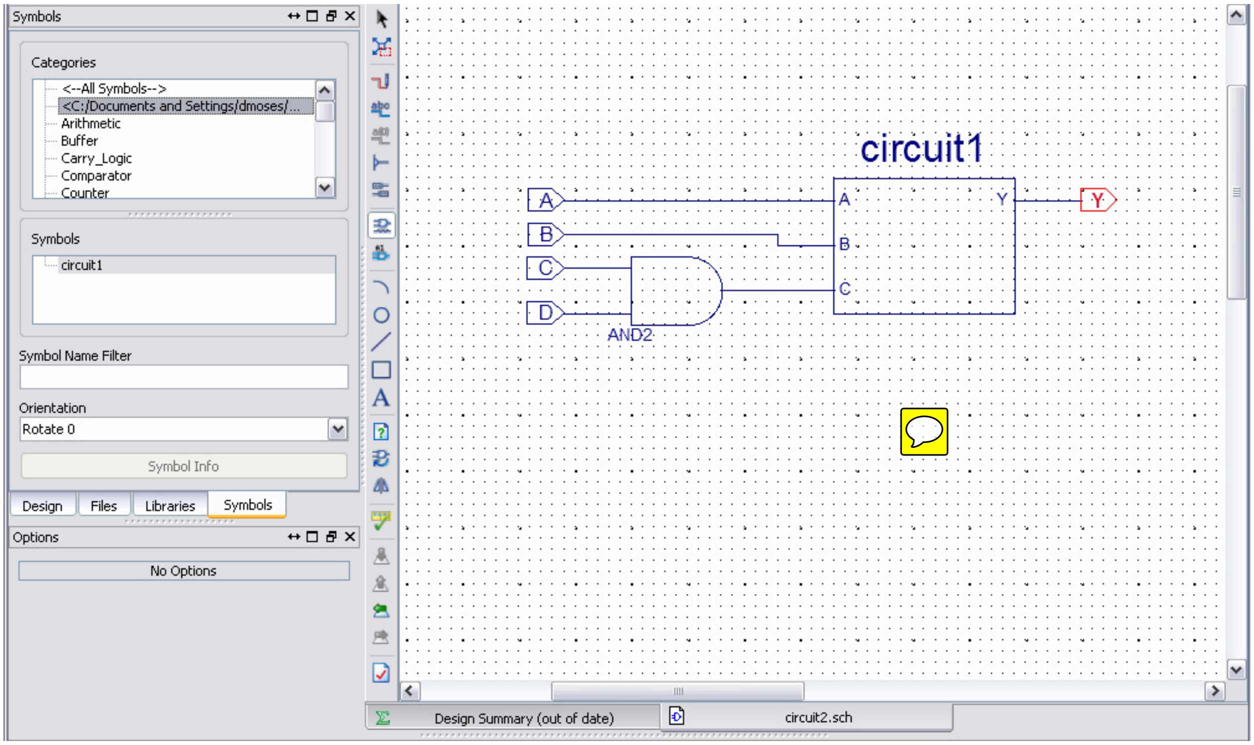
Now we create a macro symbol for future use of this completed circuit. To create a macro circuit, select Tools > Symbol Wizard from the main menu toolbar. The Symbol Wizard dialog box appears. Under "Pin name source", select "Using schematic" which defaults to the circuit you just created, circuit1.



The next two dialog boxes present general pin placement and symbol size options that you can modify depending on how you want the schematic symbol to appear. The final dialog box gives you a preview of the symbol, given the settings you've selected. Click Finish to finalize the symbol or Back to modify the symbol settings.

Save changes to the current schematic editor session before closing the schematic file and return to the Project Navigator main window. Right-click on the device to add a new schematic source file; we call ours circuit2. In the new schematic editor session you can now find the newly created symbol for circuit1 in the symbols window and add it to your overall design.

The following figure shows our completed example of using a macro symbol to build the overall circuit Y<= (A · B) + (C · D):



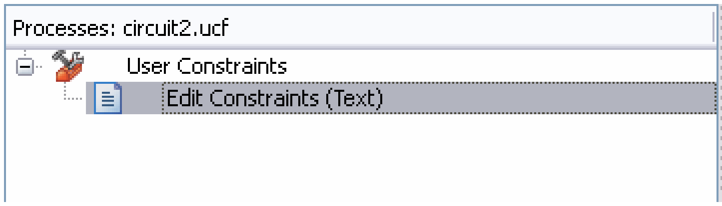
Using this method, you can create complex circuits by adding any components from the Xilinx library or by building your own components and adding them to an even larger design.

UCF File Creation

The Xilinx tools use a User Constraints File (.ucf file) to define user constraints like physical pin to circuit net mappings. This is sometimes referred to as an Implementation Constraints File. The .ucf file can be modified inside ISE using a text editor.

To add a .ucf file to your design, go to the Sources window and right-click the source file that requires user constraints. Select the Add New Source option in the drop-down menu. The New Source Wizard prompts you for the Source type and file name. Select Implementation Constraints File and give it a meaningful name (we name it circuit2).

To edit the .ucf file, select it in the Sources window, expand the User Constraints option in the Processes window below, and double-click the Edit Constraints (Text) option. A blank text editor appears.



To associate a physical pin with a given net name, type: NET “netname” LOC =”XXX”; on a line in the .ucf file. In the statement, “netname” (quotes included) is the name of the net to attach to pin number XXX (quotes included).

For our example project, the four inputs are assigned to switches 0 through 3 and the output is assigned to LED0 on the Nexys2 board. The finished .ucf file is as follows:

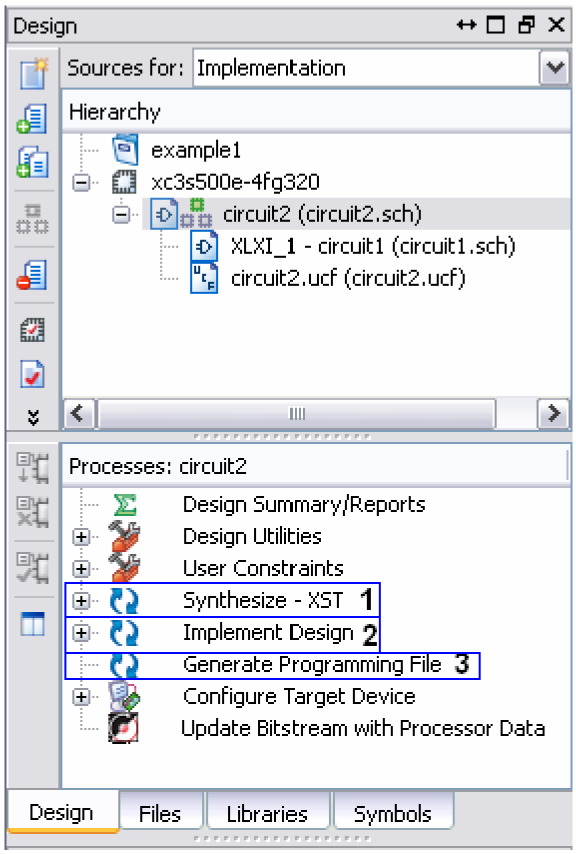
```
NET "AT" LOC = "G18";
NET "BT" LOC = "H18";
NET "CT" LOC = "K18";
NET "DT" LOC = "K17";

NET "YT" LOC = "J14";
```

Programming File Generation

Now we are ready to create a programming file (.bit) for the Nexys2 FPGA.

Go to the Sources window and select the top-level module (indicated by the three blocks shown with the source name.)



Now go to the Processes window where there are three particular processes in a row:

1. Synthesize – XST
2. Implement Design
3. Generate Programming file

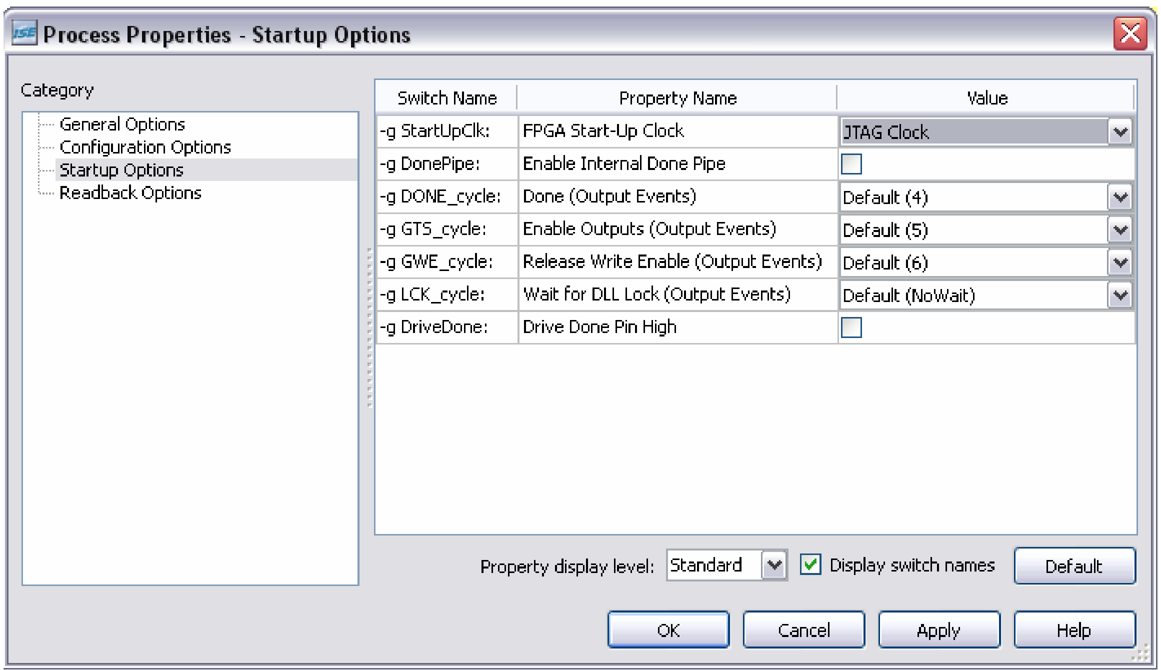
Run the synthesis process by either double-clicking on Synthesize or left-clicking and selecting the run option. This process analyzes the circuit you have created, checking for valid connections, syntax, and structure, to verify that the circuit is valid and synthesizable.

If the Synthesize process does not return any errors, you can move on and run the Implement Design process. This process uses various algorithms to map out the digital circuit and then creates place and route information so that it can be placed on the physical FPGA.

Startup Clock Options

If the Implement Design process does not return any errors, you can run the Generate Programming File process. Before we do this, right-click on the Generate Programming File process and select Process Properties. In the category pane of the Process Properties window, select Startup Options.

The first item in the right panel is the FPGA Start-Up Clock property. This option allows a configuration file to either configure a board straight from the PC, or load a configuration from platform flash memory on the board. To configure the board from the PC, the start-up clock value should be JTAG Clock. To configure the board from platform flash, the start-up clock value should be CCLK.



After selecting the appropriate start-up clock value, click OK and run the Generate Programming File process. After this process completes, a configuration .bit file should appear in the directory where your project is located.

Board Configuration

The configuration .bit file that has been generated can now be used by Digilent's Adept software to configure a Digilent system board. See the *Adept Software Basic Tutorial* for more information.