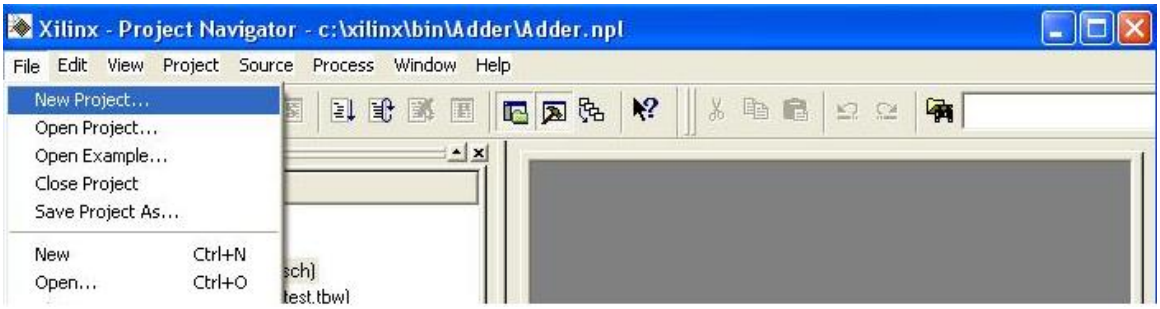
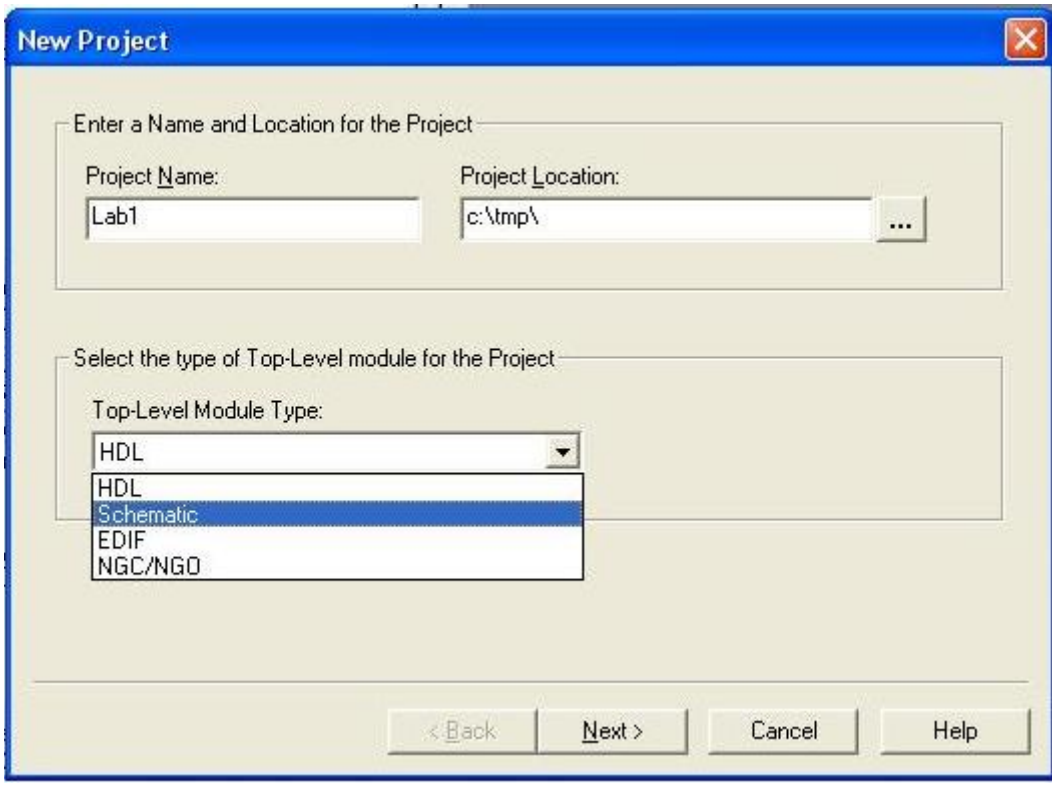
ECE 327 LAB
Introduction to XILINX Simulation

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 for this tutorial:

Select **File → New Project**.

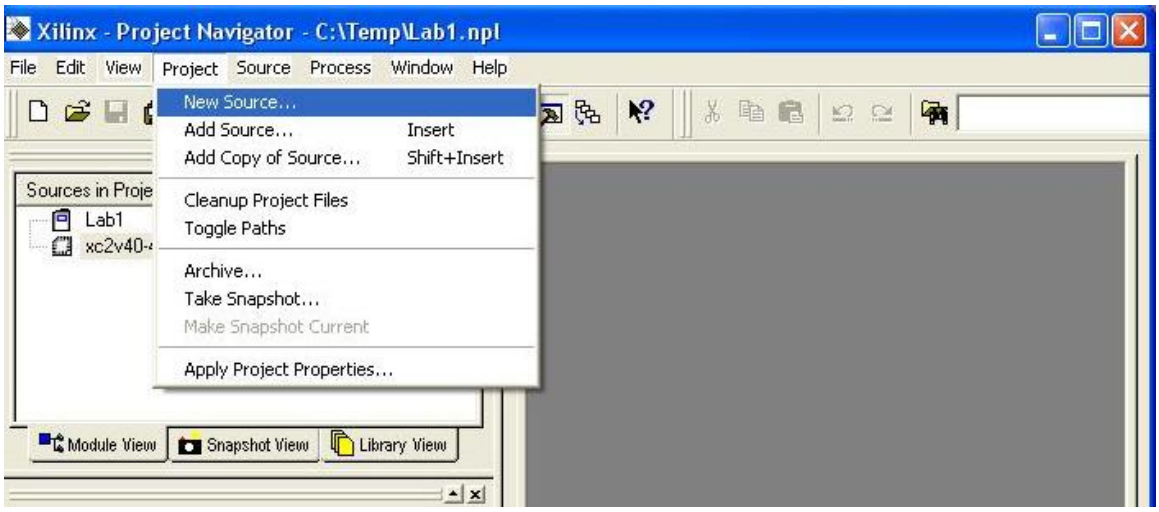


In the **New Project** Wizard dialog box, give a Project Name as **Lab1** for Lab1. Type C:\tmp in the Project Location field. Select Schematic from the drop-down menu of Top-Level Module Type field. A schematic is a drawing that represents all or part of an electronic circuit

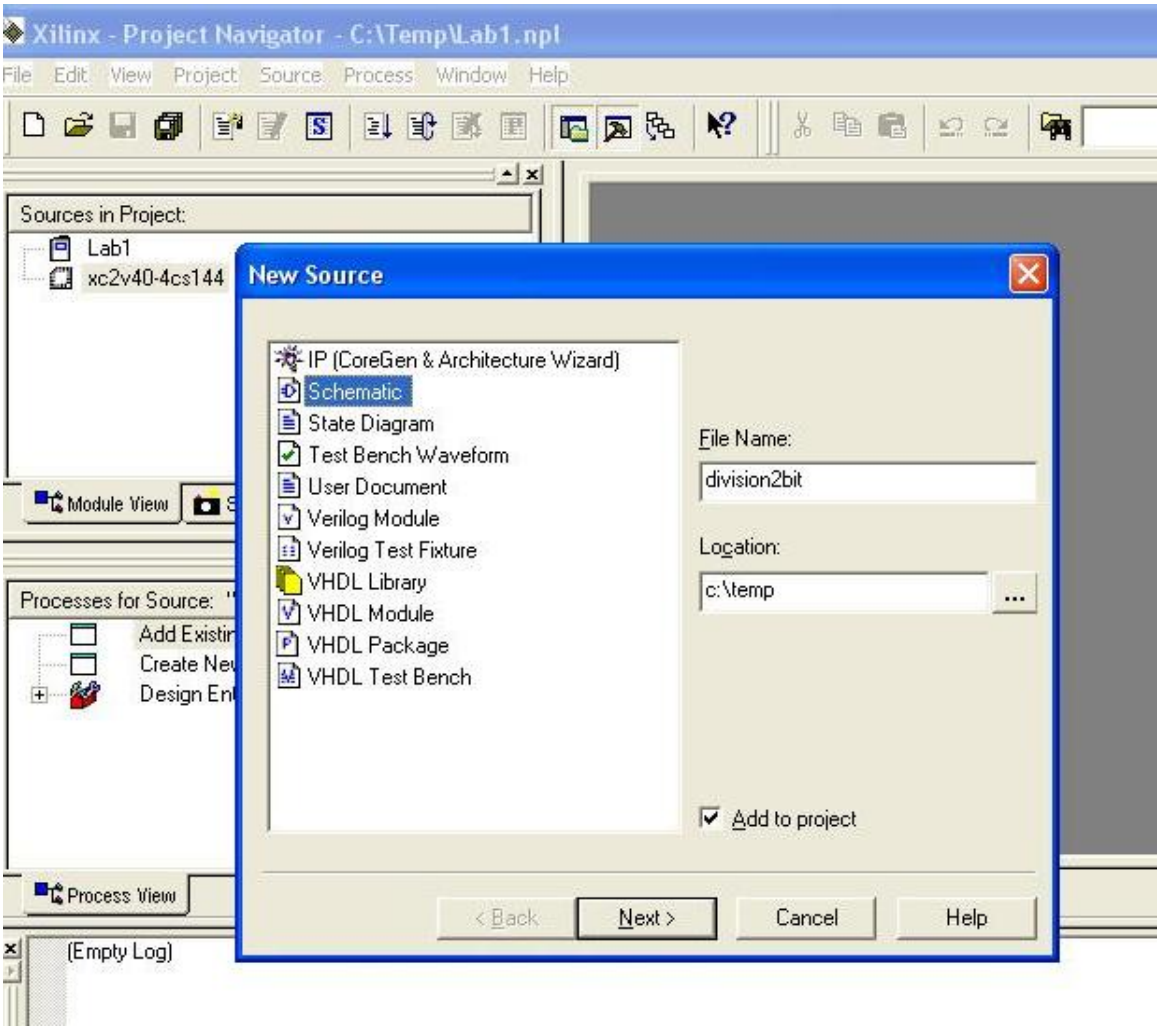


Click **Next->Next**.. **Finish** .

Select **Project -> New Source**



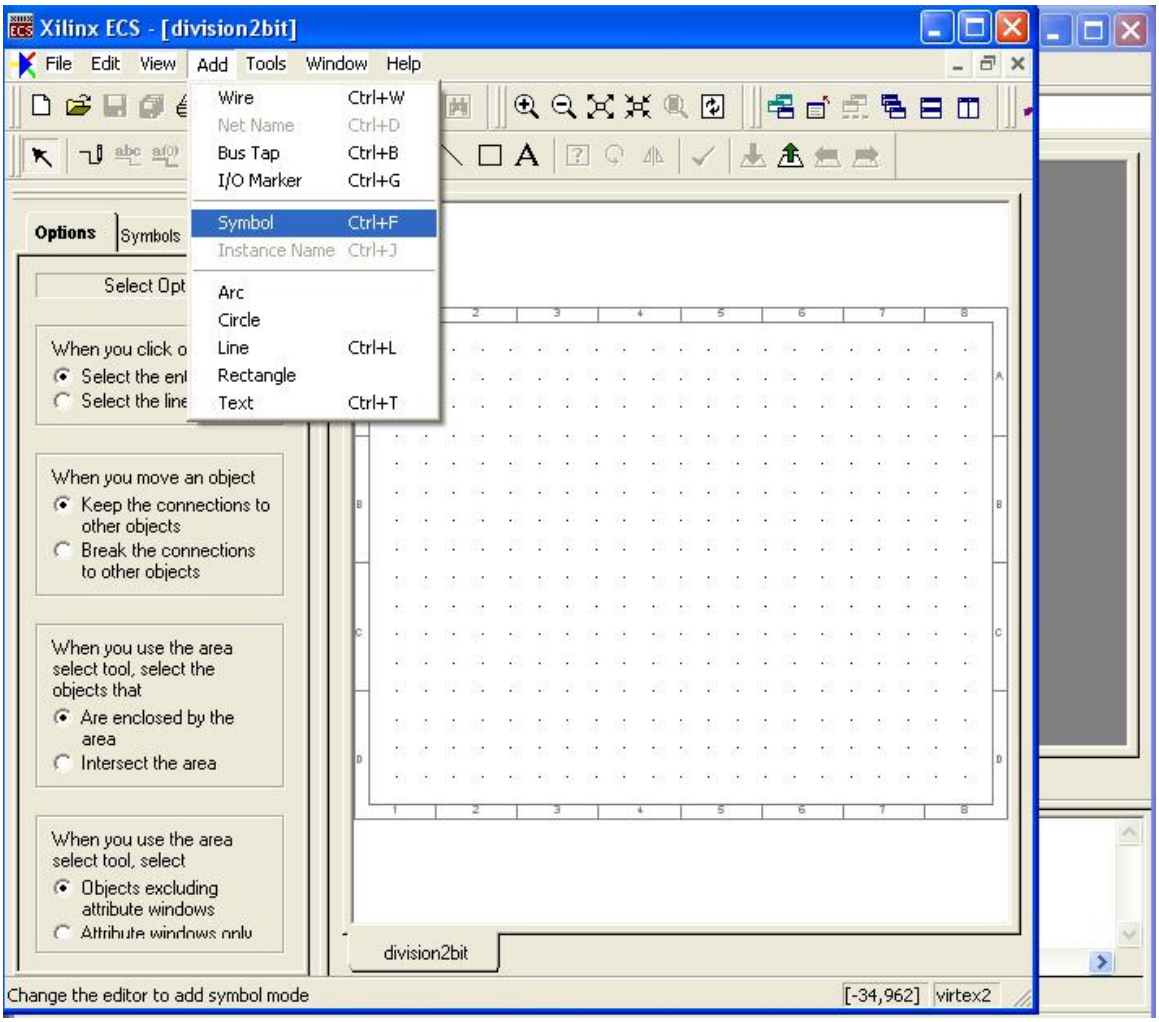
In the **New Source** wizard box select Schematic as shown. Then give a file name to your schematic and also the location in **Filename** and **Location** respectively.



Click **Next .. Finish.**

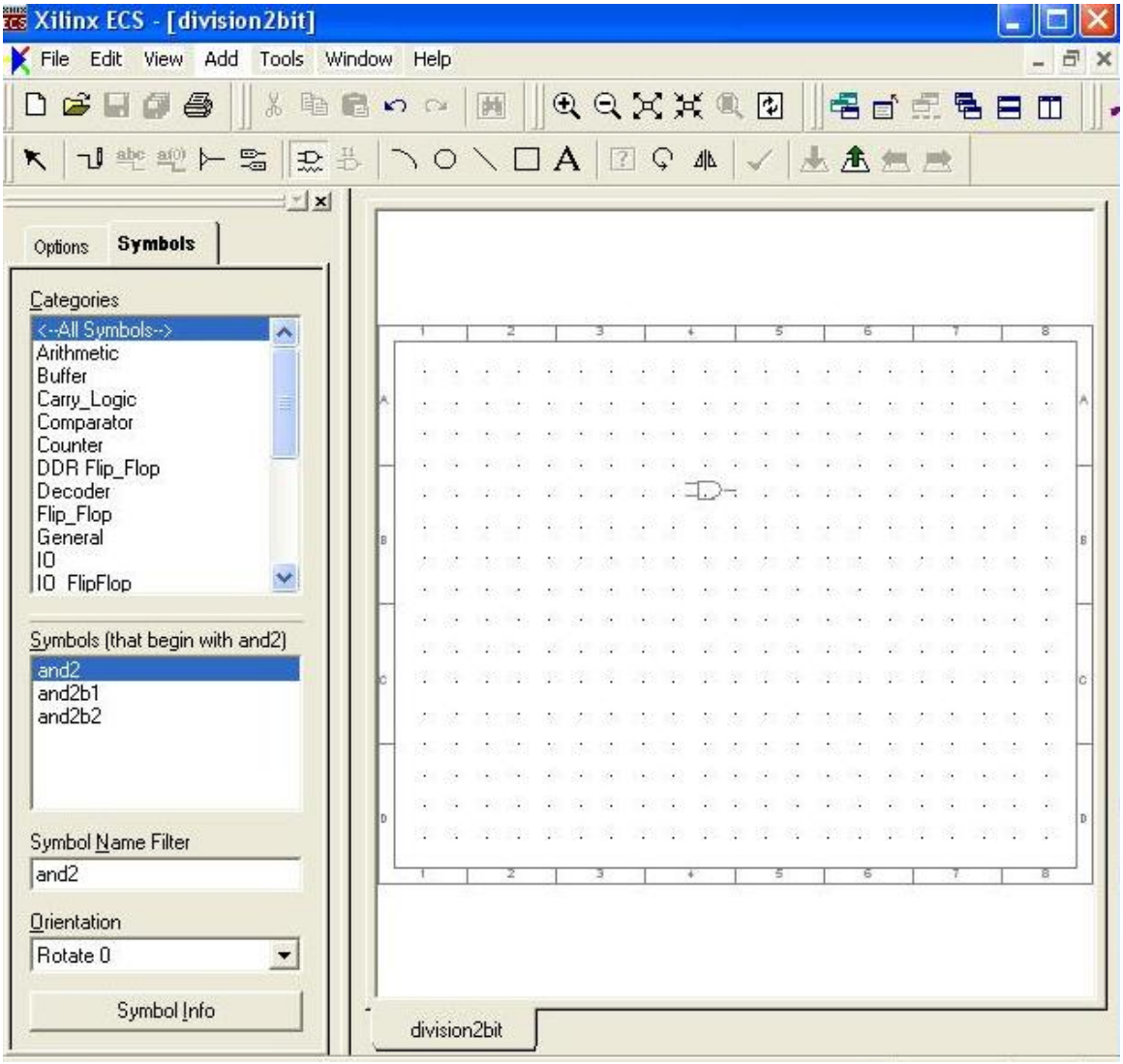
This will open Engineering Capture System editor. **The Engineering Capture System (ECS)** is a graphical user interface (GUI) that allows you to create, view, and edit schematics and symbols for the Design Entry step of the Xilinx® design flow. The Schematic Editor mode of ECS allows you to open and edit schematic (SCH) files.

Click **Add -> Symbol**.

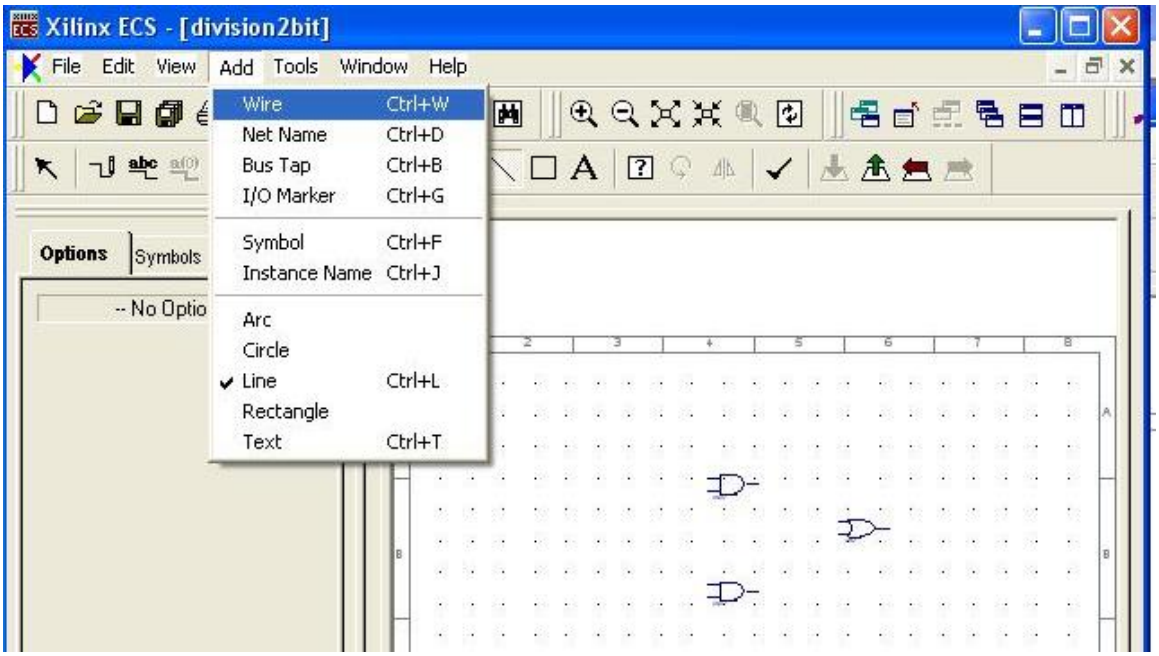


This wll open the Symbol Editor.The Symbol Editor mode of ECS allows you to open and edit symbol (SYM) files. Symbols are the basic elements of a schematic.

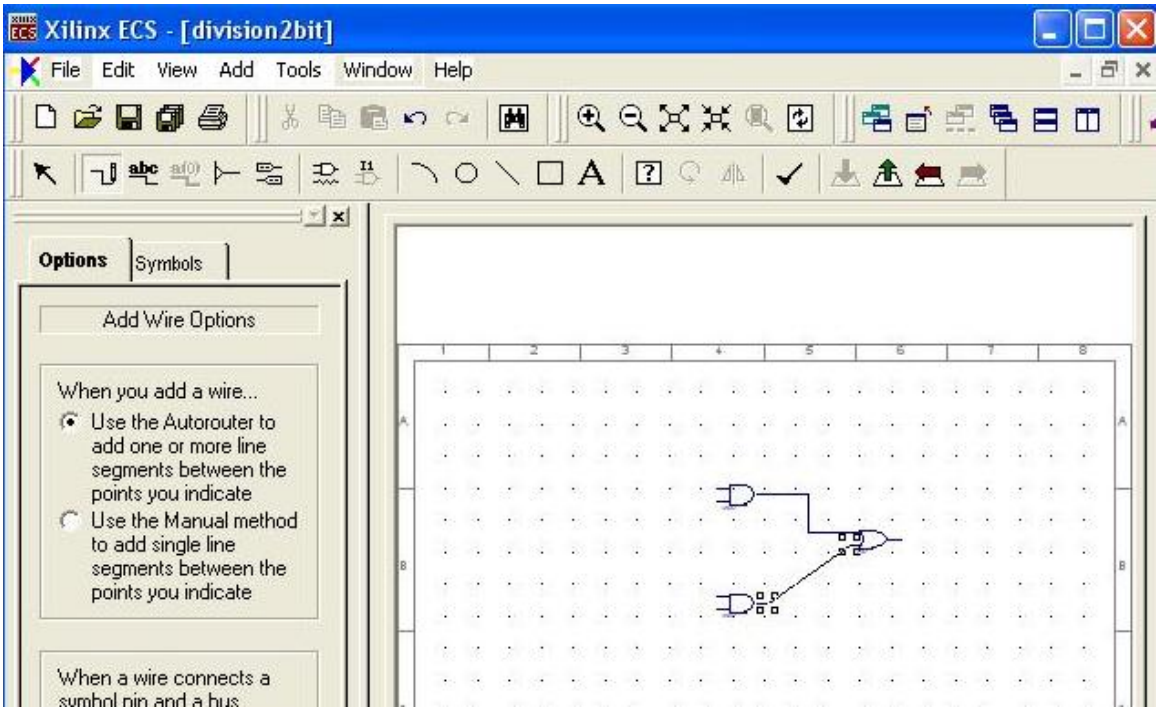
To place a 2-input AND gate, type **and2** in the **Symbol Name Filter** field. It gives all the gates whose name starting with **and2.** Select **and2** and place it on the editor(White Space) as shown below. Select required gates according to your circuit.



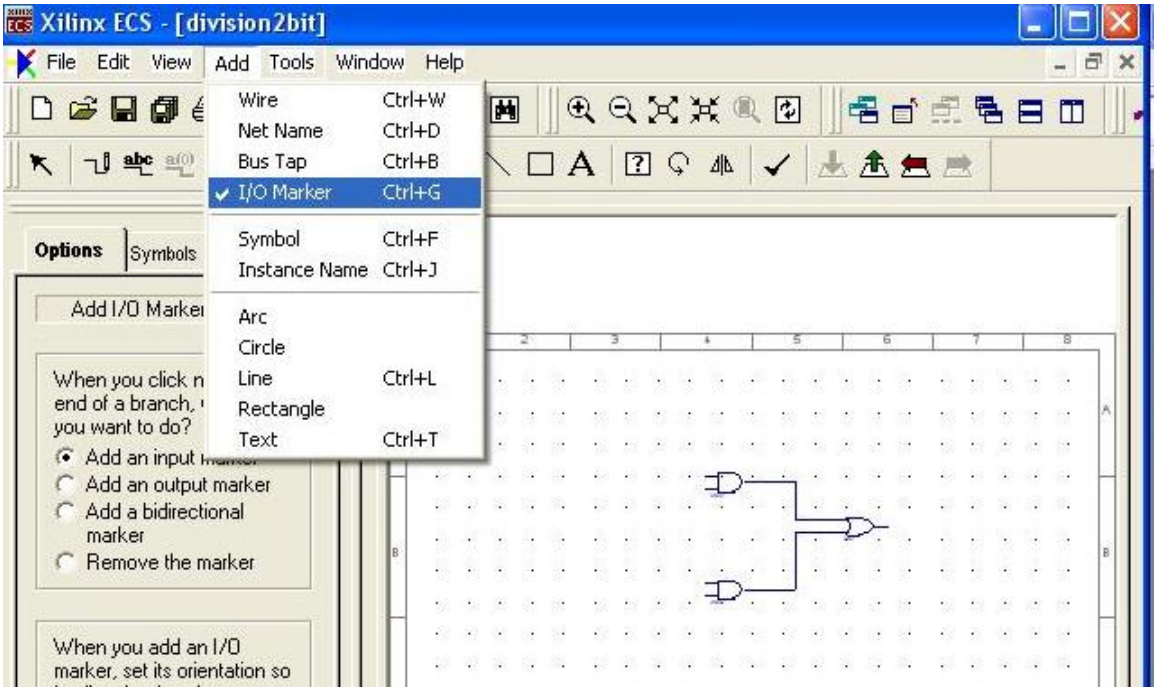
After placing the gates, you have to connect the gates. You can do this using **wire.**
Click **Add->Wire.**

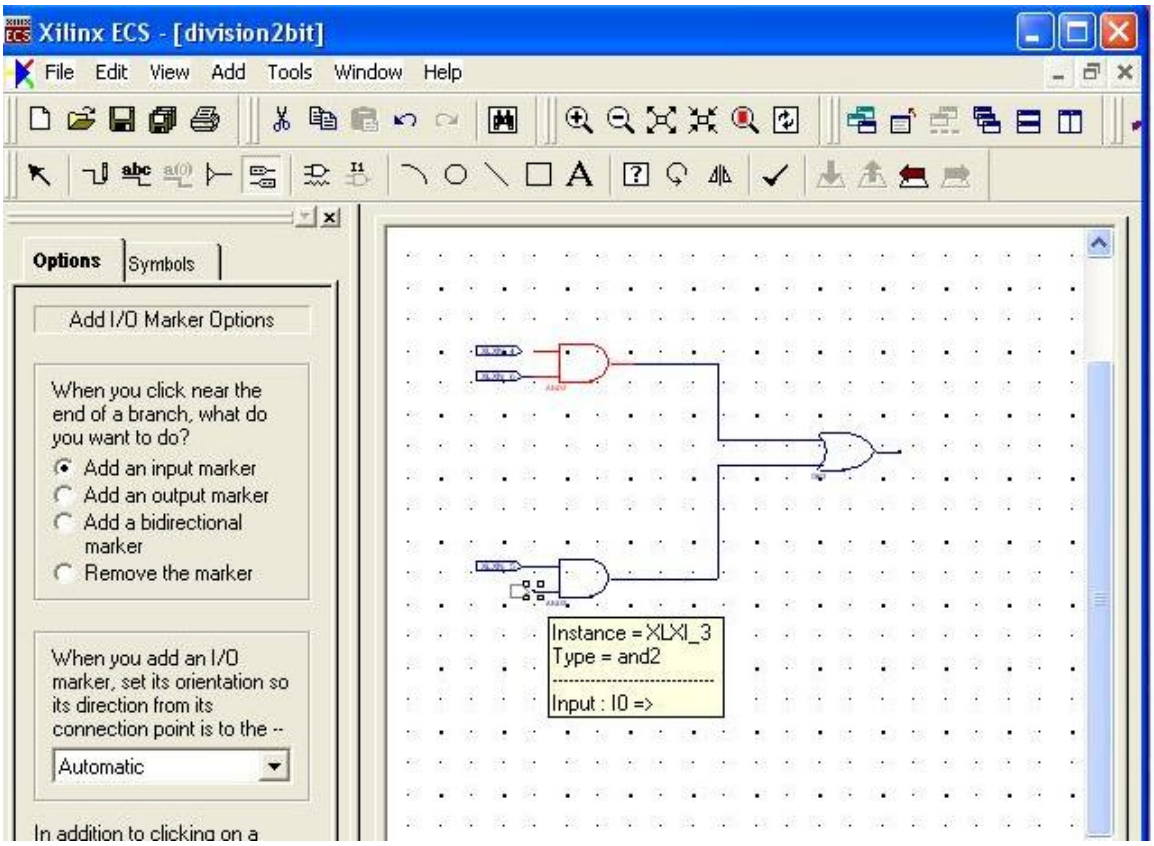


And connect the terminals of the gates as shown.

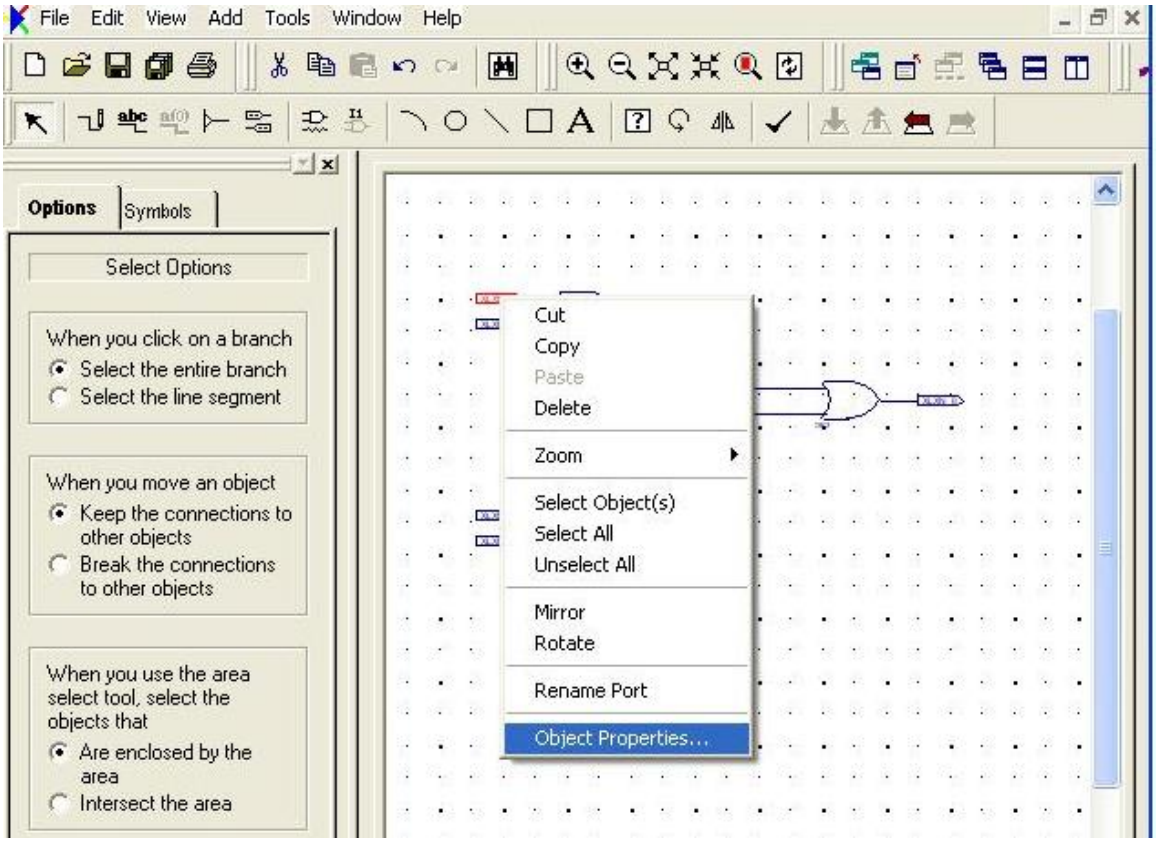


To apply the signals, you must have pins at the terminals of gates. You can place pins by clicking **Add->I/O Marker** as shown.

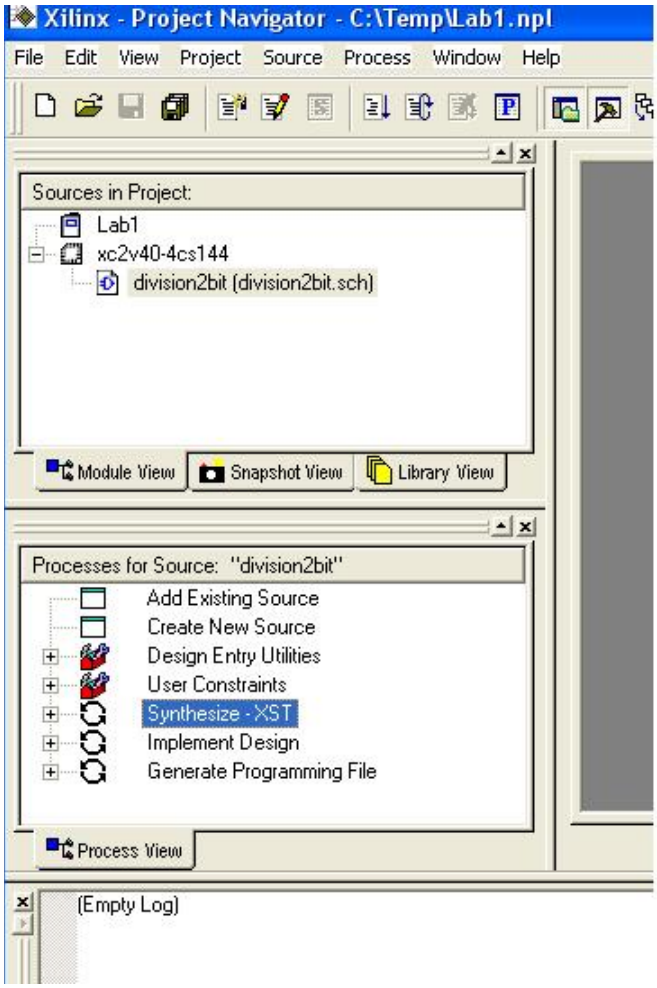




Change the name of pins by right-clicking pin and select **Object properties.** Change the Name field to required name. If it is a **A** input change name to **A.**

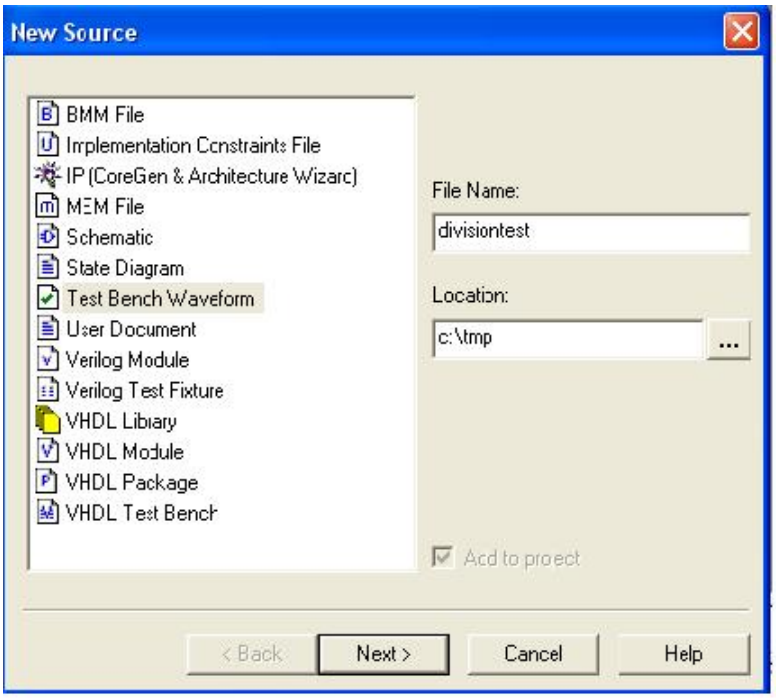


Then in main window(Project Navigator) **double-click** the **Synthesize-XST** as shown.

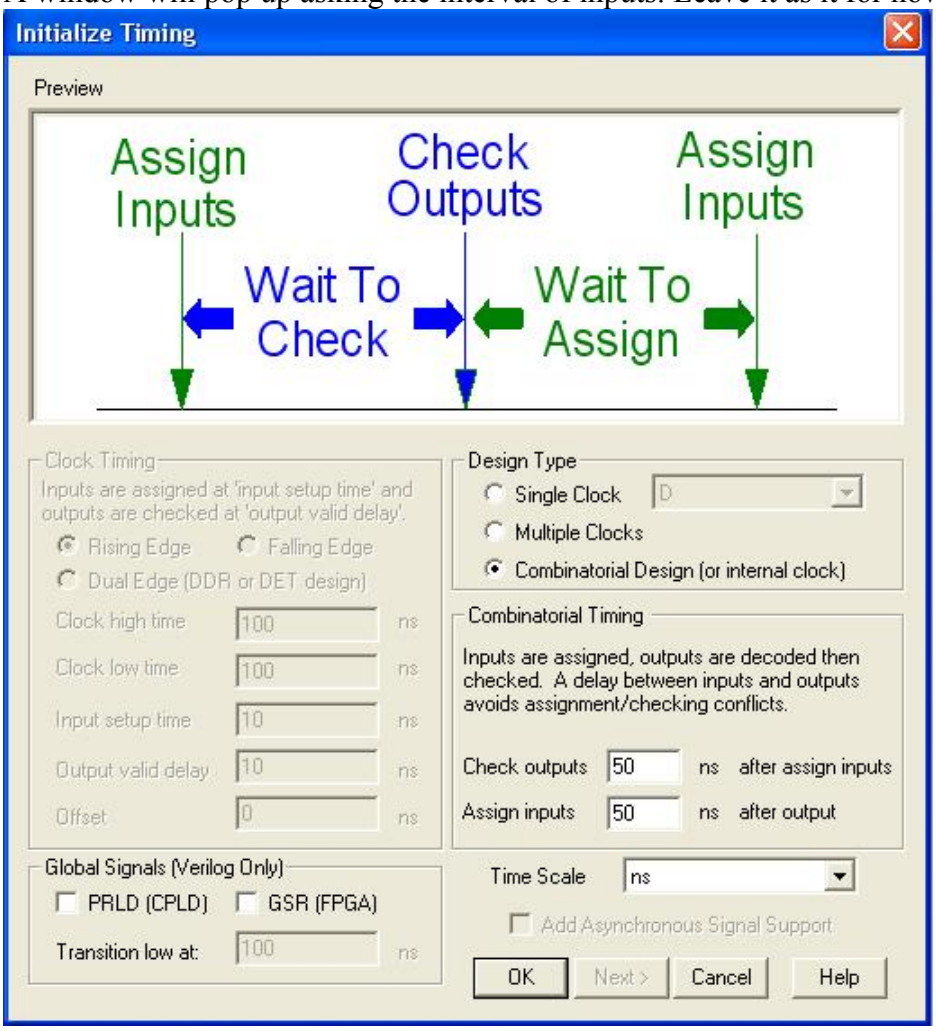


If everything is correct, go to **Add->New Source.** In the **New Source** wizard box click **Test Bench Waveform.** After synthesizing, you have to apply the inputs. You can do this using a special file called Test Bench file. In this file you mention the required inputs to your circuit.

Give a name to your test bench file and also location as earlier.



A window will pop up asking the interval of inputs. Leave it as it for now.



Click **OK.**

Then select requird inputs as shown. Then save your file.
Then click **Generate Expected Simulation Results** on left hand side as shown.