

Xilinx ISE

<Release Version: 10.1i>

Tutorial

Department of Electrical and Computer Engineering
State University of New York – New Paltz

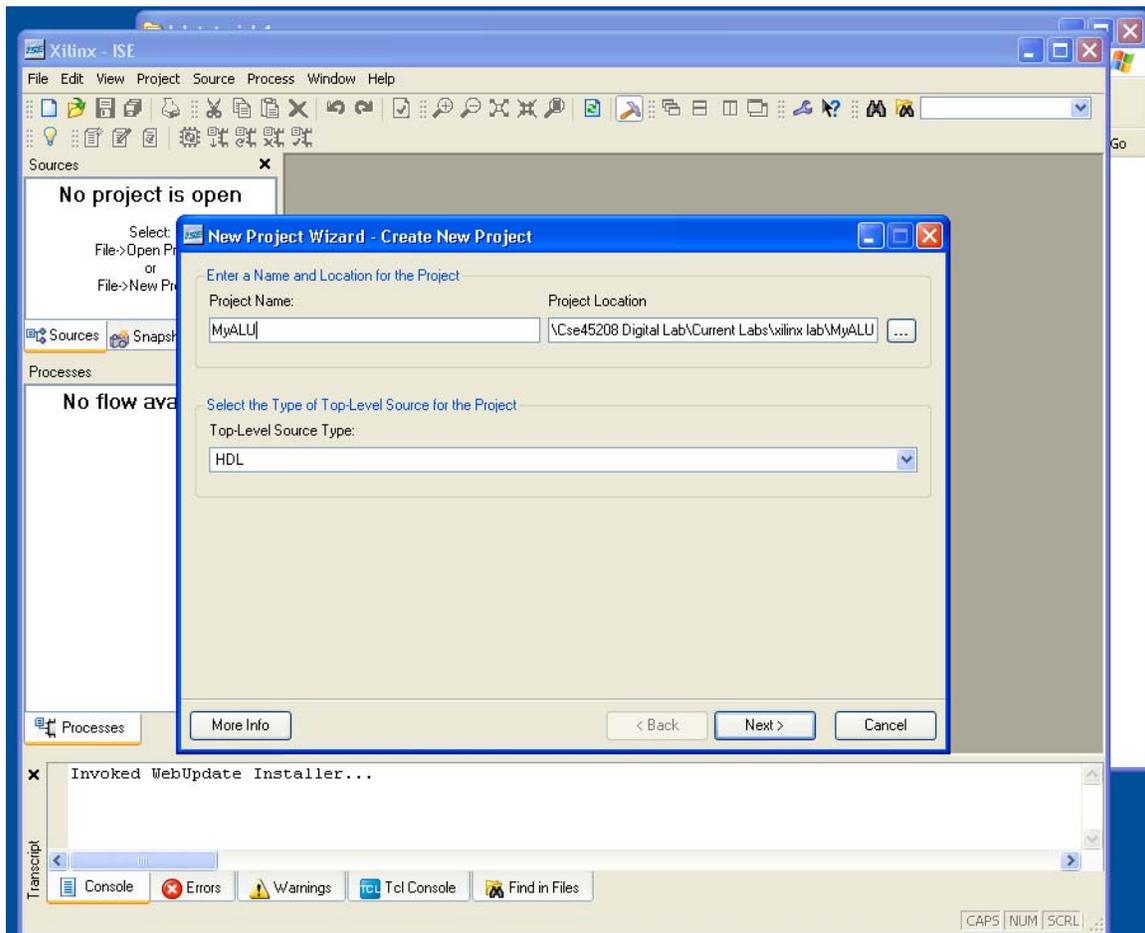
Starting the ISE Software

Start ISE from the Start menu by selecting **Start >Programs >Engineering Software > Xilinx ISE 8.2 >Project Navigator**.

Creating a New Project

A project in ISE is a collection of all files necessary to create and download a design to the selected device. To create a new project:

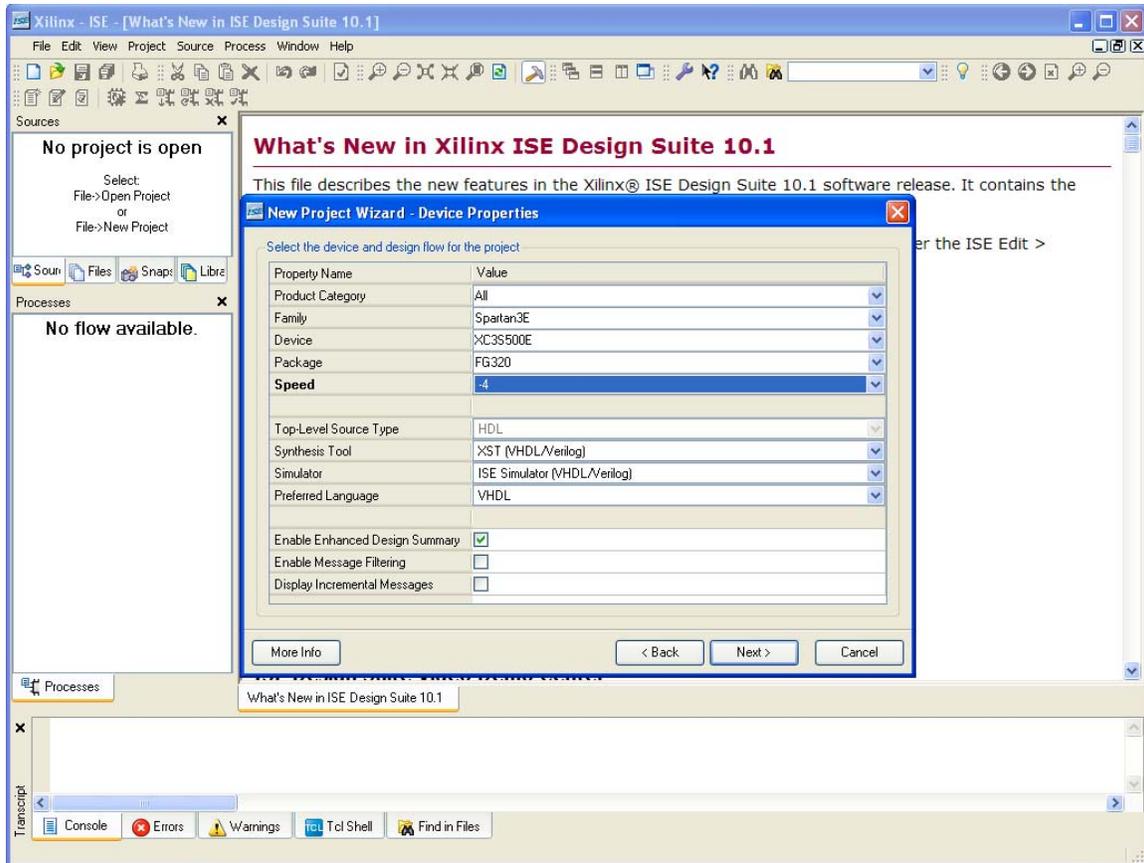
1. Select **File > New Project**.



2. In the New Project Wizard dialog box, enter the project name 'MyALU' in the Project name field. You can browse to the desired directory using the browse button next to the Project Location field.
3. Use the pull-down arrow to select HDL from the Top-Level Module Type field.

Click **Next**.

4. In the New Project Wizard Device and Design Flow dialog box, use the pull-down arrow to select the Value for each Property Name. Click in the field to access the pull down list.

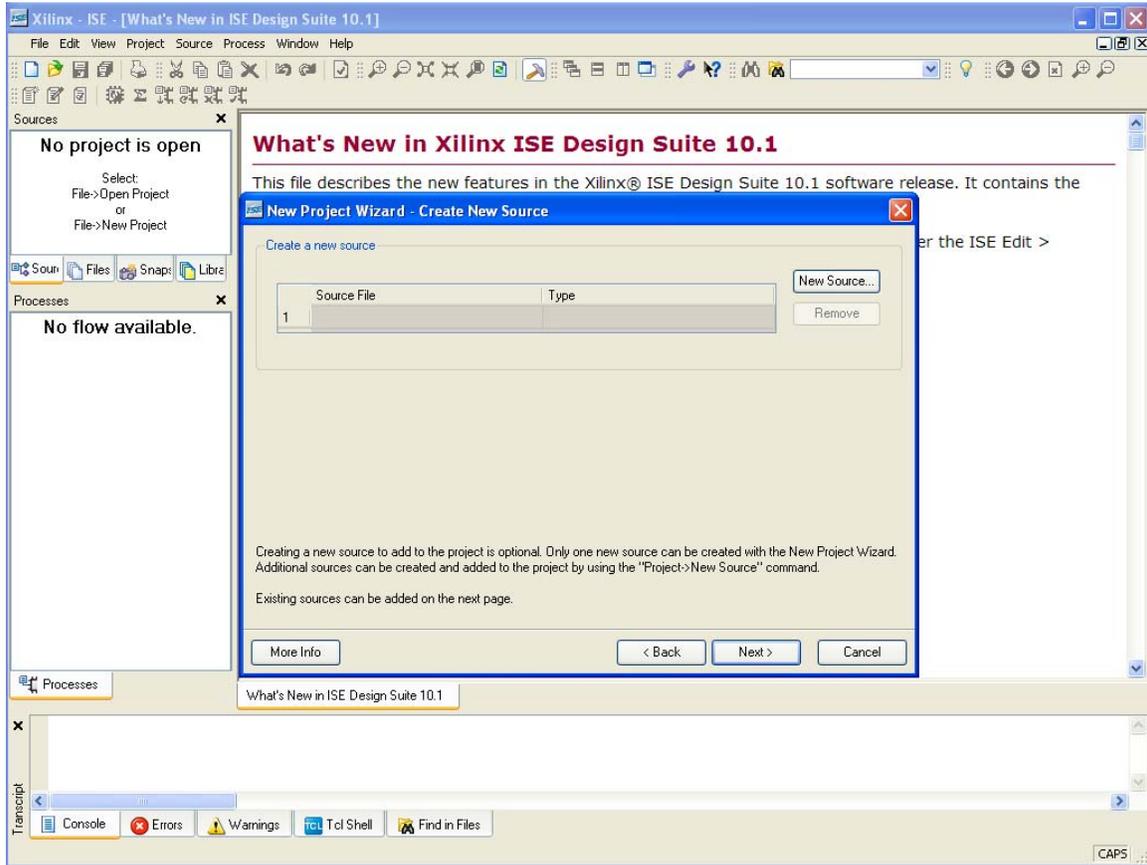


Change the values as follows:

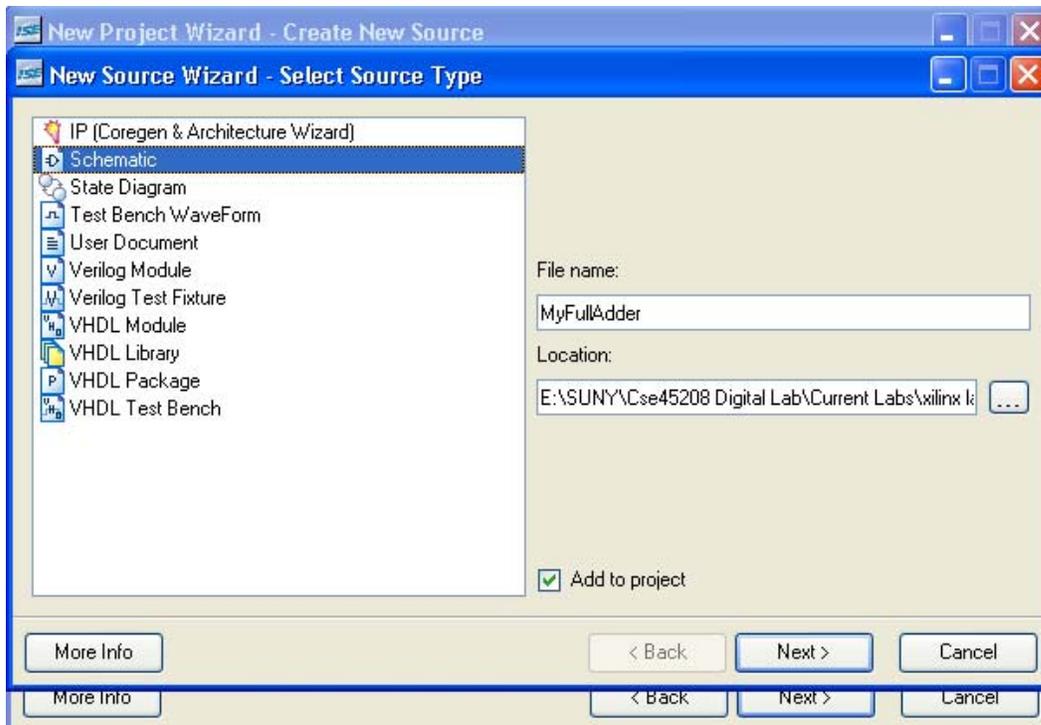
- Device Family: Spartan3E
- Device: xc3s500E
- Package: FG320
- Speed Grade: -4
- Synthesis Tool: XST (VHDL/Verilog)
- Simulator: ISE Simulator (VHDL/Verilog)
- Preferred Language: VHDL

Click **Next**

5. This will open a New Project dialog box. Click on 'New Source' to create a new source to be added to the project.

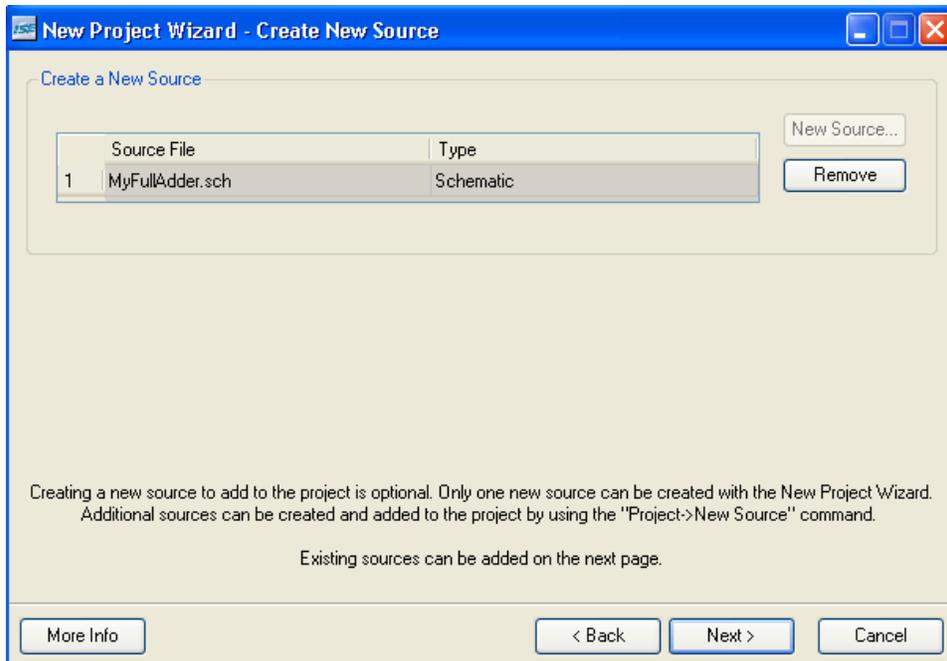


6. In the New Source dialog box select **Schematic** from the list. Name the file as **MyFullAdder** in the File Name field. Verify that the Add to project check box is selected.



Click **Next!** Then **Finish!**

7. The New Project dialog box will reappear with confirmation of the Source file name and type.

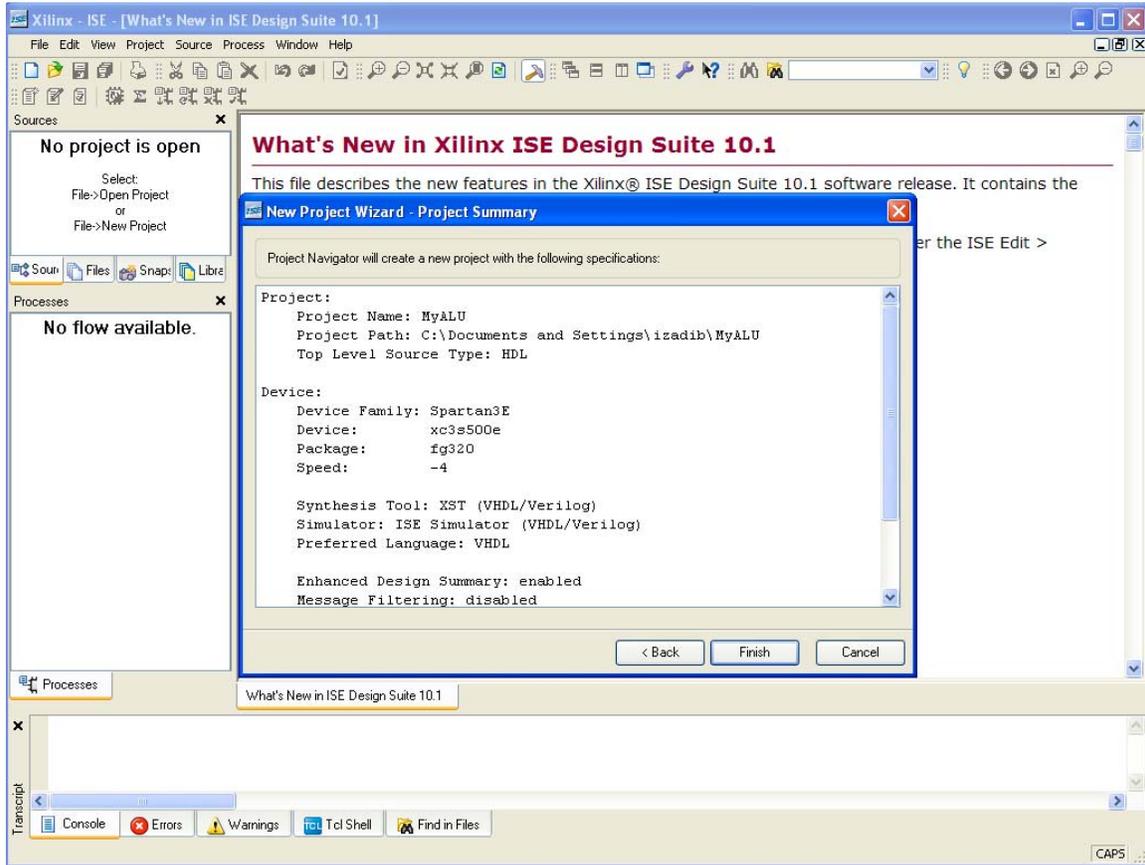


Click **Next.**

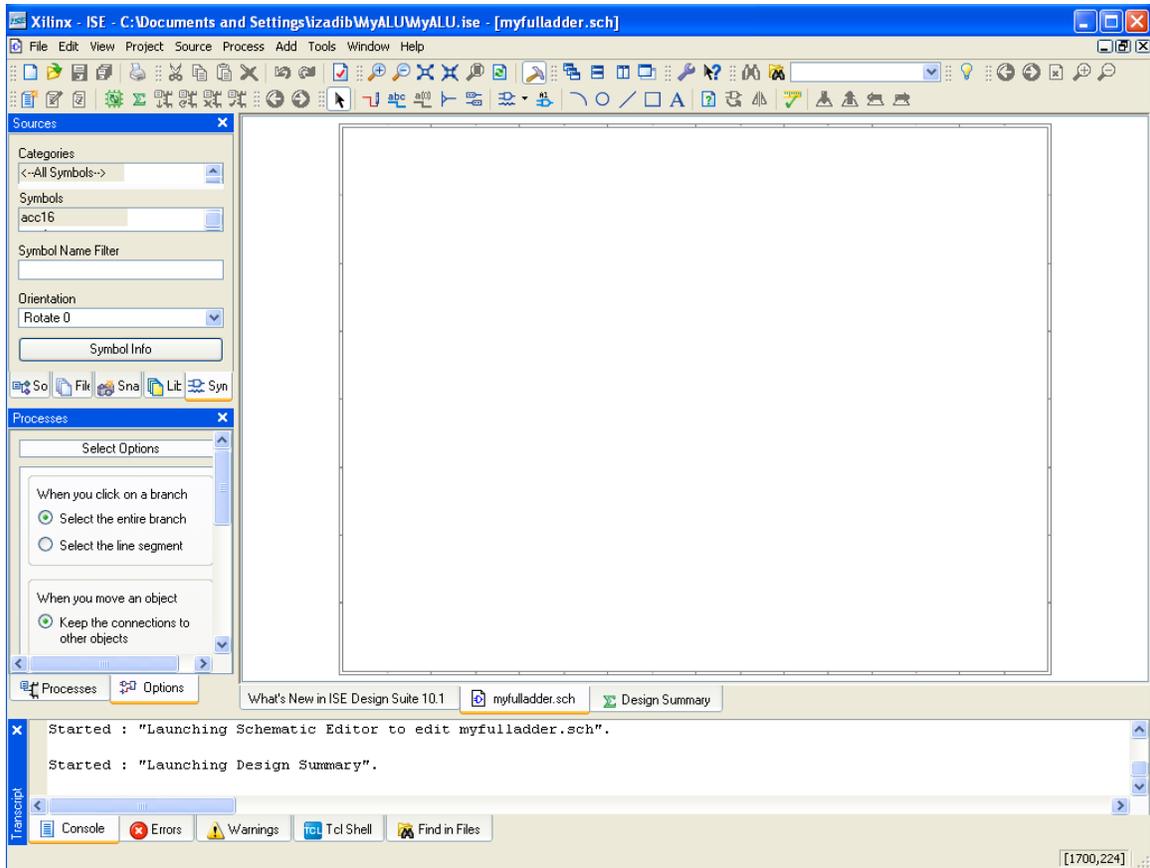
Click **Next.**

Finish.

The project Navigator creates a project using the information that you specified and displays a window as follow.



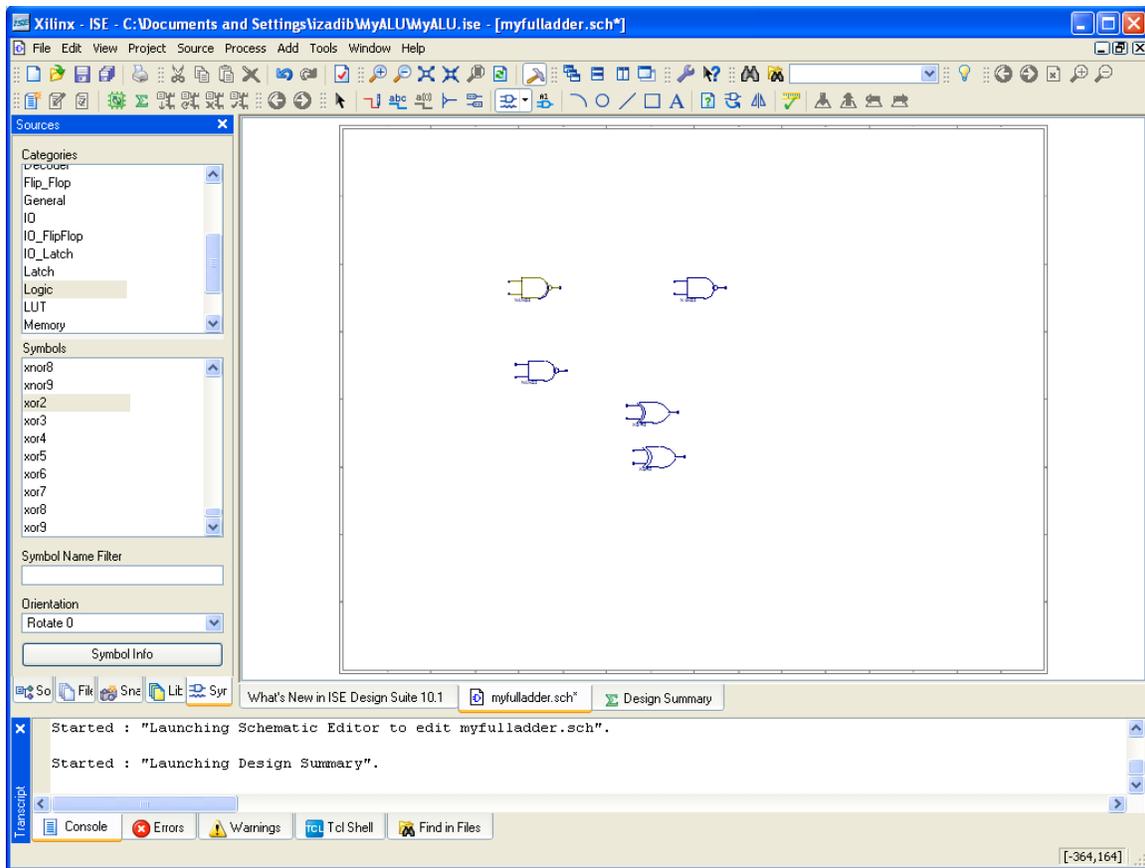
This will launch Engineering Capture System (ECS). You may have to choose the tab labeled MyFullAdder and close other windows in the left side (don't close the Sources window). A blank sheet opens in an ECS schematic window. In ECS, you can create a schematic diagram from scratch.



Creating a Schematic

On the left side of the screen you will see two small windows, Categories & Symbols. This is where we will find all the components we will need to assemble our full adder.

1. Select Logic in the Categories window. What this does is to only display logic gates in the Symbols window. From the Symbols window select a NAND2 and drop three of these gates onto your workspace. Also grab two XOR2 gates.

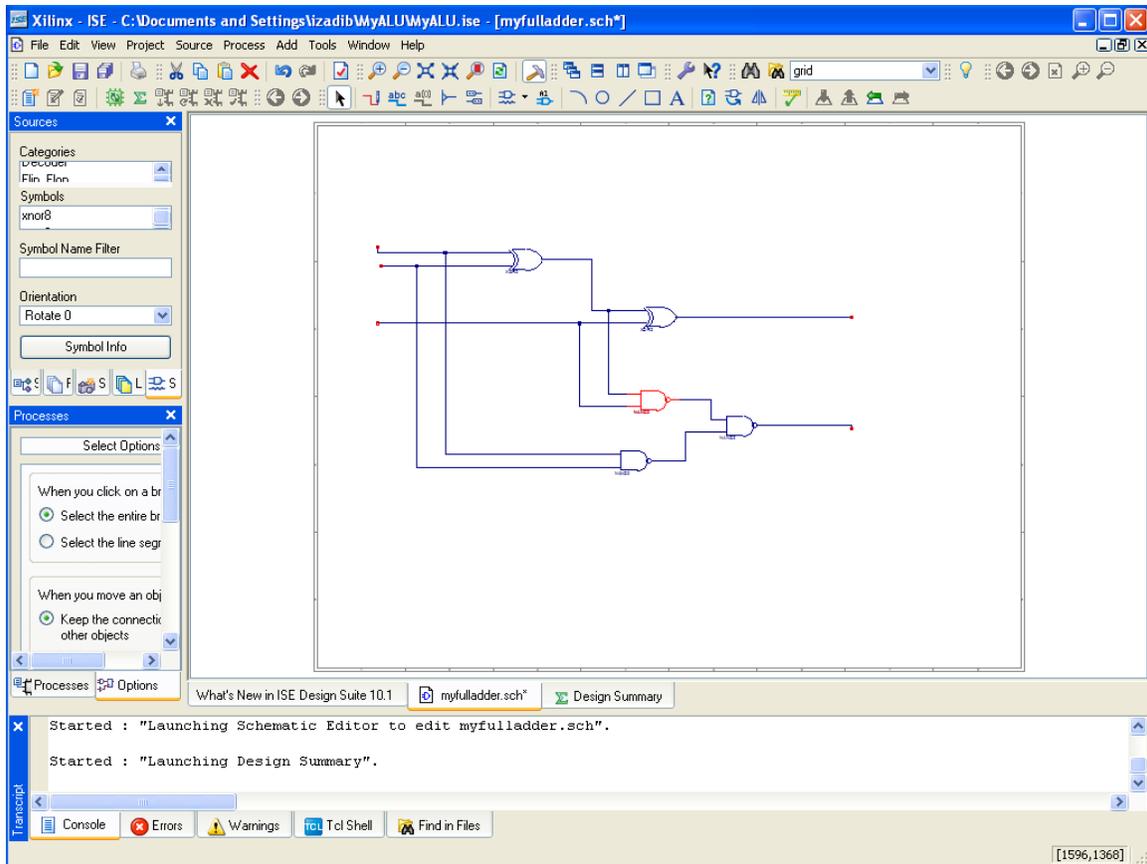


2. Click on the Add Wire  icon. This will allow you to connect the components together. Click on the lead of one gate then on the lead of the next and a wire will be drawn that connects the two together. Note that

$$S = A \oplus B \oplus C$$

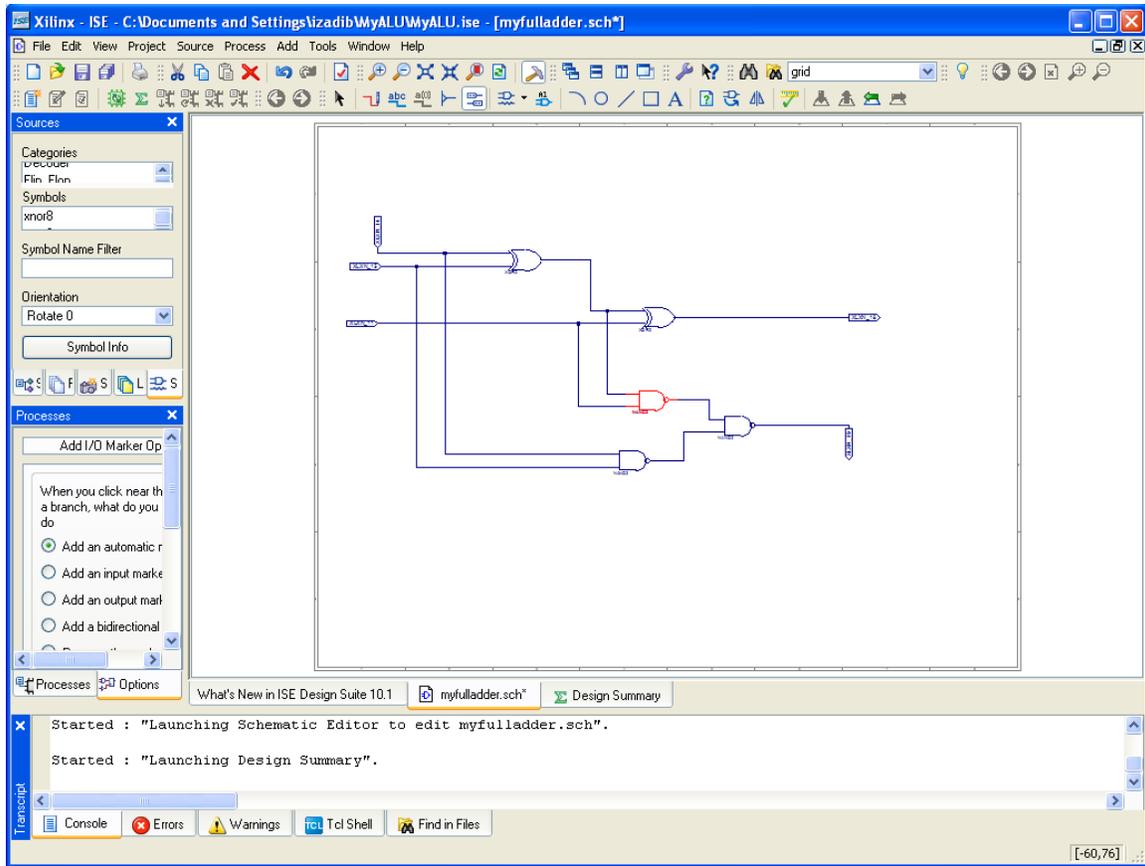
$$\text{Cout} = (A \oplus B) C + AB$$

After the wires have been connected your circuit should look at follow. Make sure to add wires to outputs S and Cout.



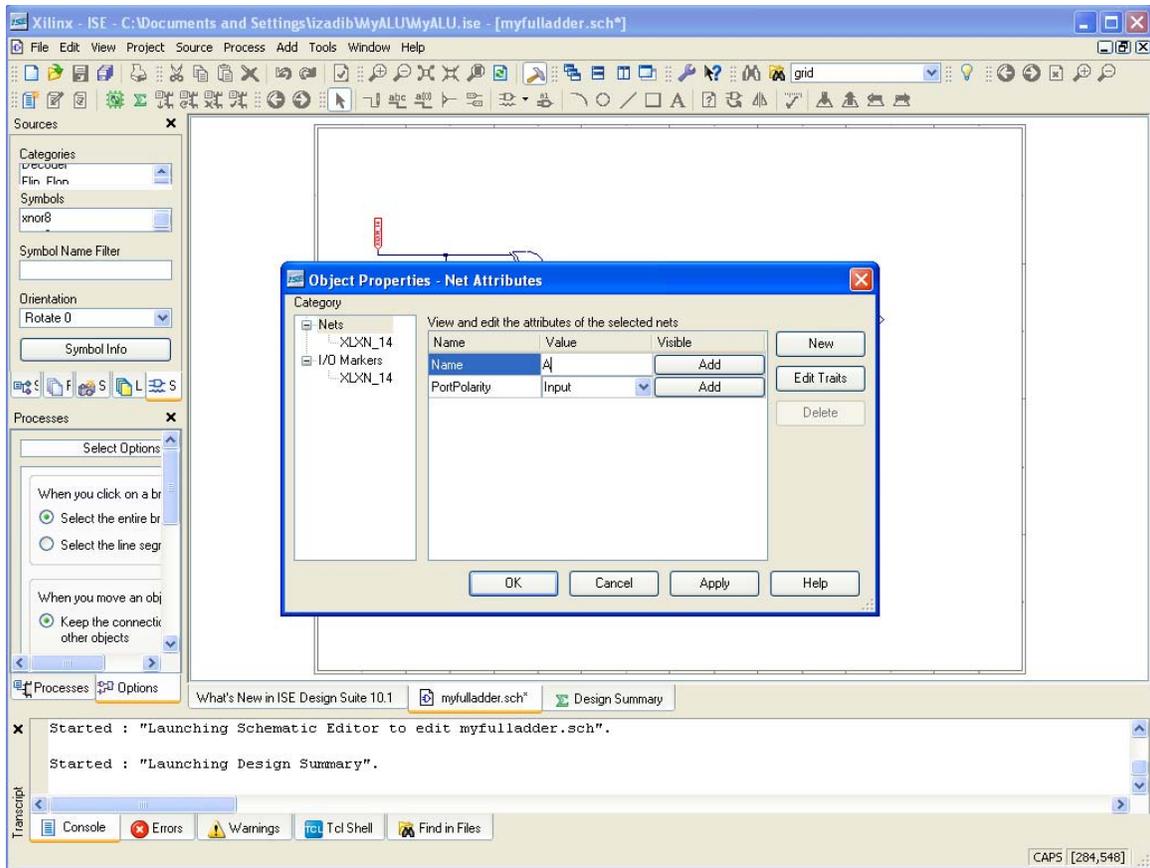
Now we need to add I/O Markers. These markers will help Xilinx determine where the inputs/outputs are when it converts this schematic into a macro.

3. Click the I/O Marker  icon. Place markers on each input and output by clicking on the terminal that you wish to place the marker.



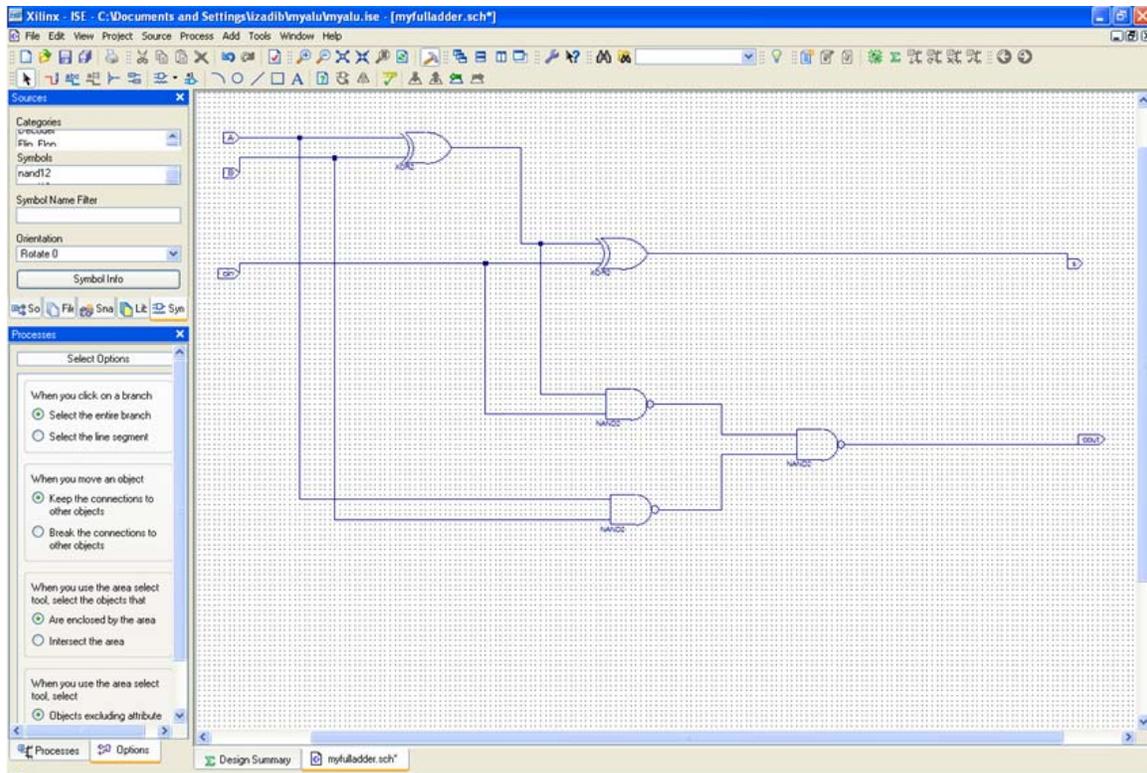
Your schematic should look like this with the markers in place. Xilinx automatically assigns names to the markers, but remembering the names can get confusing really fast.

4. For clarity, change the names. Right click on the marker and rename. Alternatively, you can select the pointer tool  and double click on the marker you wish to rename.



In the Object properties dialog box change the name of the marker by changing the name in the Value field. Click Ok! Similarly rename all your markers.

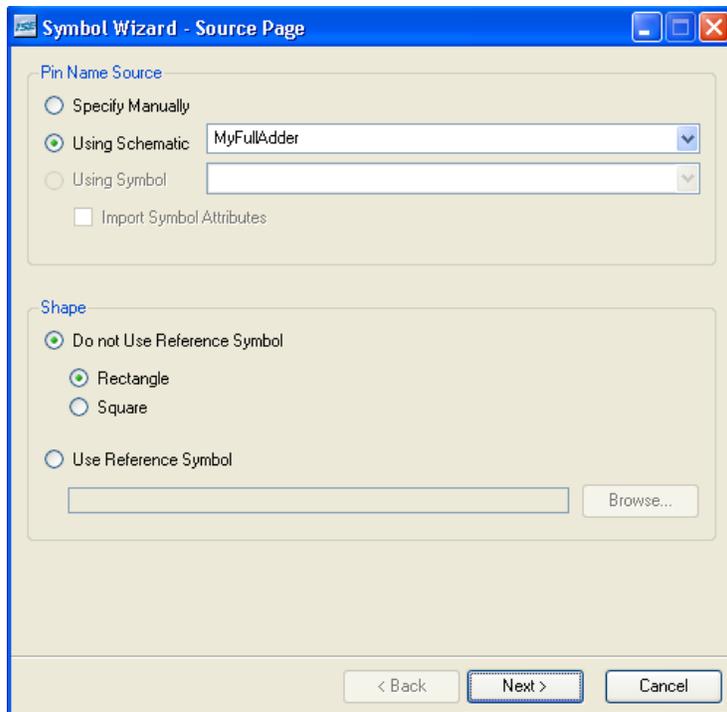
When you are done the schematic will appear as follow.



Creating a Macro

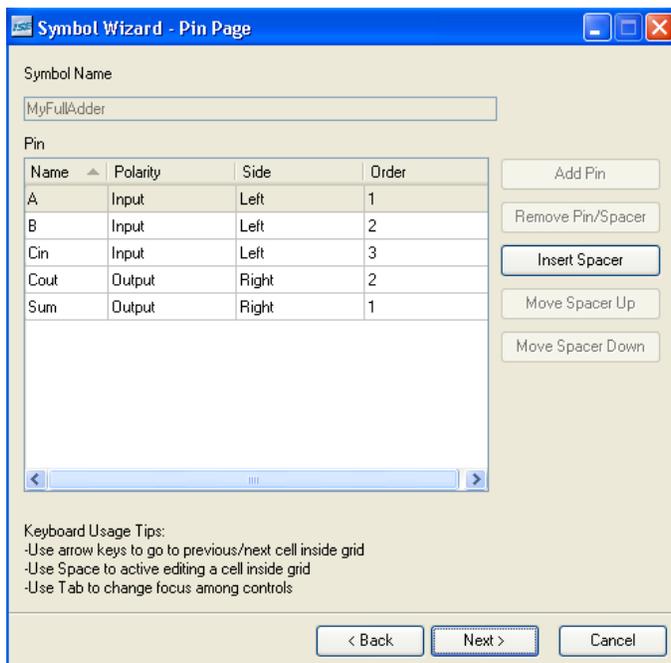
Now that our schematic is complete we would like to create a macro. A macro is user created symbol that can be used just like a component. The advantage of creating a macro is that once you have a macro for your full adder you can use the same macro over and over again.

First, save your schematic! Then from the tool bar select Tools > Symbol Wizard. Select Using Schematic.



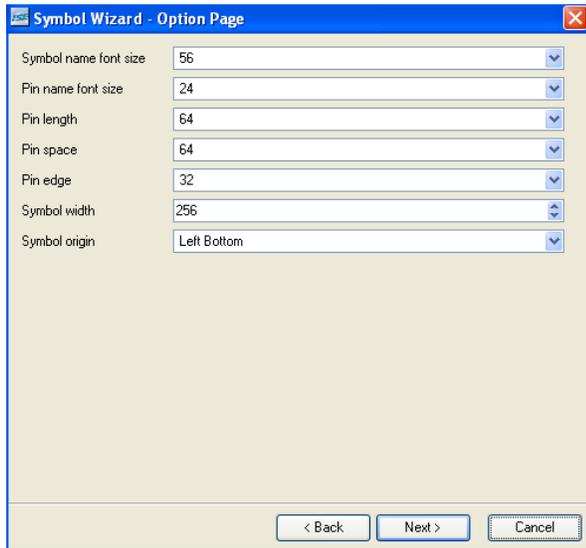
Click Next!

A dialog box with the input and output ports of the schematic open up. Check whether the input and output ports are defined as you intended.



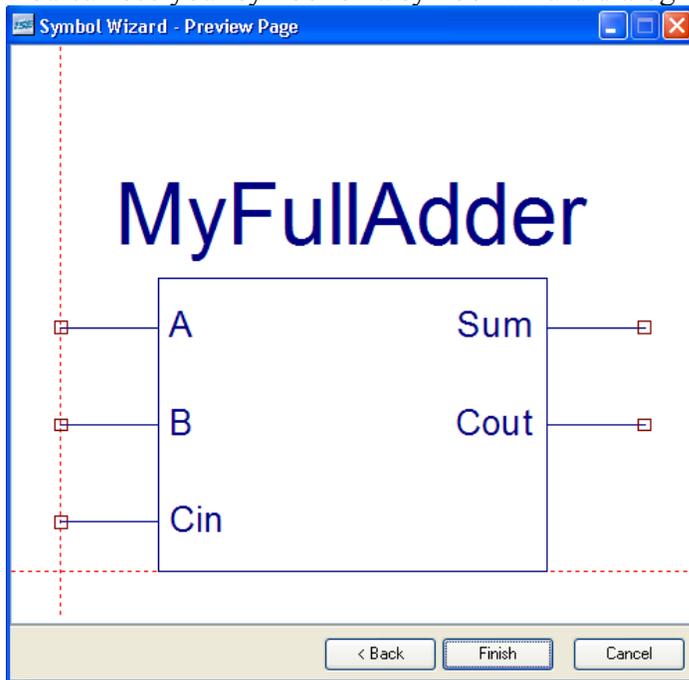
Click Next!

In this dialog box you can change some appearance properties of the symbol being created.



Click Next !

You can see your symbol on a symbol wizard dialog box. Click Finish !

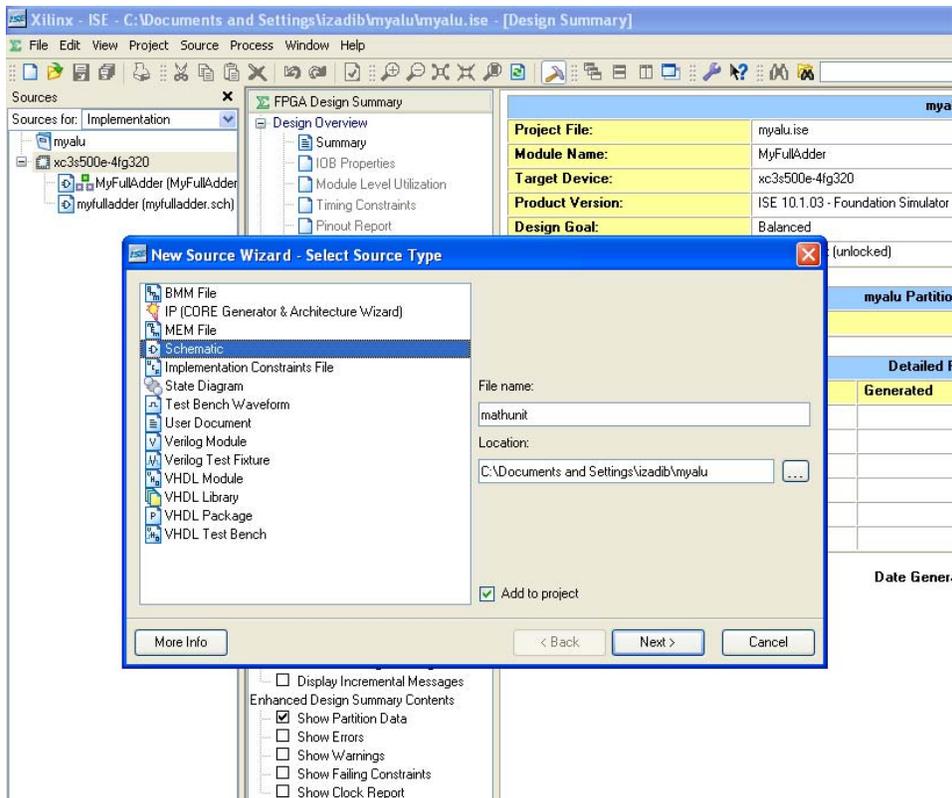
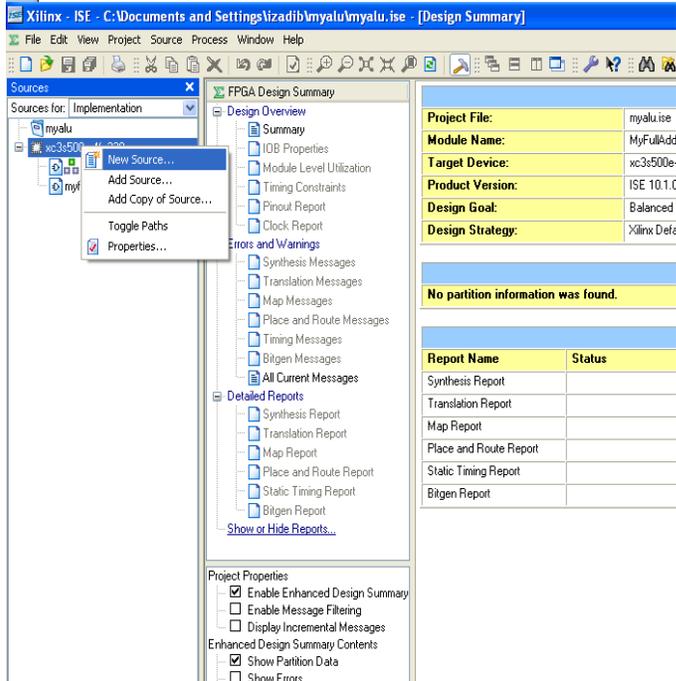


Now close the Symbol editor. Close the schematic we had just been working on, and return to the project navigator.

In the **Project Navigator**, on the left we can see, under the device, is a symbol has been created representing the schematic you just finished working on.

Now lets create a new Schematic.

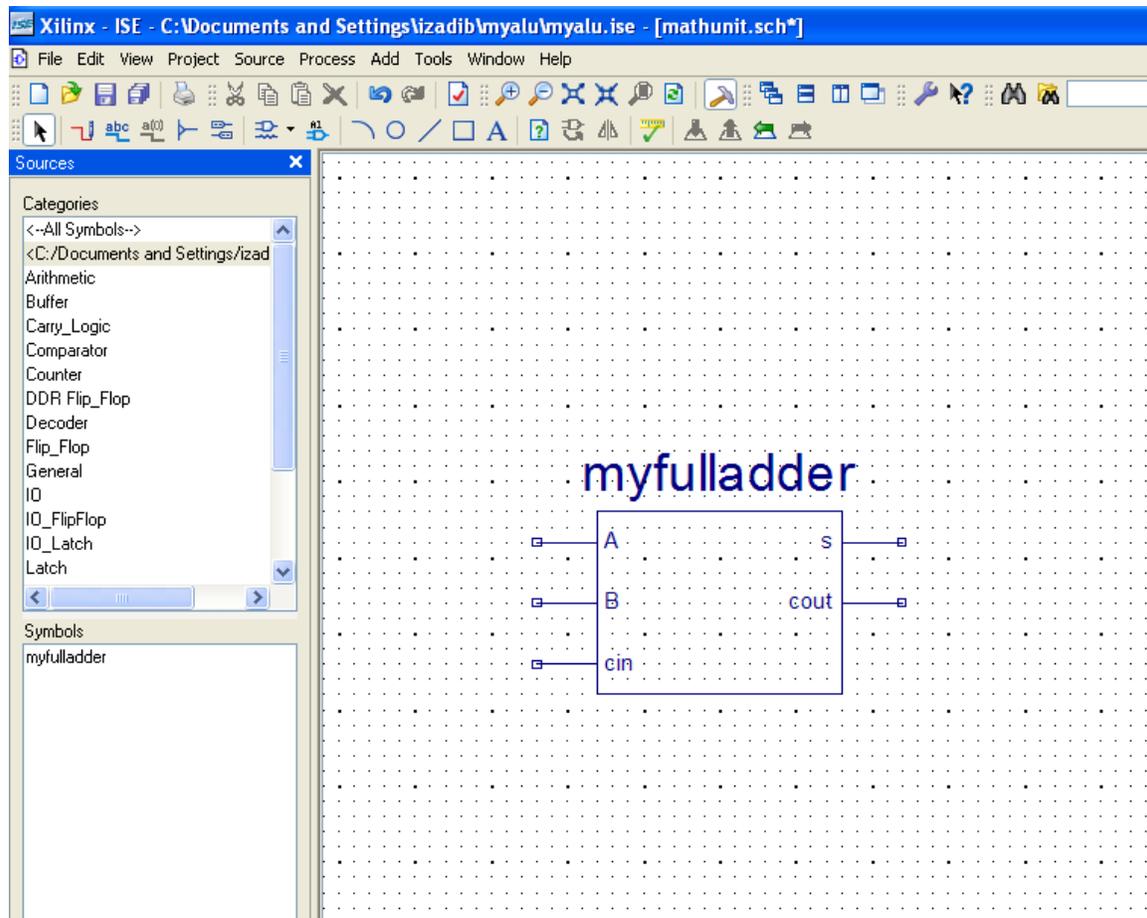
1.Right click on the Device -> New Source...



Select Schematic from the list and name it MathUnit. Next! Finish!

This will reopen the schematic editor (ECS). Now select the symbols tab and look in the Categories Window, a new category has been added. <E:/.../MyALU>, where you can find your full adder.

Click and drag the component onto the workspace.



You have just completed creating your first macro!

Now click on All Symbols in the Categories Window, you can also find your macro there. Try designing a four bit adder or even a four-bit adder/subtractor using your full adder as well as other gates. Good Luck!