EE 140 Digital Design

Project Navigator Tutorial

by Ahmet Burak Yoldemir

Table of Contents

1 Overview

Xilinx Project Navigator is an Integrated Development Environment for digital logic design projects with Xilinx FPGAs. Project Navigator provides a simple way to centrally manage the files in your project as well as automatically invoking all of the other CAD tools. Please make sure that you have correctly installed the required software. This tutorial will help you to use Project Navigator to simulate your circuits. Be aware of the *Tip* parts, they are remedies to problems you will most probably face with.

- TIP 1: No project or file may have a space in the path or filename in Project Navigator.

2 Detailed Instructions: Step 1 – Creating a project

1. Start by opening *Project Navigator* from Desktop or start menu.
2. Go to *File>New Project.*
3. Enter your *Project Name*, specify the *Project Location* and select *Schematic* as Top-Level Module Type.
4. Click *Next* in the following three boxes, and click *Finish* in *New Project Information* screen after making sure that your settings are same as the settings given in Figure 1. Else, click *Back* and make necessary changes.

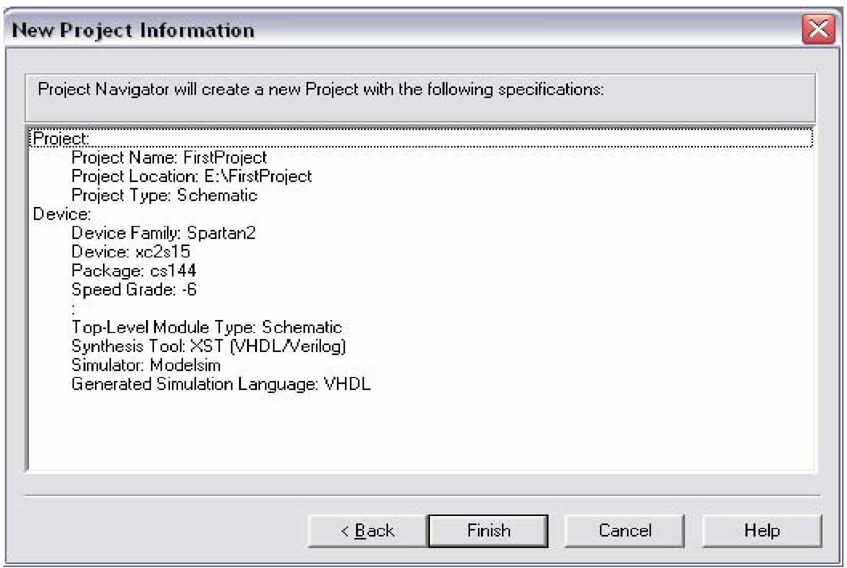


Figure 1. New Project Information

3 Detailed Instructions: Step 2 – Adding files to the project

1. Right click in the Sources in Project box (at the upper left side of the screen), and select New Source. Select *Schematic* form, enter the name of your new file and click *Next* after making sure that *Add to Project* box is selected.
2. After controlling the information about your source, click *Finish*.

Now, you have your project with a schematic source. We are ready for design simulation.

4 Detailed Instructions: Step 3 – Adding symbols to schematic source

Symbols are graphic representations of components. Usually, symbols are electrical components, such as gates.

1. Click *Add>Symbol.*
2. Select the category of the component you want to add, than choose it from the *Symbols* list. Most of the components you will be firstly using are in *Logic* category. Logic category contains gates such as and, or, nor, inverter etc. and different types of these gates. You might also need VCC and GND, which you can find in *General* category. In Figure 2, a sample set of gates is supplied for your guidance.

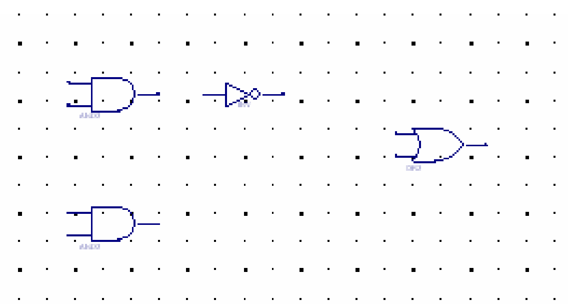


Figure 2.Sample Set of Gates

5 Detailed Instructions: Step 4 – Adding wires between components

1. Click *Add>Wire.*
2. Click two terminals of the wire you want to connect.

In Figure 3, you can see our sample set of components wired.

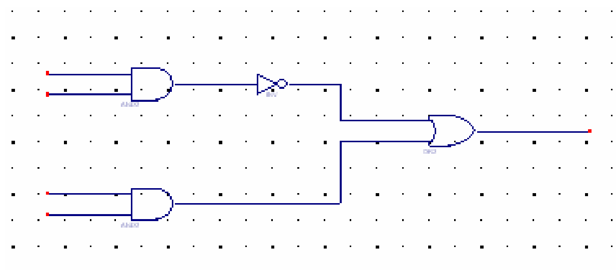


Figure 3. Wiring components

- TIP 2: You can press Esc anytime you want to stop adding the current circuit component such as wires, buses, gates, I/O Markers etc.

6 Detailed Instructions: Step 5 – Adding input/output markers

1. Click *Add>I/O Marker.*
2. Choose the marker type you want to add from the *Add I/O Marker Options* box on the left.
3. Click the terminal you want to add a marker to.

Figure 4 shows our sample circuit with input and output markers added.

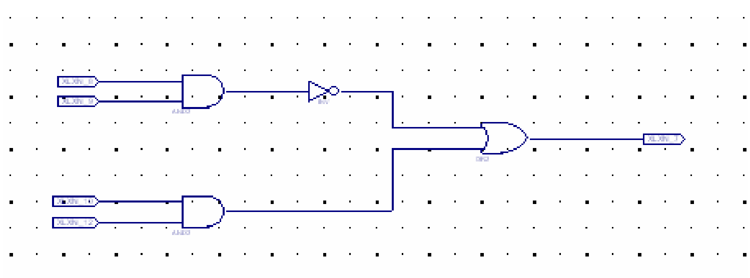


Figure 4. Adding I/O Markers

- TIP 3: You are strongly advised to change the input/output marker's name in a more understandable form such as IN1, IN2, OUT1 etc. Project Navigator assigns default XLXN_(rand#) names to the markers and it is very hard to trace the outputs of a circuit when you do not rename them.

7 Detailed Instructions: Step 6 – Creating a test bench for simulation

Now, you will create a test bench waveform containing input stimulus you can use to simulate your circuit. You will use the Waveform Editor to create a test bench waveform (TBW) file.

1. In the main Project Navigator window, right click to your project name from *Sources in Project* box you have formerly created, and then click *Test Bench Waveform*. Enter the name of your test bench, and click *Next*.

2. Select the schematic source to which you want to attach your test bench, and click *Next*.
3. Check whether your new test bench information is correct, and click *Finish.*
4. You will now see the *Initialize Timing* window. As this is an elementary tutorial, you should not bother with the left side of this box, click *Combinatorial Design (or internal clock)* from right side, and click *OK*.
5. Now you can enter your inputs to the ports that are shown in blue. To change the value of a signal, simply click on the blue square corresponding to the time interval you want. A sample input set is prepared for you in Figure 5.

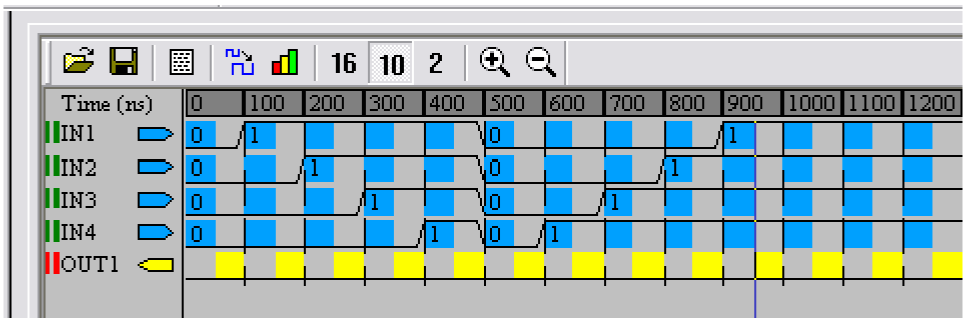


Figure 5. Determining the input sequence

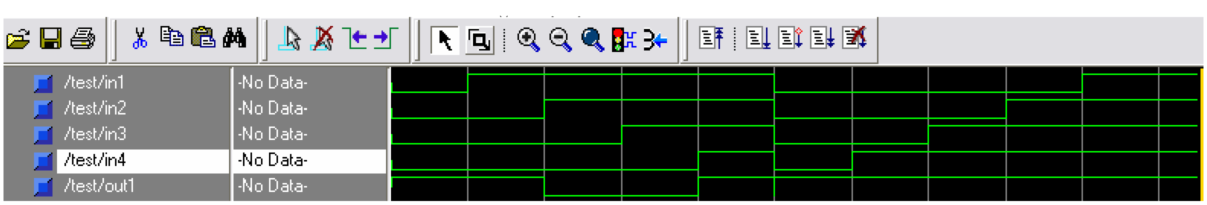
➤ TIP 4: If you cannot see the test bench you have created in the Sources in Project window, press Ctrl+S to save your project. Your test bench will most probably come in sight now.

8 Detailed Instructions: Step 7 – Simulating your circuit

Now that we have our test bench and inputs, we are ready to generate the outputs of our circuit.

1. From *Sources in Project* box, click the test bench you have created.
2. From the *Processes for the Source* box beneath, double click *Simulate Behavioral Model*.
3. If there is no error with your ports or wiring, a ModelSim window opens and generates the outputs according to the inputs you have in your test bench.
4. You can observe the outputs from the Wave window that is opened in ModelSim. I recommend you to maximize the Wave window to analyze signals easily.

Figure 6. Output waveform screen



- TIP 5: If you think your simulation results are meaningless, try clicking Zoom Out several times. It shows the waveforms for the last time interval.

9 Detailed Instructions: Step 8 – Creating modules (subcircuit/symbol)

1. Open your schematic source, then click *Tools>Symbol Wizard*.
2. Select *Using Schematic,* after choosing your source name, click *Next.*
3. Select the order of the pins of your module, and click *Next.*
4. Make necessary font size and component size changes if you wish, click *Next*.
5. Now you see the subcircuit you are about to create. Click *Finish.*
6. The subcircuit you have created is just like any other ordinary symbol in Project Navigator. You can use that symbol by choosing the target file path that you have saved your project to from *Categories* box.

In Figure 7, you can see our new symbol, with the analogous circuit we have formerly formed.

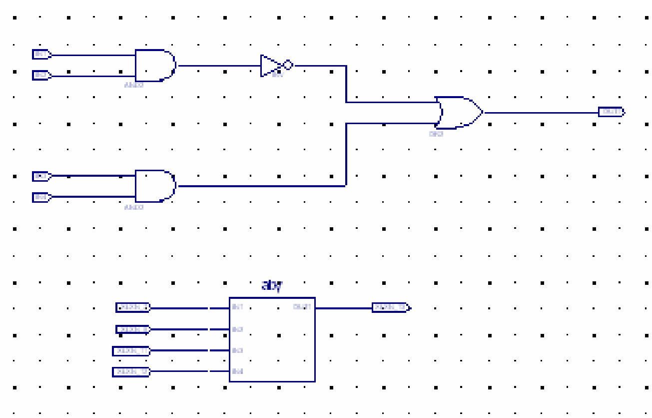


Figure 7. Two analogous sample circuits

10 Detailed Instructions: Step 9 – Using Bus Taps to Transmit Multiple Signals With a Single Wire

To point out that we'll transmit multiple signals with a single wire, we first click the icon ("Add net name"). Then we specify a name from the *Add Net Name Options* box on the left side. The standard for specifying net names is as follows:

Name [0:x] → Name: The name you'll specify

→ x: Signal # - 1 (we subtract 1 as we start from the 0^{th} signal)

After specifying the name, we click on the wire that we want to convert into a multiple signal carrying net. At this instant, Project Navigator makes the net thicker to make it distinguishable from ordinary cables. Then, we add bus taps to points we wish using the icon. After this process, we can use the ends of bus taps as any other pin. In the figures below, you can see a 4-bit data carrying net, I/O Markers and I/O's in the simulation waveform of the circuit. Note that input contains 4 bits of data although we have only one input marker.

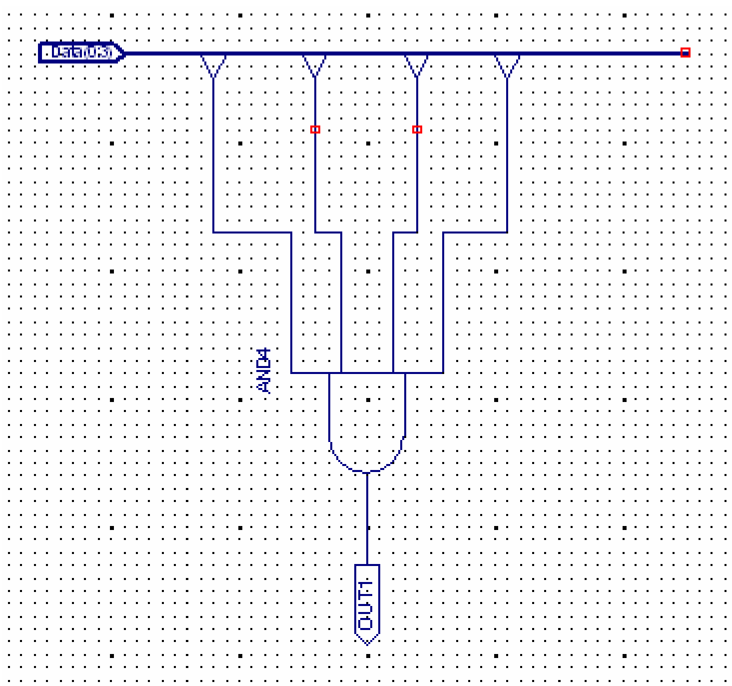


Figure 8. Using Bus Taps

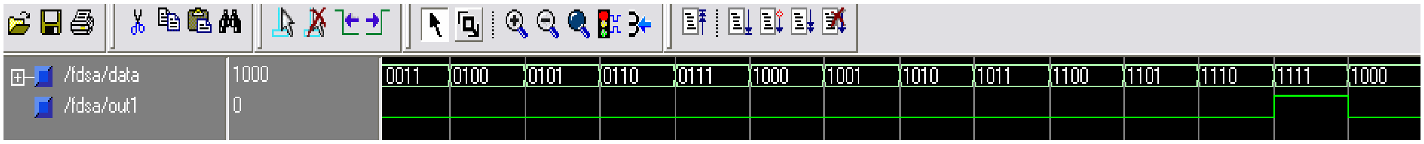


Figure 9. Simulation results of the circuit in Figure 8

You can always get further information from *Help* menu, from the ISE7 Deep Tutorial you can get from http://www.xilinx.com/support/techsup/tutorials/tutorials7.htm.