Tutorial on
FPGA Design Flow
based on
Xilinx ISE Webpack
and ModelSim

ver. 2.0

Prepared by Marcin Rogawski, Ekawat (Ice) Homsirikamol, and Dr. Kris Gaj

The example codes used in this tutorial can be obtained from
http://ece.gmu.edu/coursewebpages/ECE/ECE448/S09/experiments/448_lab3.htm.

The current version of the tutorial was tested using the following tools:

CAD Tool

- Xilinx ISE Webpack Version : 9.1

Synthesis Tool

- ISE&Webpack Synthesis&Implementation Version : 9.1
- Synplicity Synplify Pro Version : 8.6

Implementation Tool

- Xilinx ISE/WebPack Version : 9.1

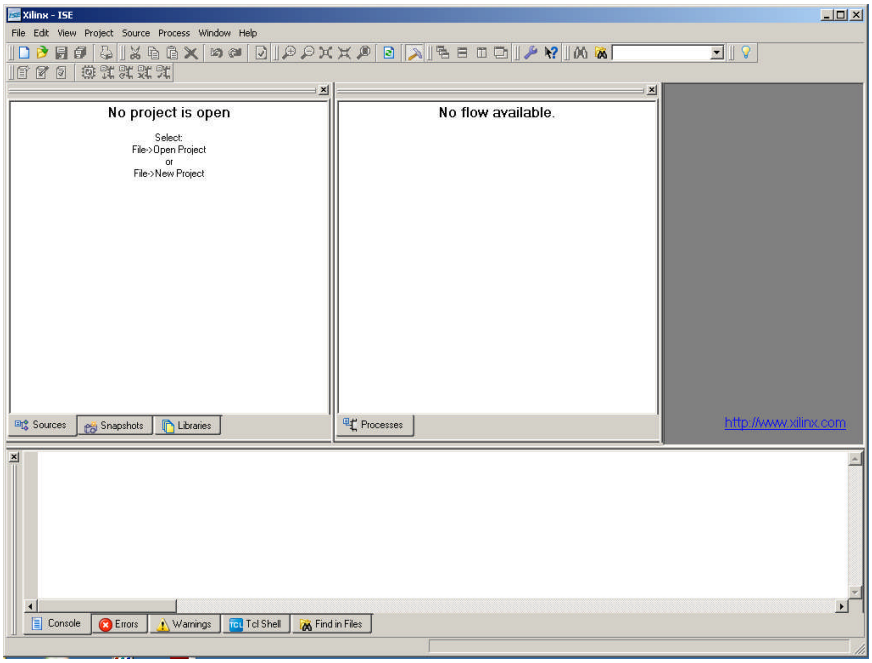
FPGA Board

- Celoxica RC10

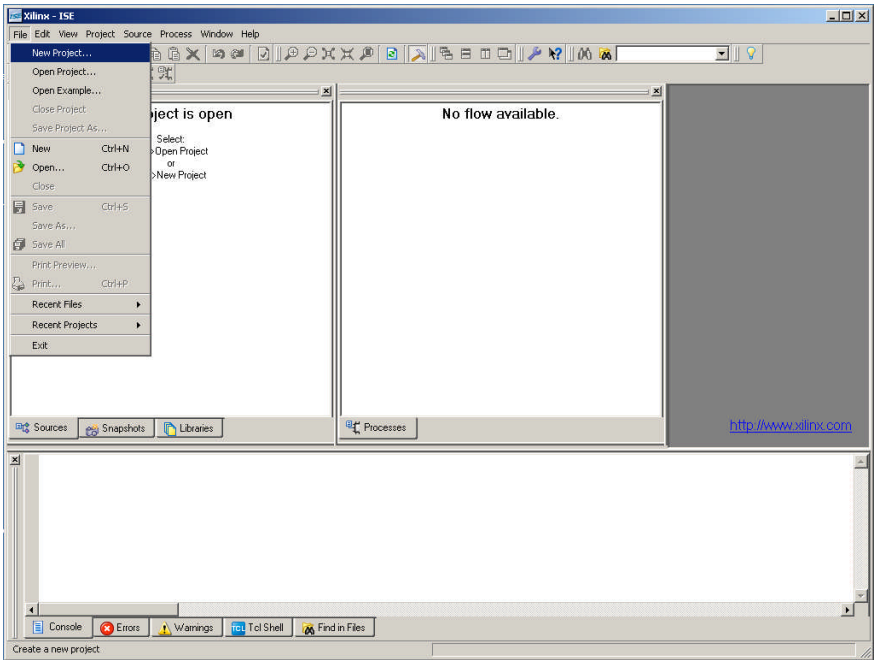
Table of Contents

1. Project Settings

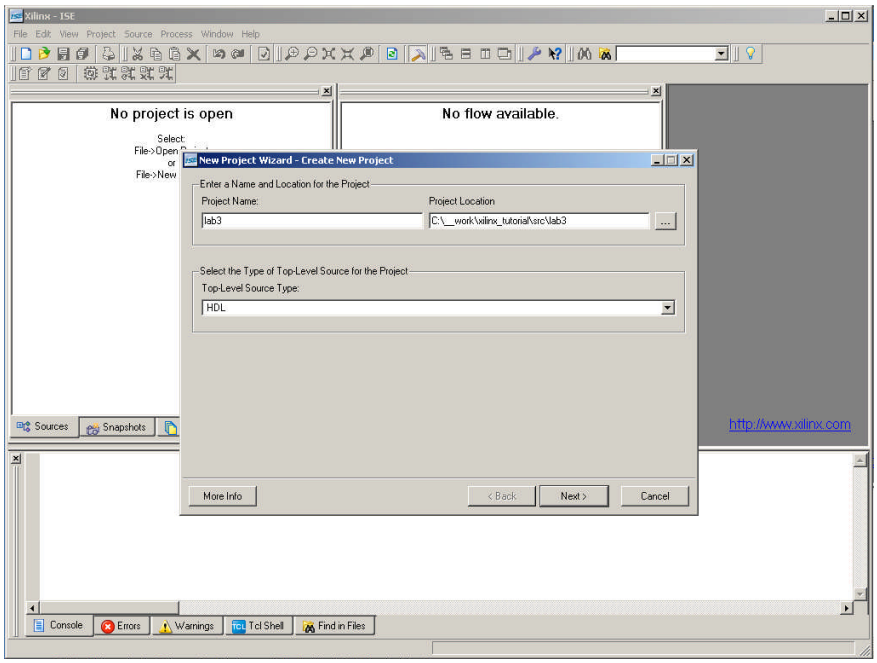
To Start Xilinx ISE go to:
Start->All Programs-> VLSI Tools->Xilinx ISE 9.1i->Project Navigator



To start new Project go to: menu File->New Project

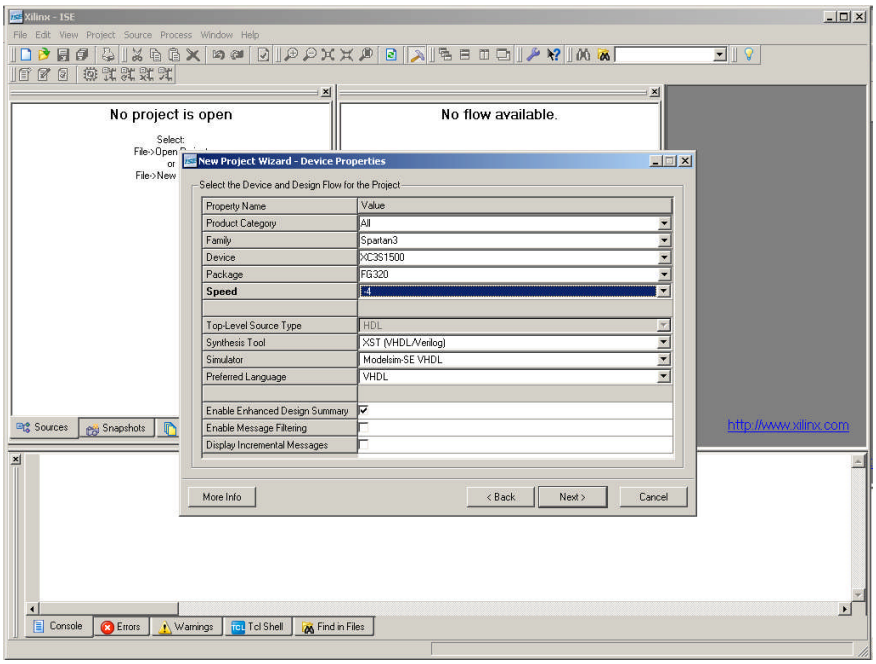


Specify Project Name and location for project files

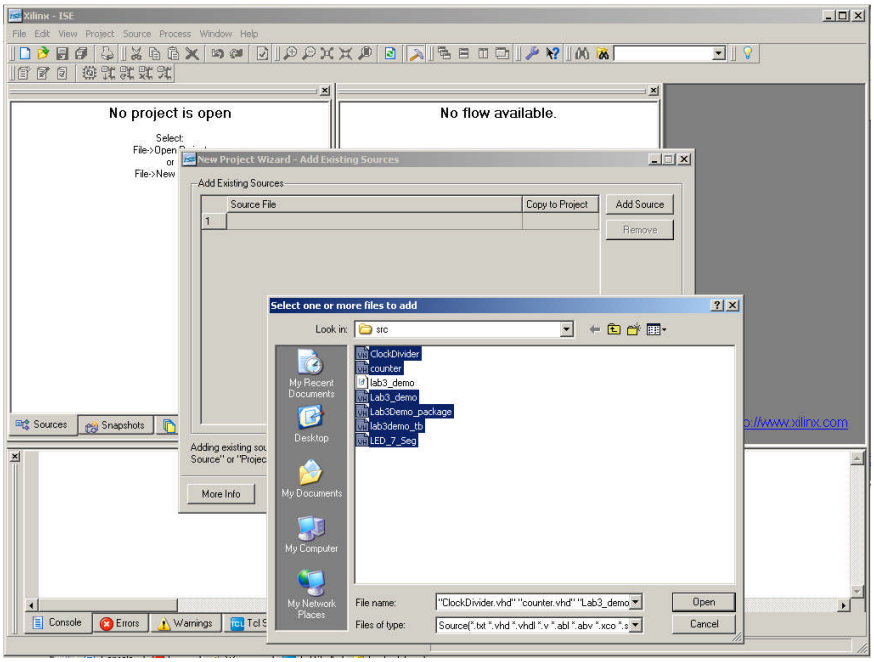


Click **Next** and specify

Family	:	Xilinx9x Spartan3
Device	:	3s1500
Package	:	fg320
Speed Grade	:	-4
Simulator	:	ModelSim SE VHDL



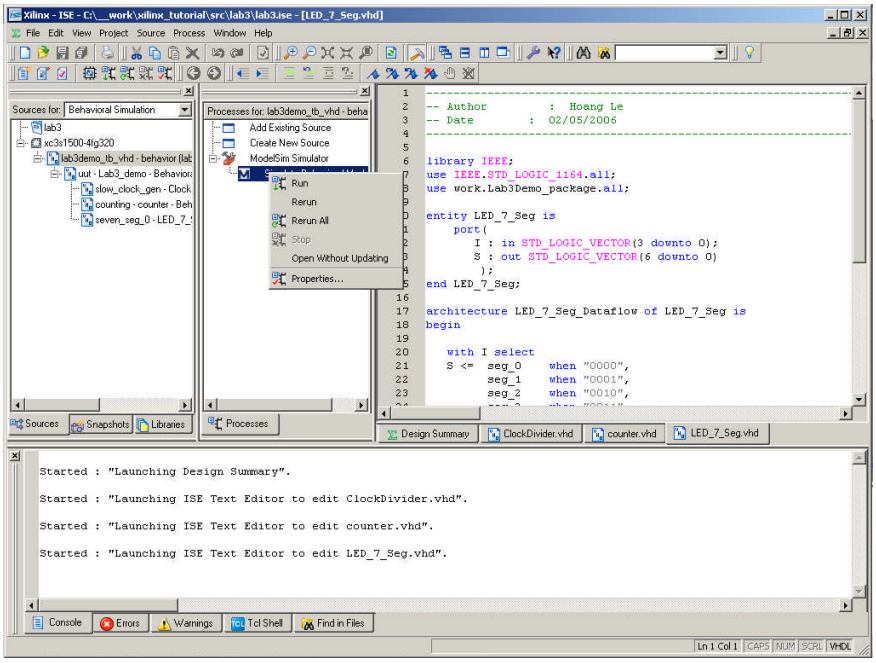
Click **Next** and then **Add Source**. Choose files for the project.

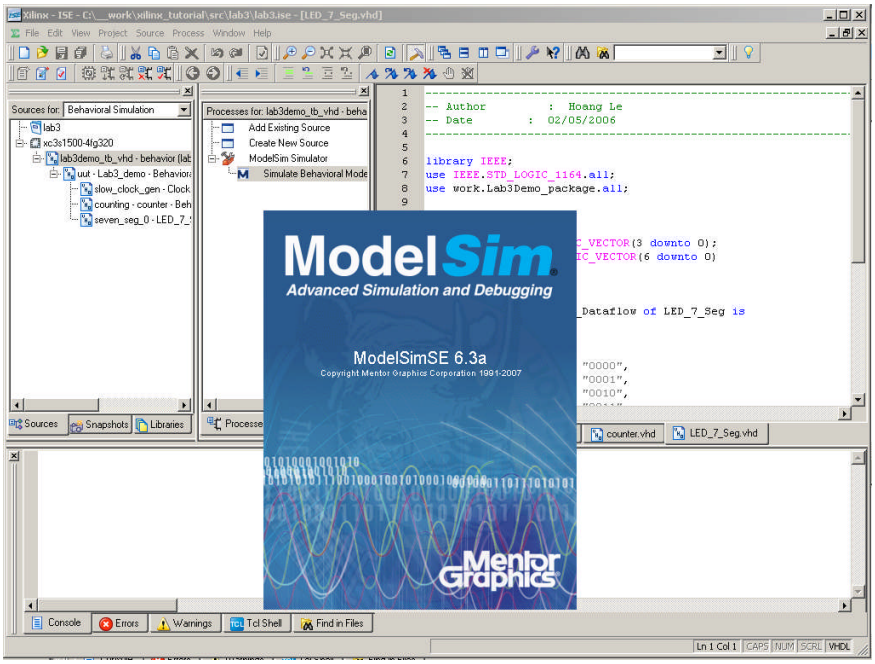


Then click **Open** and **Finish**. Your project has been defined.

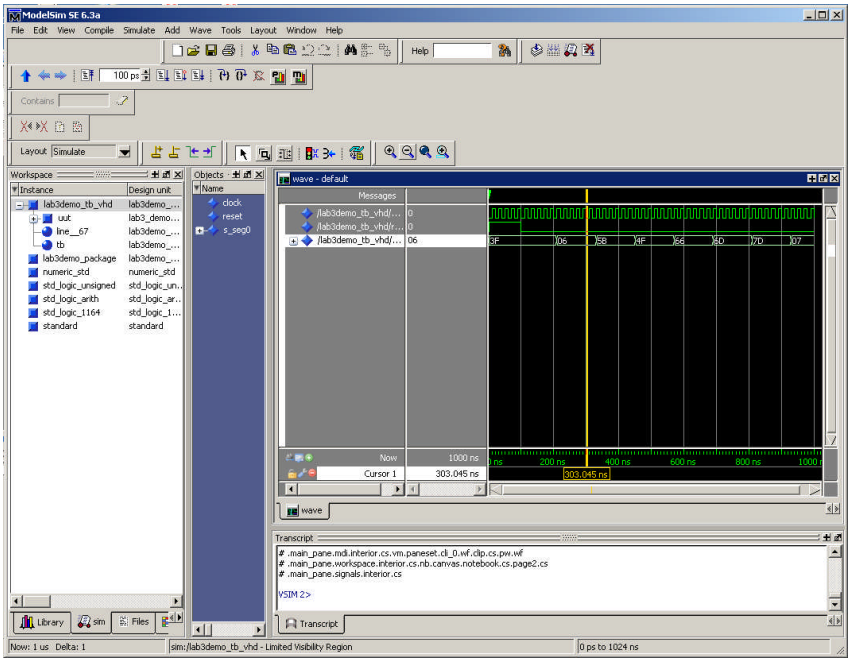
2. Behavioral simulation

In the menu S**ources for** we choose **Behavioral Simulation**. The testbench for this design was set as a top level. In the menu **Processes for** choose **ModelSim Simulator**, right-click and choose **Run**.





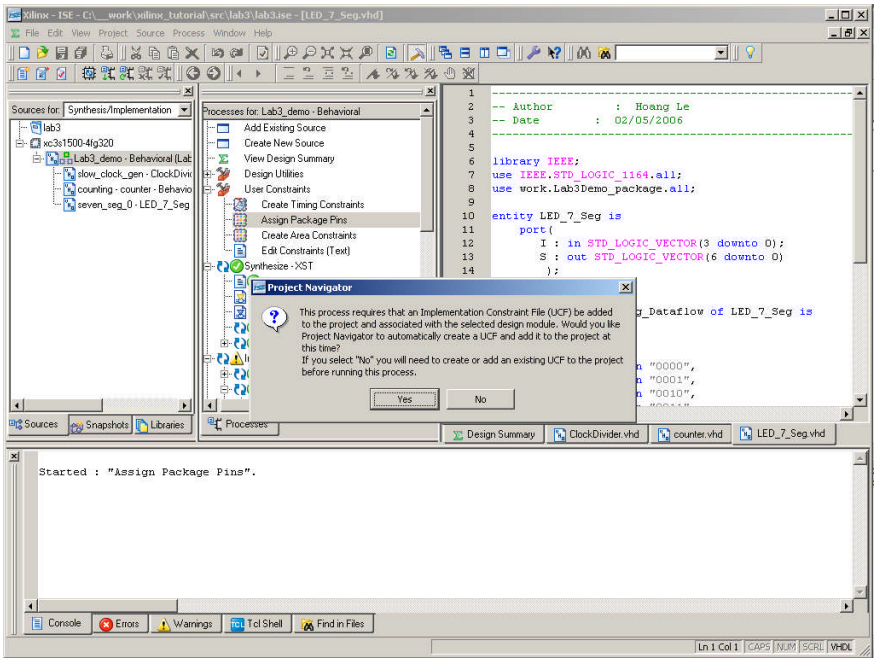
You will see a picture similar to the one above.

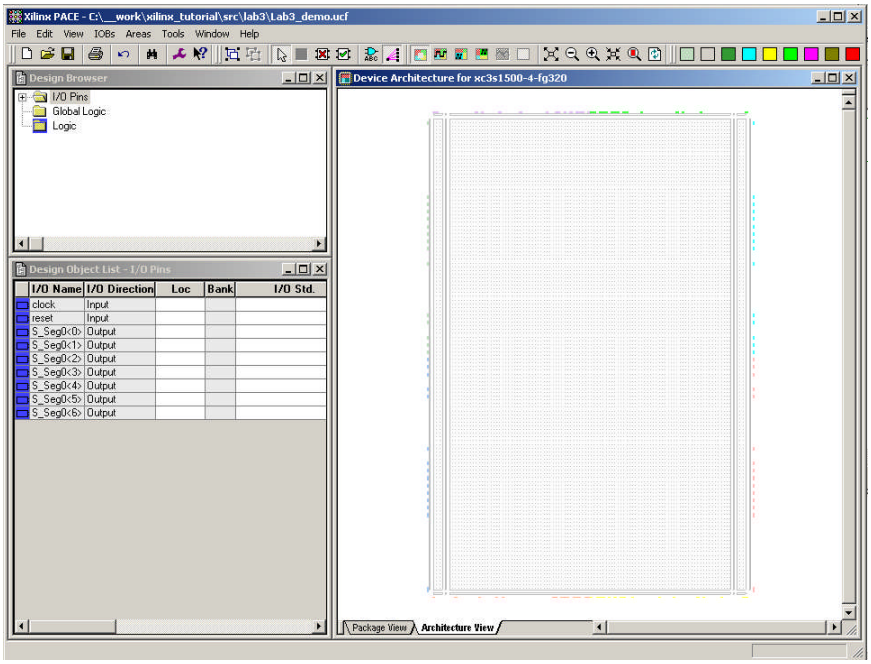


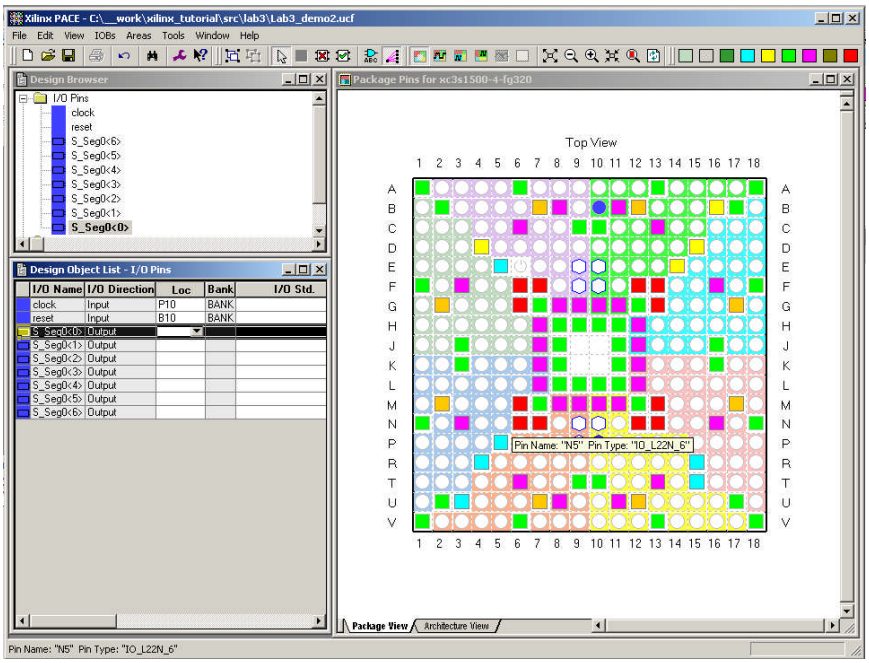
Finally you should be able to observe waveforms for your design.

3. Pin Assignment

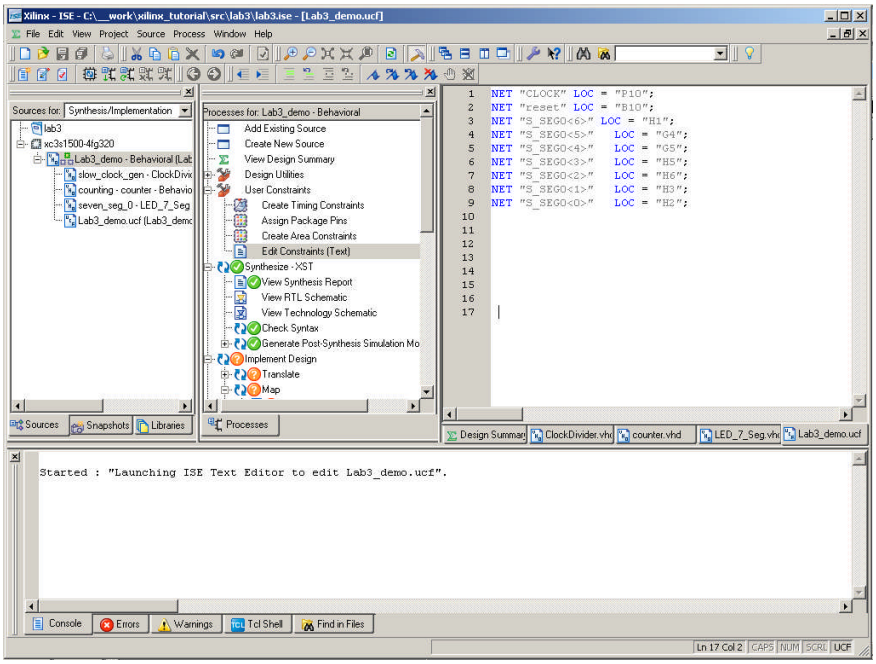
Go to menu **Sources for** and change this option to **Synthesis/Implementation.** Click on **Assign Package Pins**. The User Constraint File (UCF) will be created if you decide to assign design port names to the physical pins. Columns I/O Names and I/O Descriptions represent ports from your design. The Loc column should be used to input the location of the corresponding pin in the FPGA device.







We can specify Pin Assignments by going to **User Constraint** menu. We can assign design port names to the physical pins of a chosen device by option **Edit Constraints** (Text). Keyword **NET** is for a port name assignment and keyword **LOC** is for a physical pin assignment.

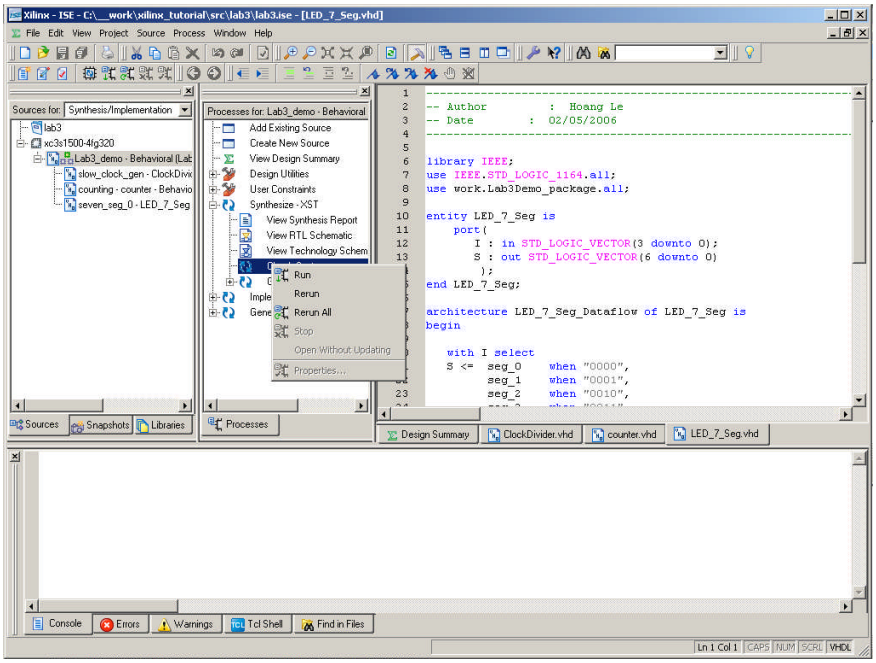


4. Synthesis and Implementation

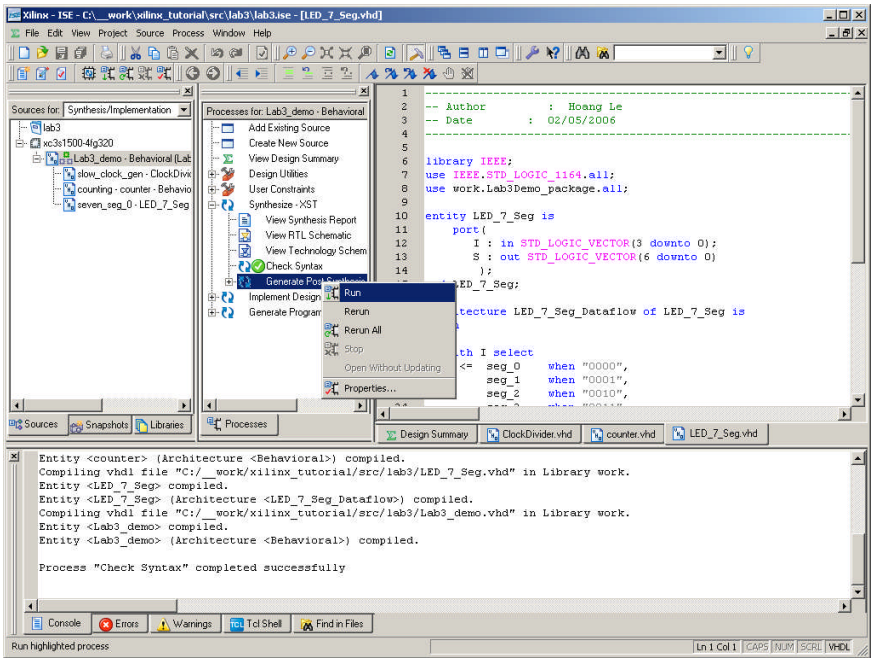
4.1 Synthesis

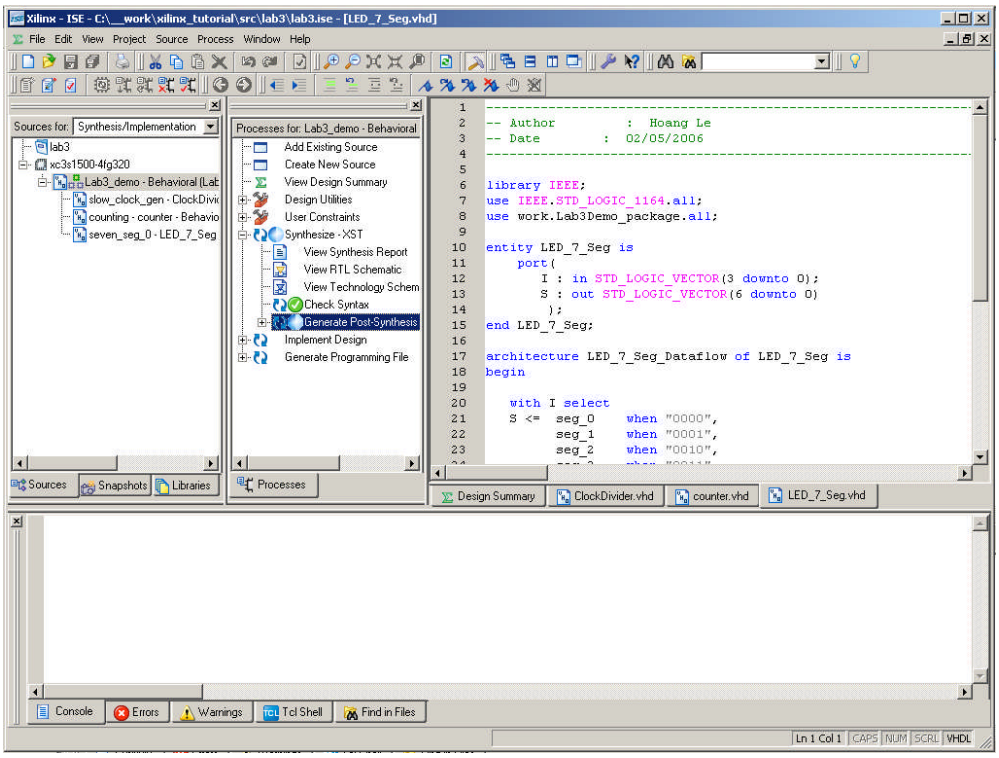
4.1.1 Synthesis with Xilinx XST

Go to the menu **Sources for** and change this option to **Synthesis/Implementation.** Click **Check Syntax** to check if vhdl sources are properly coded.

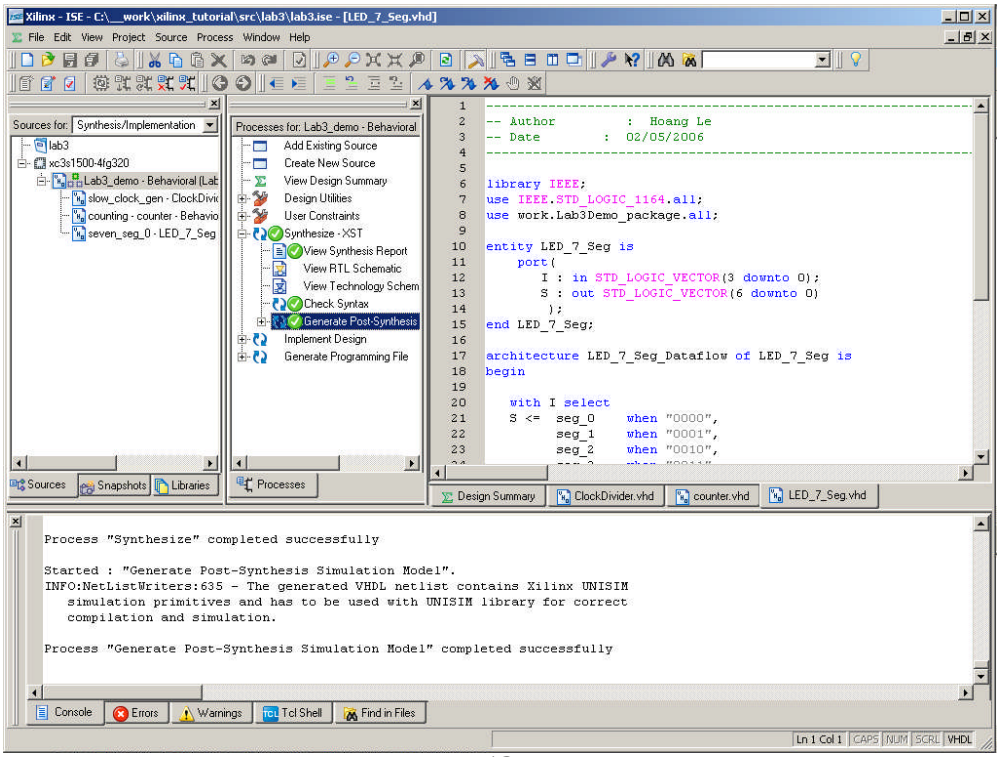


When you right-click on **Generate Post-Synthesis Simulation Model** then pop-up menu appears. Choose**Run** to start synthesis (There is no option to simulate post-synthesis netlist for Xilinx ISE WebPack).



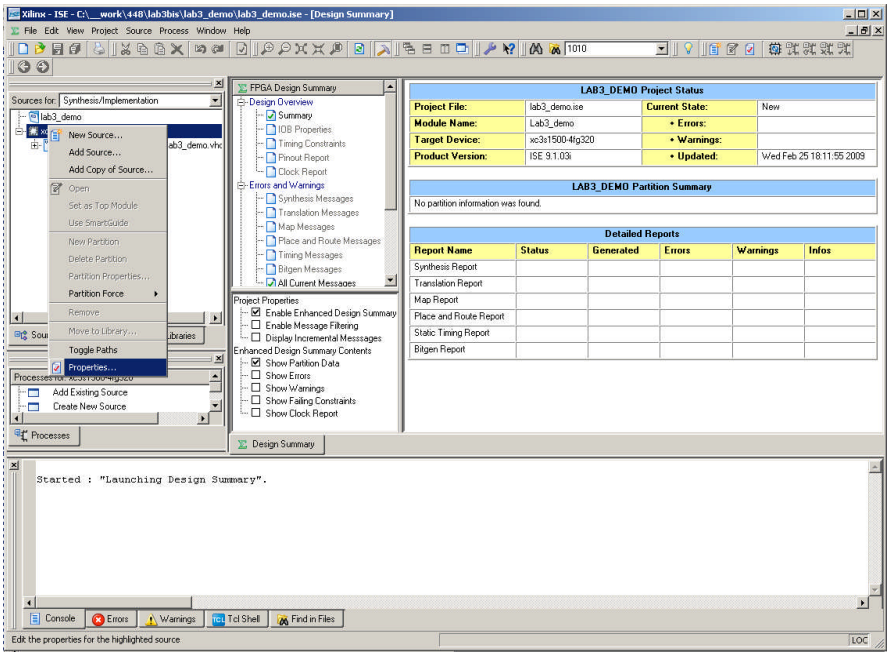


Synthesis and generation of Post-Synthesis Simulation Model were completed successfully and we can start Implementation part.

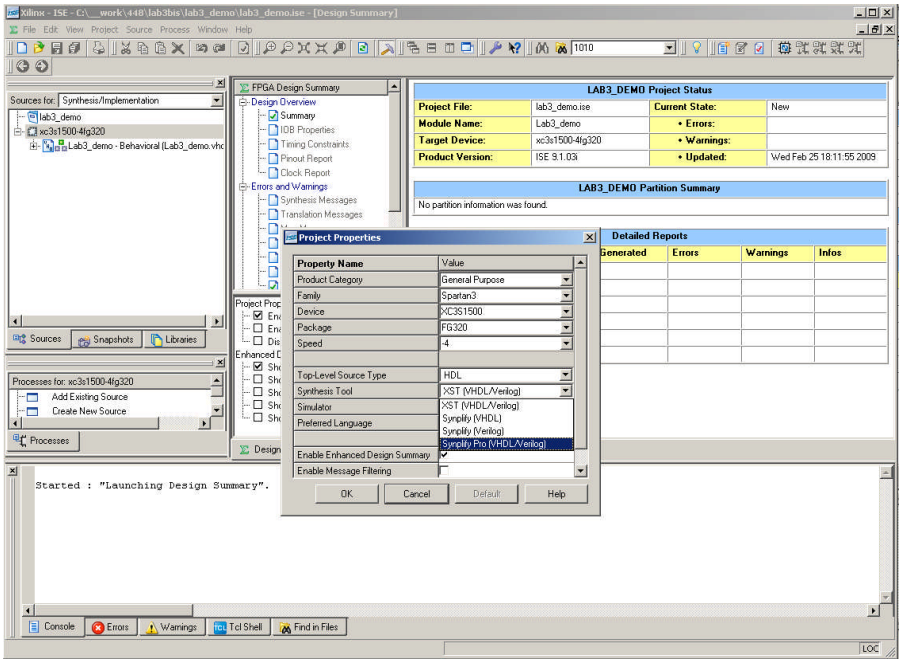


4.1.2 Synthesis with Simplify Pro

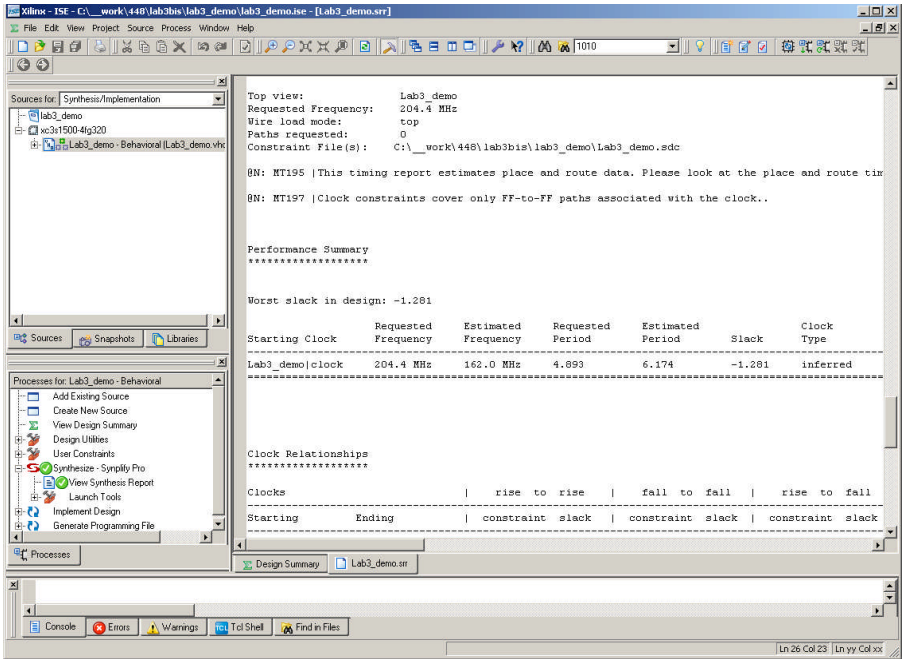
There is a possibility in Xilinx ISE to specify other vendors of synthesis software. Go to **Sources for** menu, right-click on the name of the chosen design. Pop-up menu should appear. Choose **Properties.**



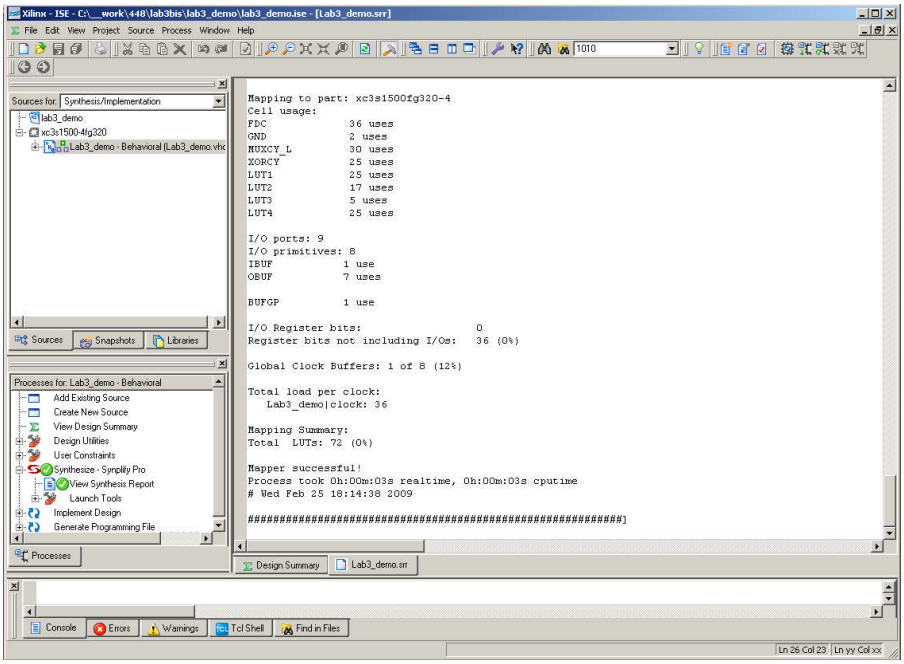
The **Project Properties** menu should be available now. Change Synthesis Tool to Synplify Pro (VHDL/Verilog). Click **OK –** your design will use Synplify Pro for synthesis.



When Synthesis process is completed, the report from synthesis becomes available. Some of the most important features of the design are the minimum clock period and the maximum clock frequency. We can find these two parameters in the report file from Synthesis. Please remember that the values of these parameters after synthesis are different than the values of the same parameters after implementation.

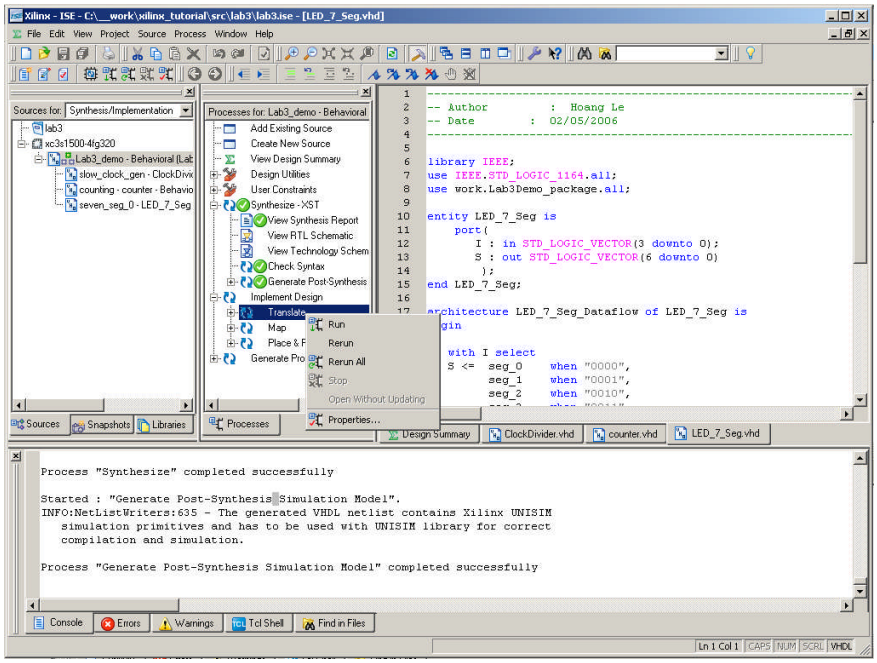


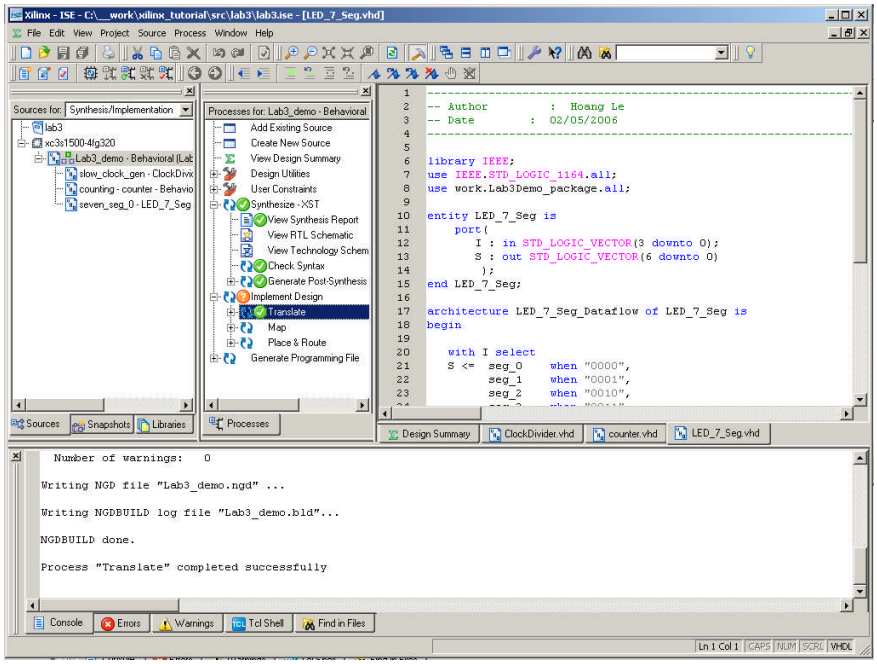
The other important information we can find in this report is the amount of FPGA resources your design requires.



4.2 Translate

The first step in the Xilinx Design Flow for implementation is Translate. Under **Implement Design** option, choose **Translate**, and then **Run**.

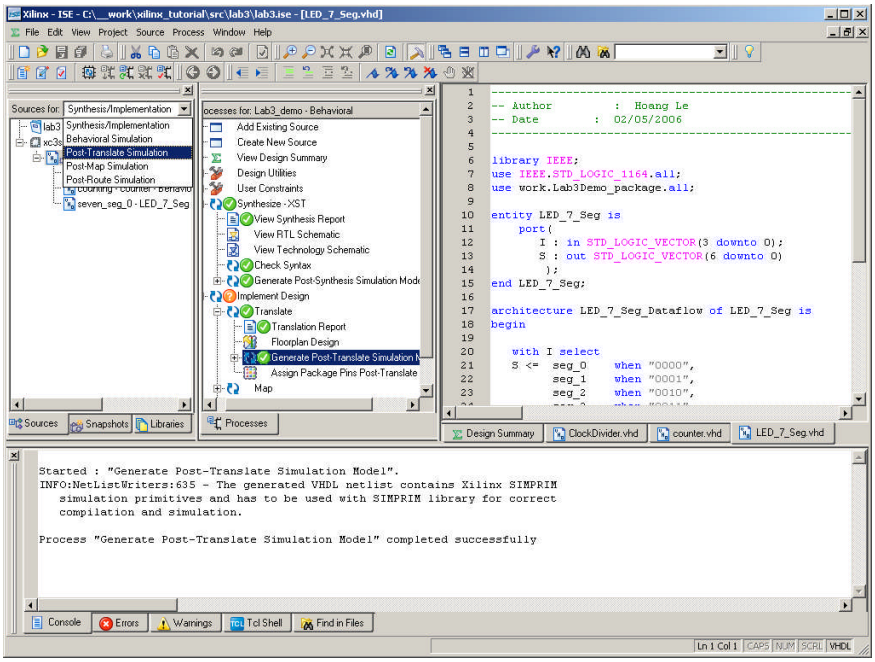




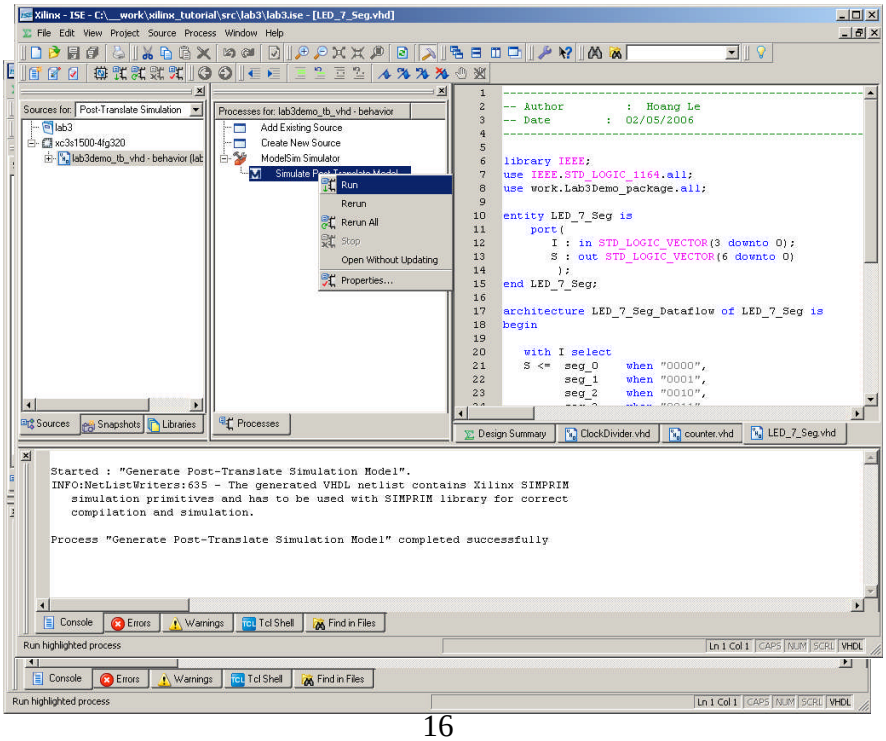
If you are successful with this part you should generate Post Translate Simulation Model.

4.3 Post-Translate Simulation

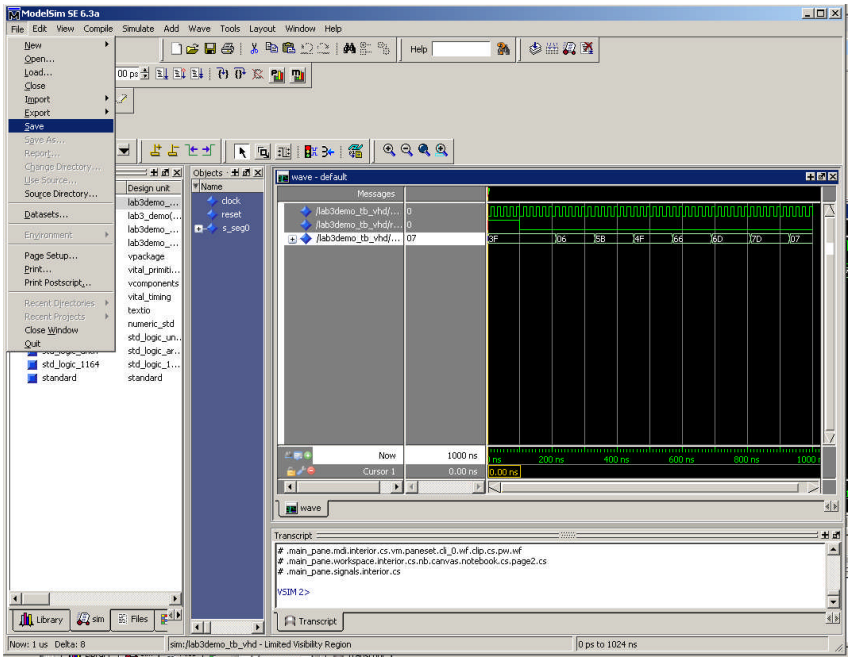
When you right-click on **Generate Post-Translate Simulation Model** then pop-up menu appears. Choose **Run** to start.



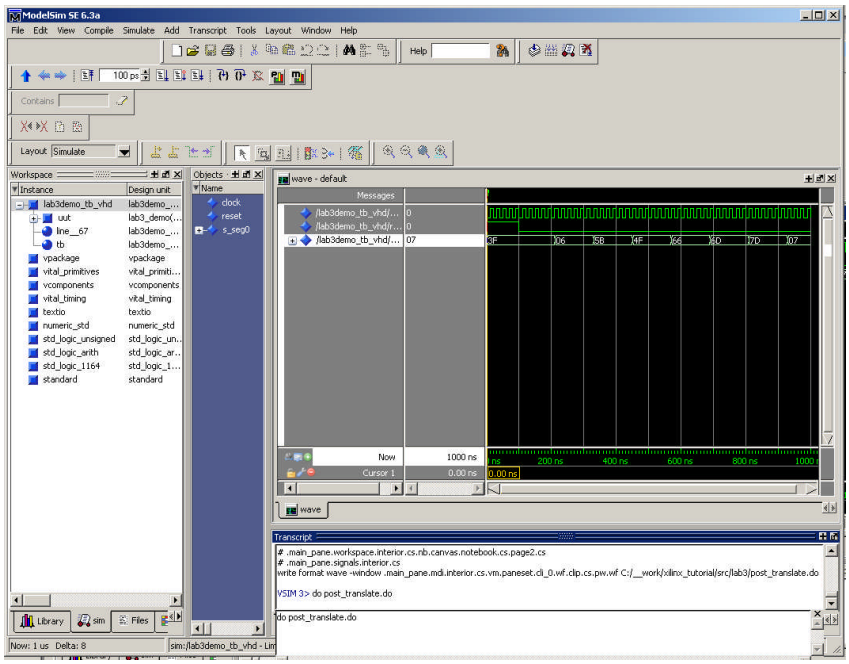
In the menu S**ources for** we choose **Post-Translate Simulation**. The testbench for this design was set as a top level. In the menu **Processes for** choose **ModelSim Simulator**, right-click and choose **Run**.



ModelSim SE will be launched and waveforms for our design should appear. We can save script for further re-simulation by saving .do type files.

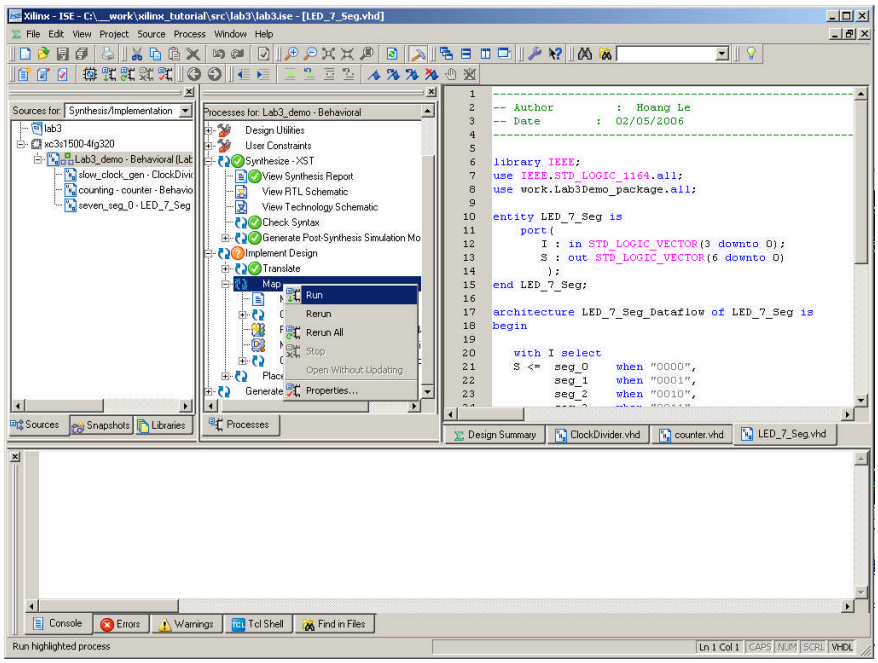


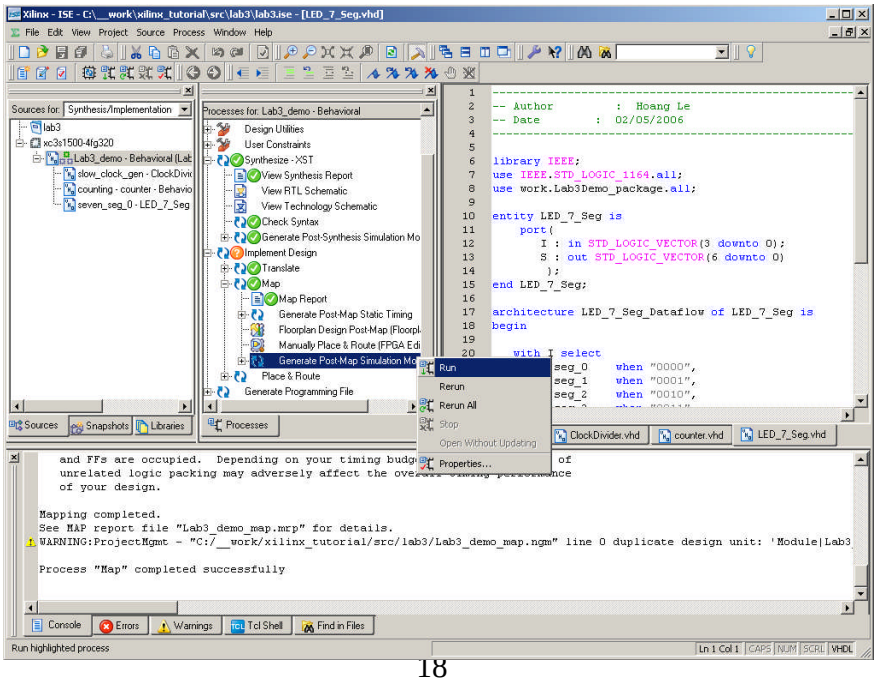
There is in the bottom part ModelSim terminal. We can type there commands such like ***do name_of_script.do*** and our waveform will be updated. It is possible to use terminal in every step of implementation.



4.4 Map

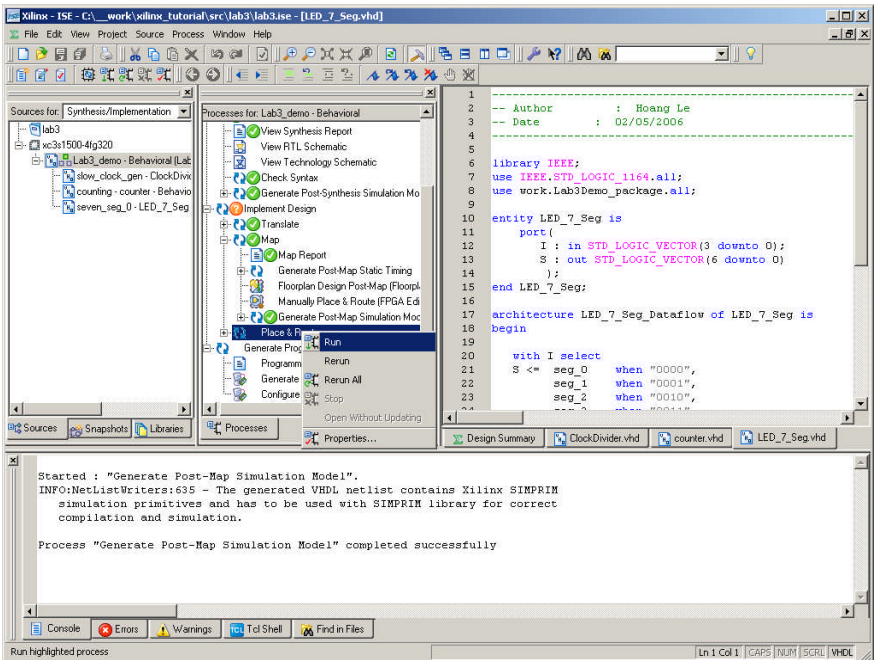
Go to the menu **Sources for** and change this option to **Synthesis/Implementation.** From the **Implement Design** menu choose **Map.** Do right-click and pop-up menu should appear. Choose **Run** to start mapping process.



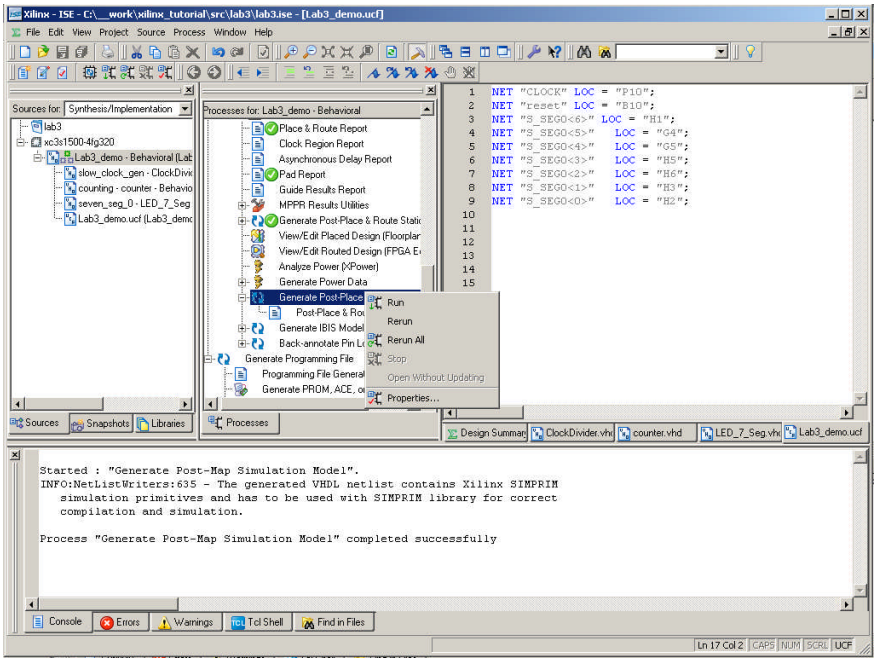


4.5 Place and Route

Go to menu **Sources for** and change this option to **Synthesis/Implementation.** From the **Implement Design** menu choose **Place and Route.** Do right-click and pop-up menu should appear. Choose **Run** to start place and route process.

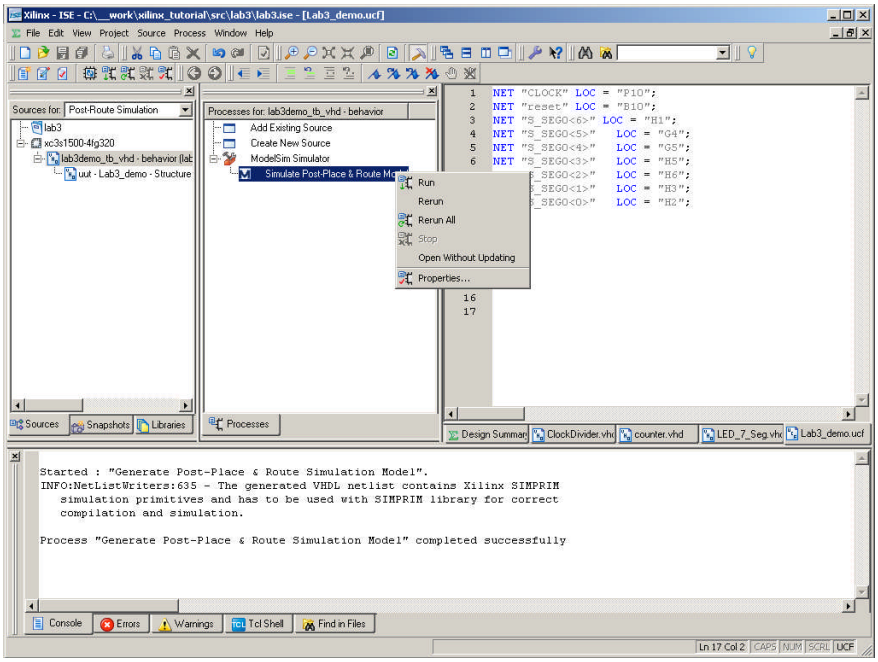


When we right-click on Generate Post Place and Route Simulation Model then pop-up menu appears and you should choose

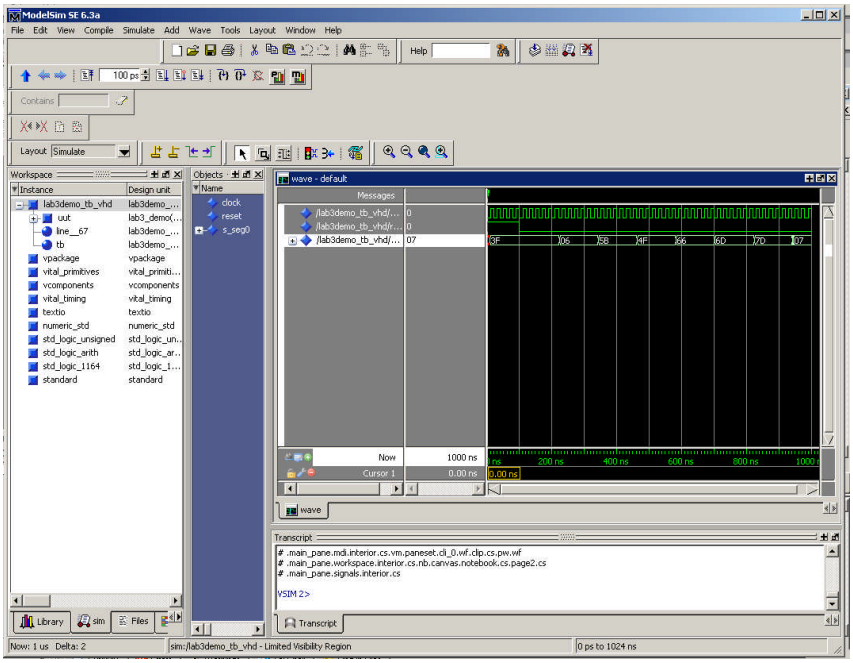


4.6 Post Place and Route Simulation

In the menu S**ources for** we choose **Post-Route Simulation**. The testbench for this design was set as a top level. In the menu **Processes for** choose **ModelSim Simulator**, right-click and choose **Run**.

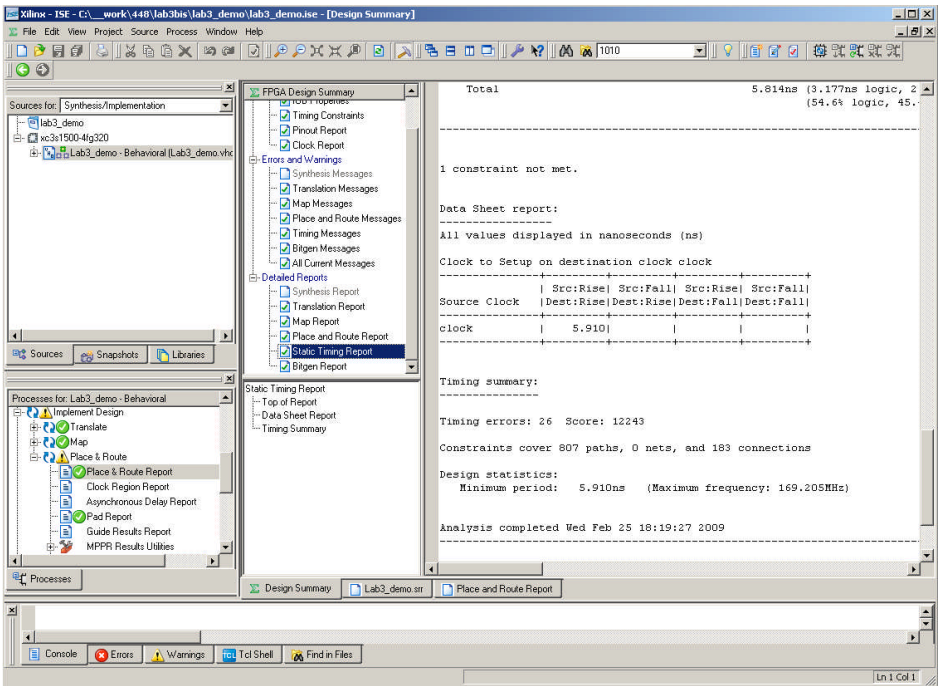


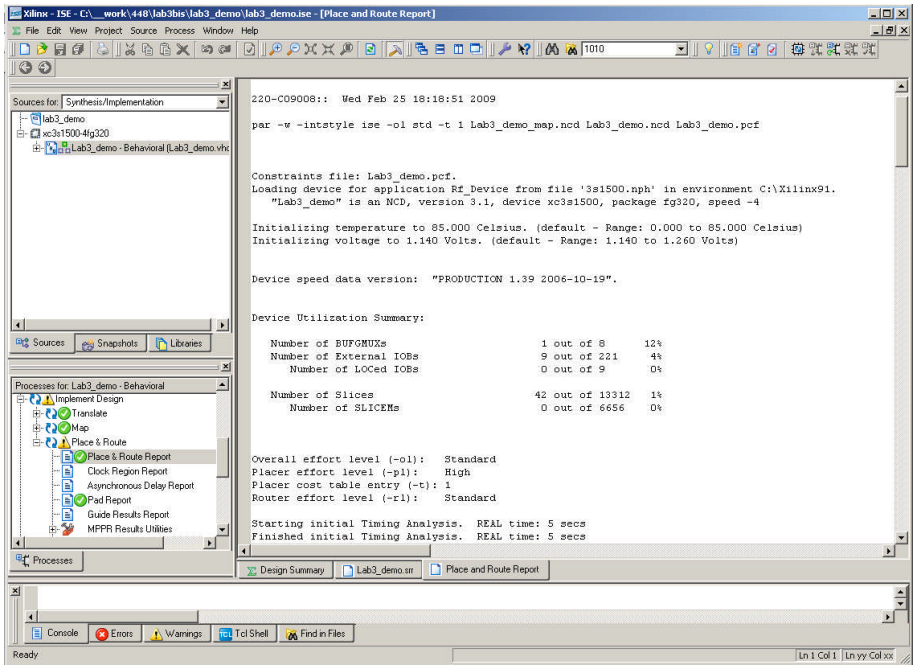
ModelSim SE will be launched and waveforms for our design should appear.



4.7 Implementation Reports

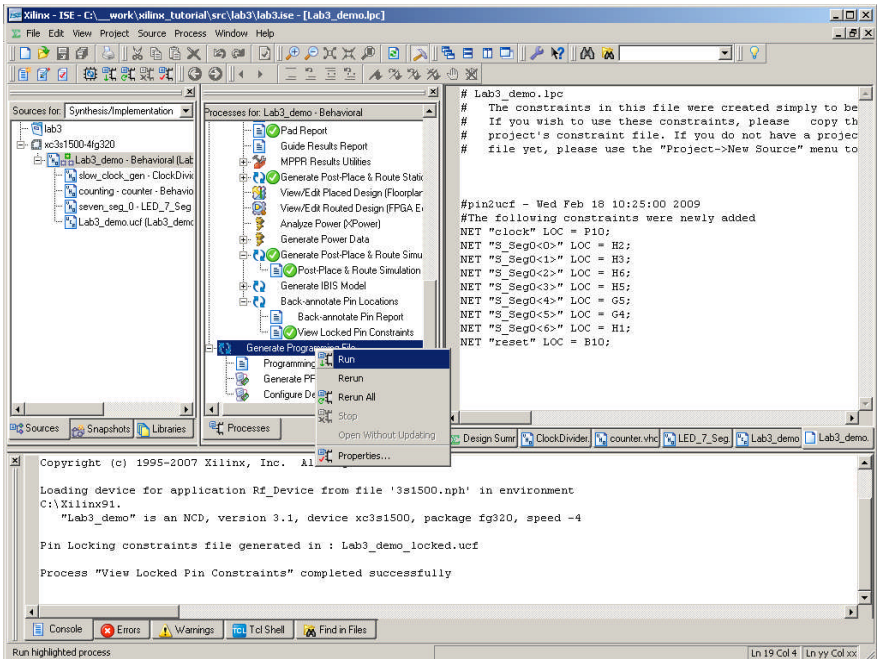
In the Place & Route Report you can find final information about the amount of resources utilized and the minimum clock period/maximum clock frequency of the design.



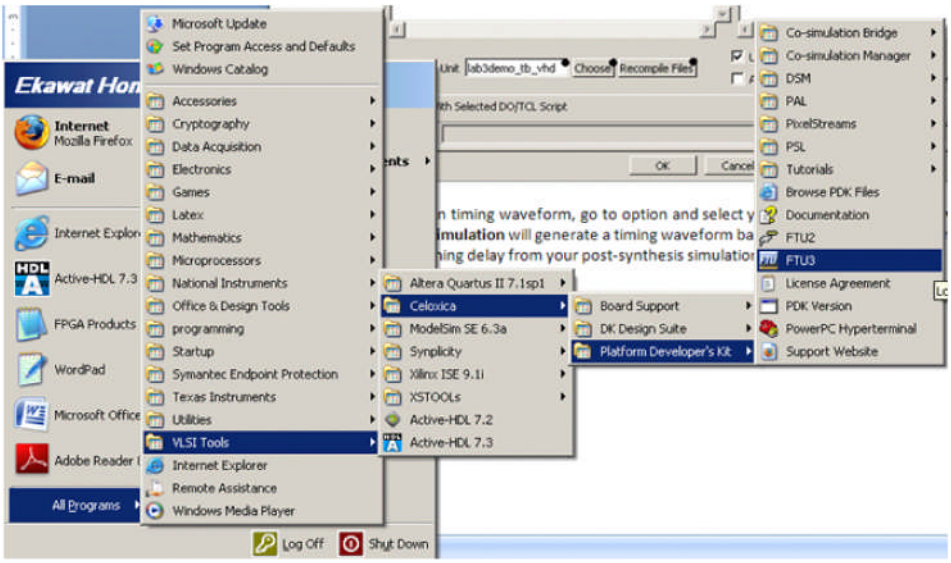


4.8 Bit Stream Generation

Go to menu **Sources for** and change this option to **Synthesis/Implementation.** Choose **Generate Programming File**, do right-click and pop-up menu should appear. Choose **Run** to start bit generation process.

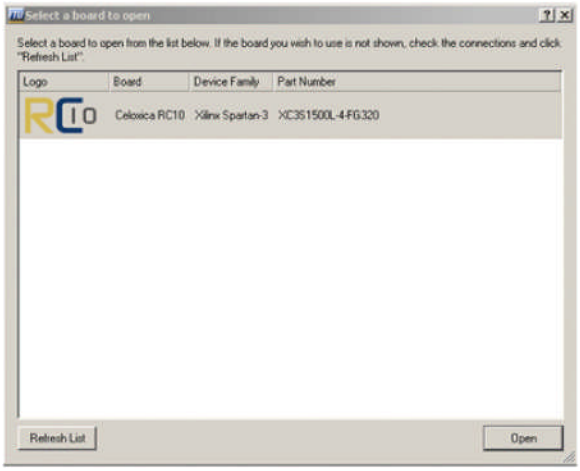


5. Uploading Bitstream to FPGA Board



Before uploading Bit file, make sure that you change your **constant values** in all your files to proper value, and re-synthesize/implement all the files. In particular, in our example, please change the value of the constant slow_clock_period in the Lab3Demo_package.vhd.

Select FTU3 program as shown in the picture above. When the program is opened, a device will be shown if it is connected and recognized. Select your FPGA and click **open.** Then, C**lear FPGA** and select the bit file located in **\implement\ver\rev1** of your workspace. Upload (**Configure)** the code and test your design whether it works correctly on the FPGA board.



Good luck! Have fun debugging =)