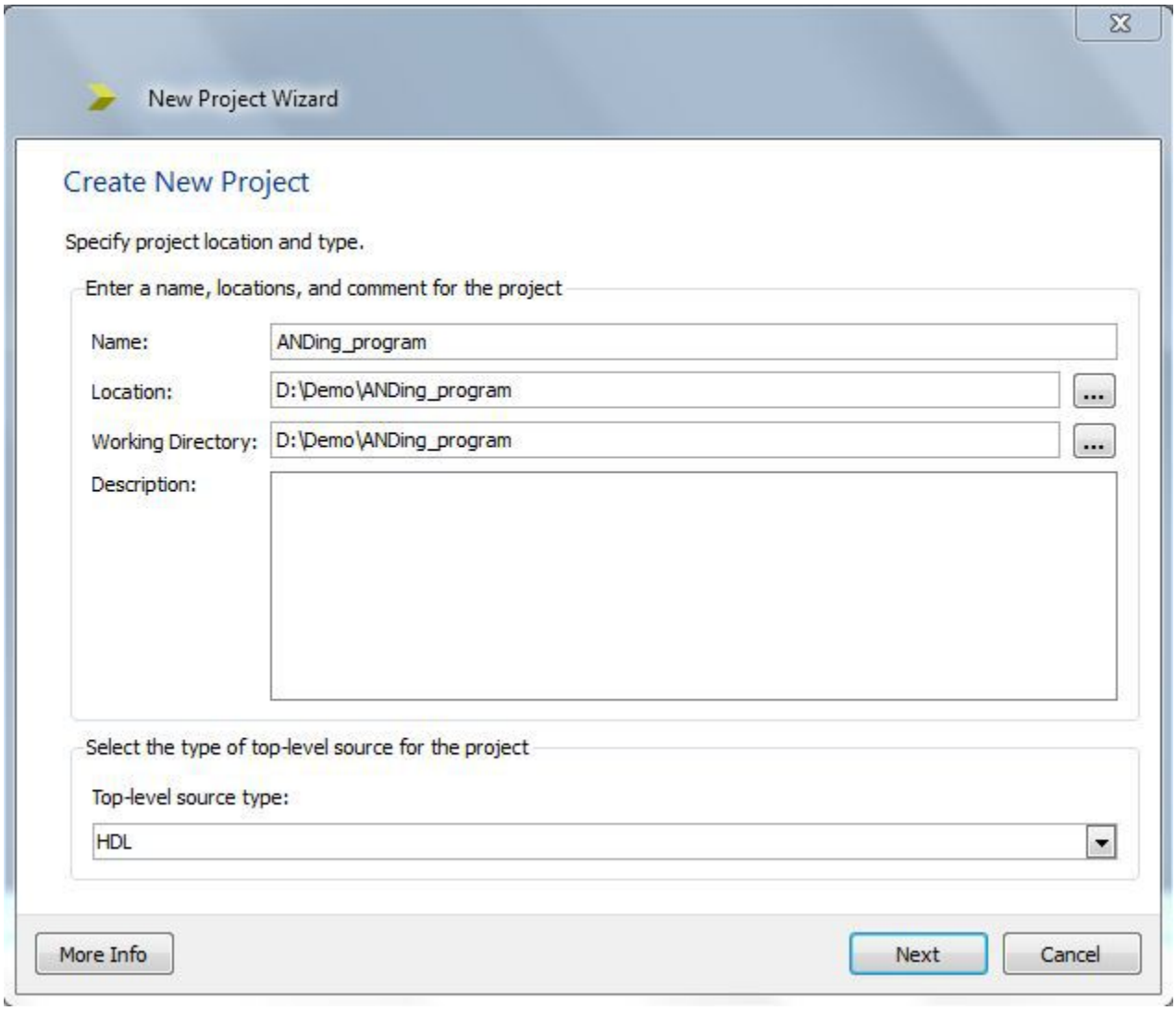
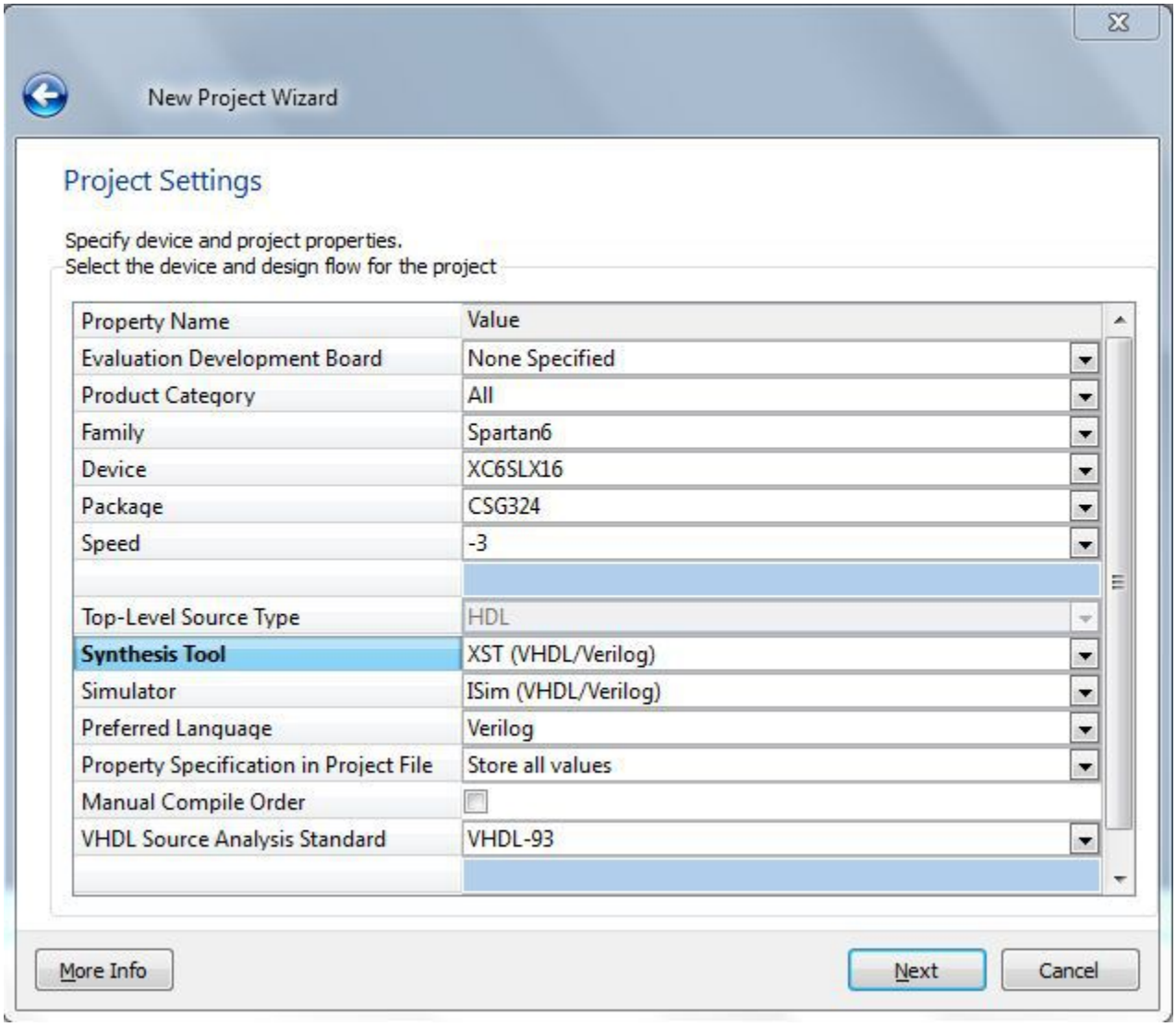
STEP1: Open Xilinx ISE and create a new project.



Select path of your working project directory and enter name of project.

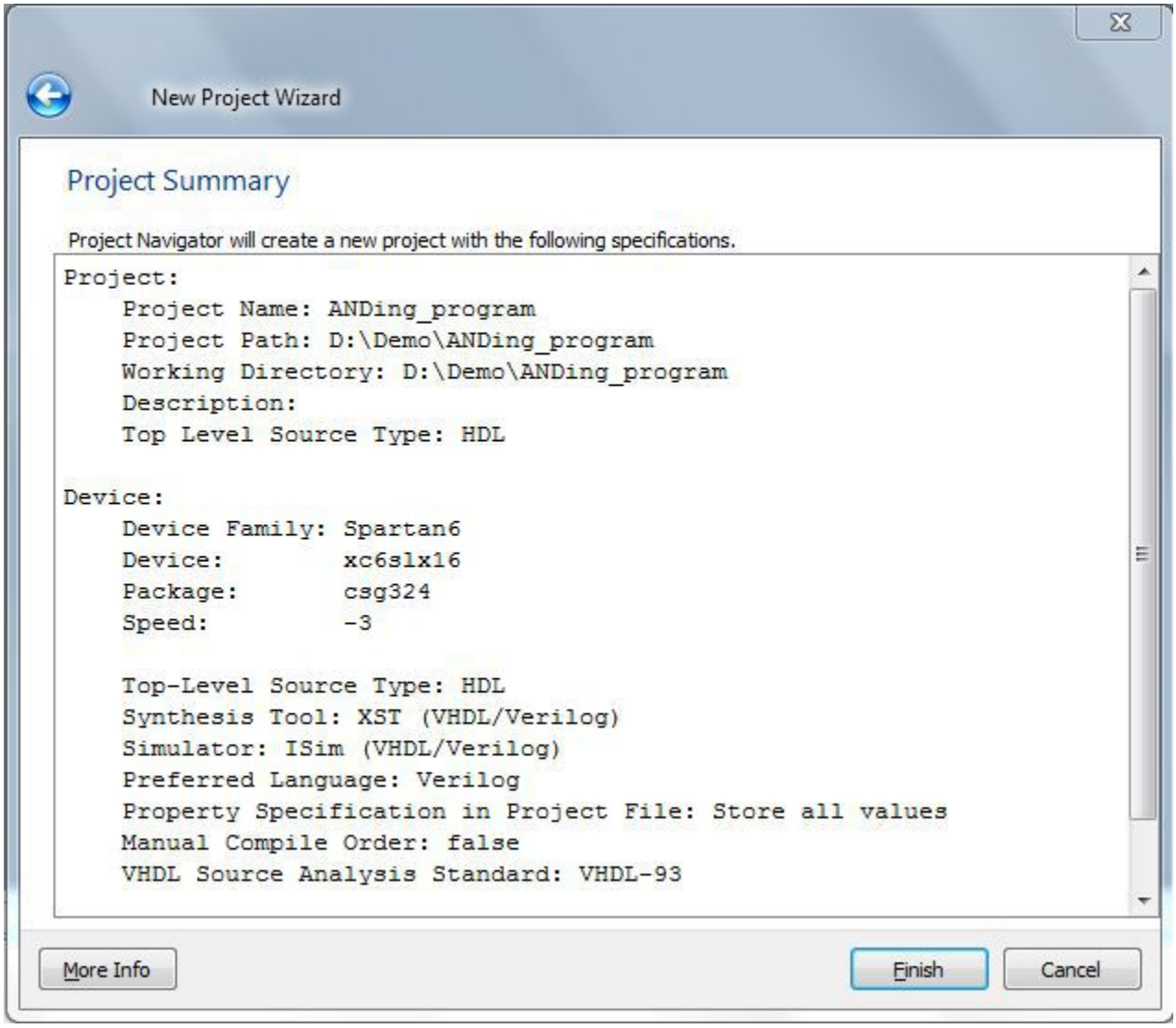
click on **Next**

Step 2: Select the Family, Device, Package and Speed of Xilinx board and also select your programming language(Verilog/VHDL). Here i am using Verilog language.



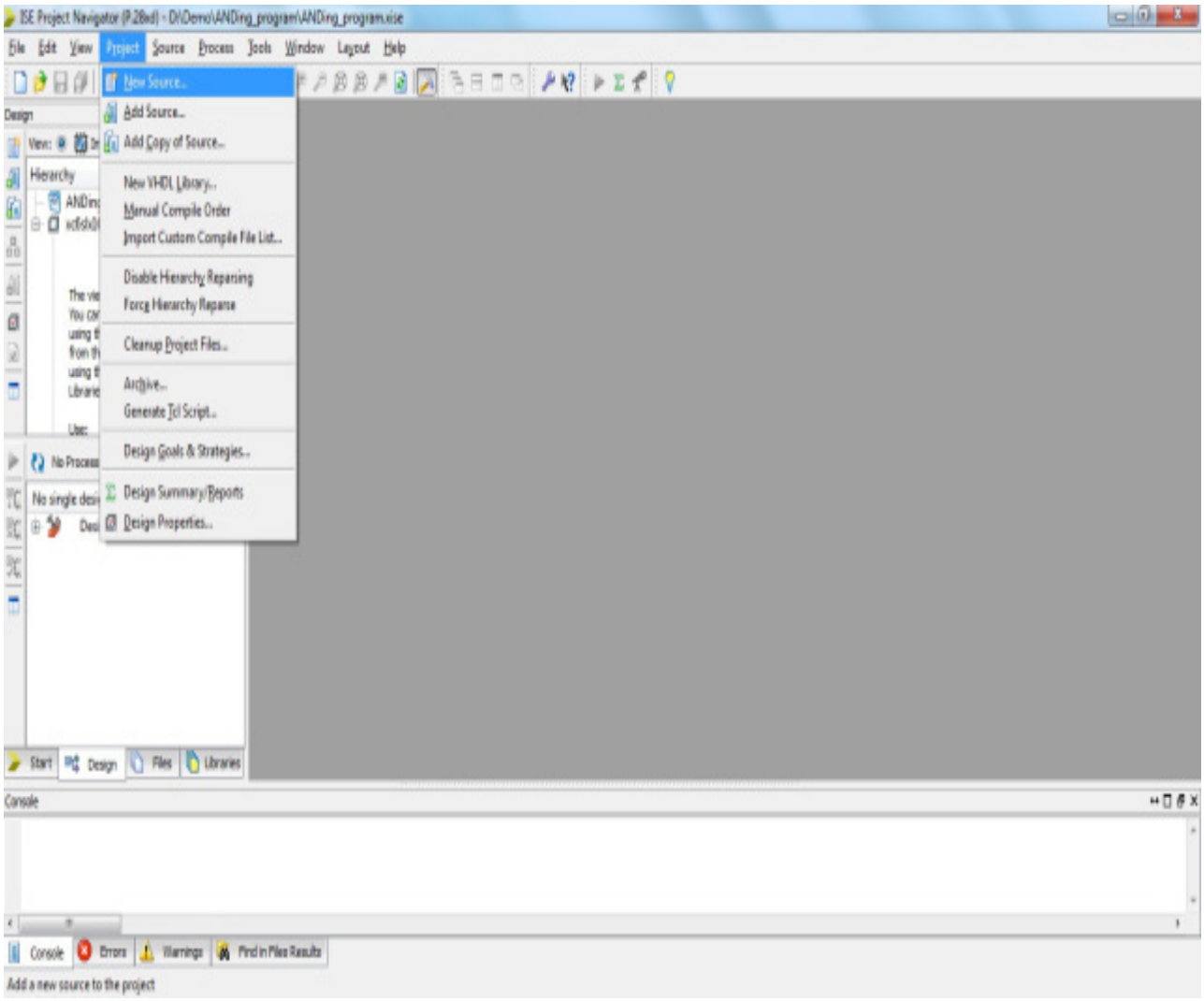
click on **Next**

Project summery window occurs.

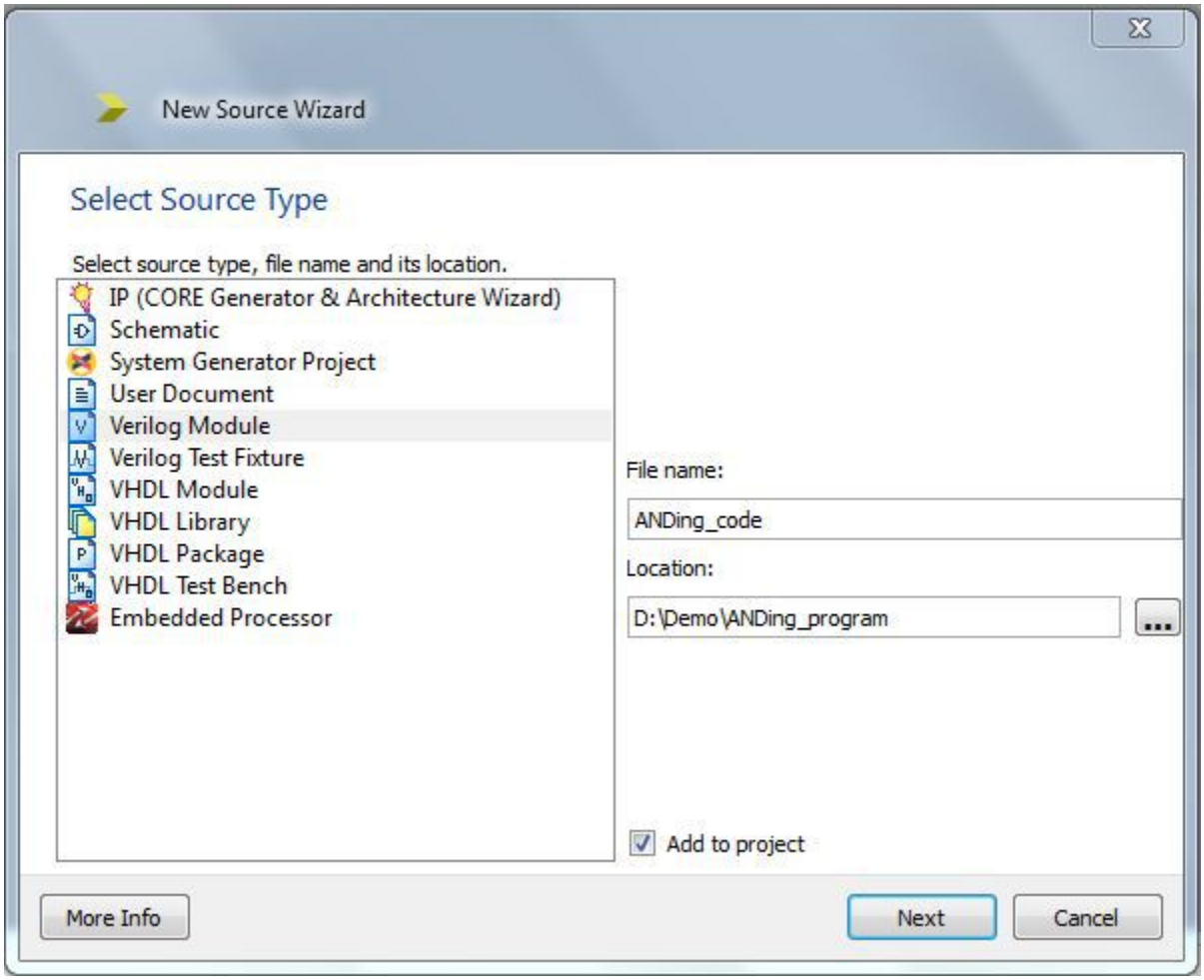


Click on **Finish**

Step 3: Click on **Project > New Source**



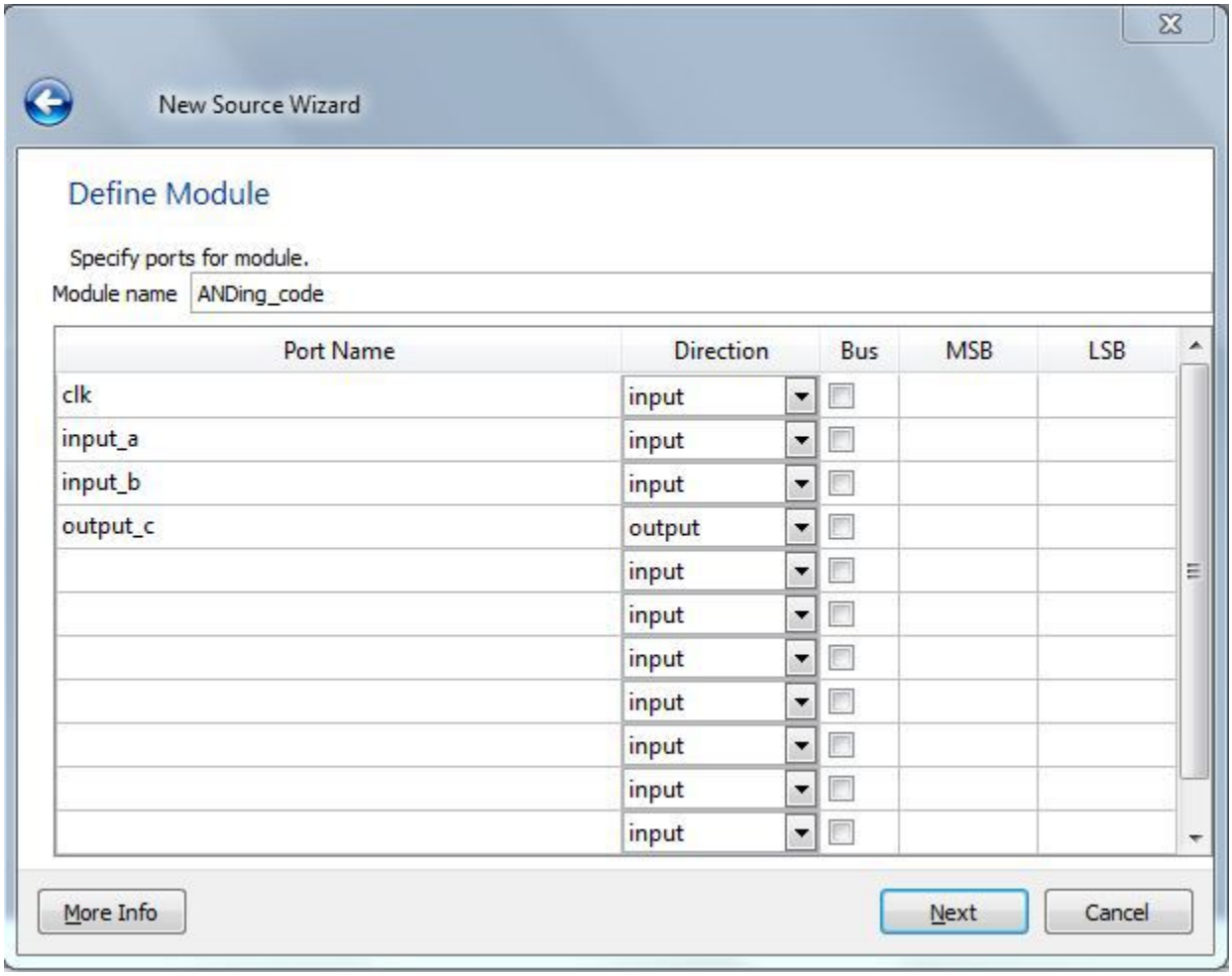
Step 4: Select Source type is **Verilog Module** and enter the file name (**ANDing_code**).



Click on **Next**

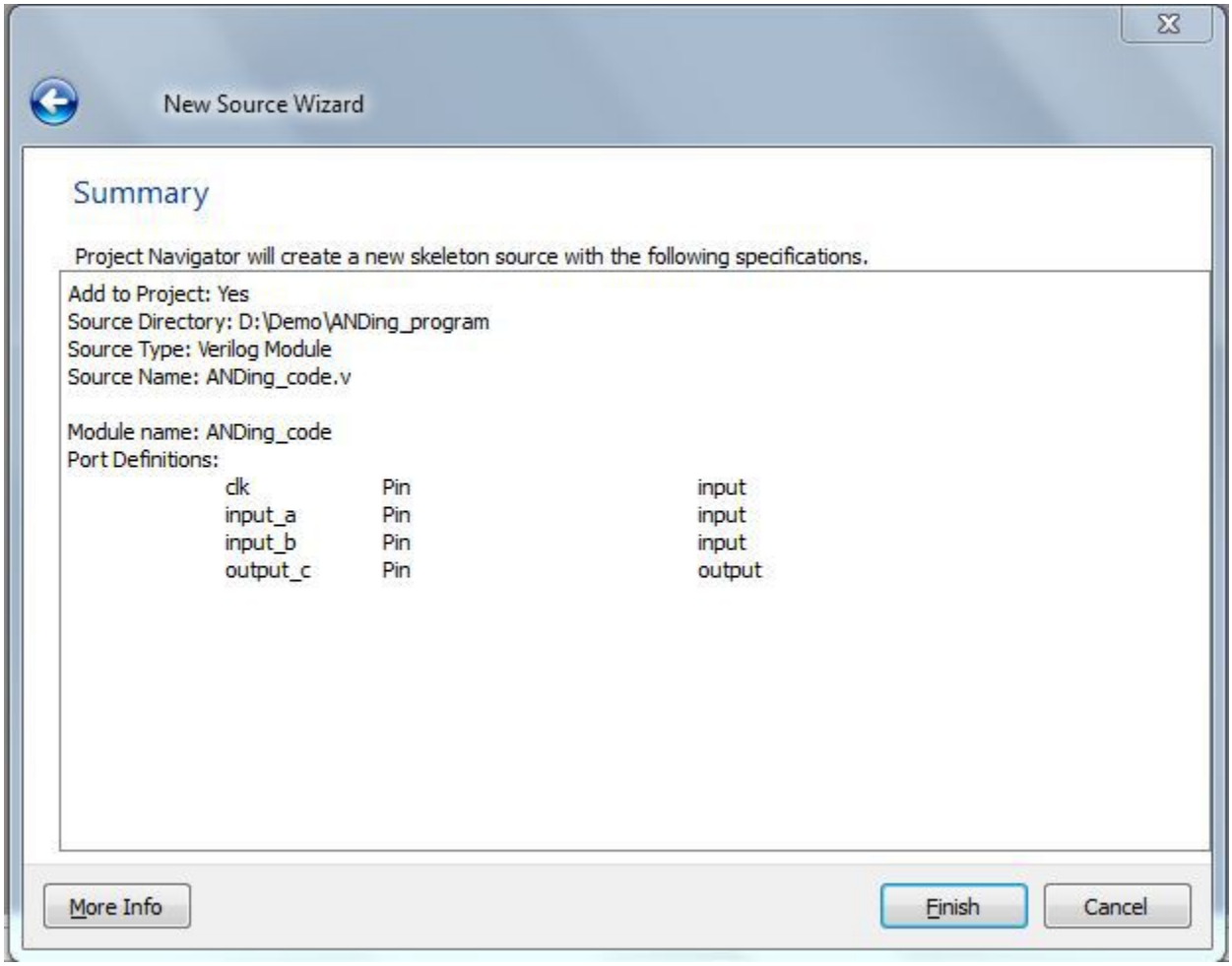
Step 5: Define Module Window.

Here we can define inputs & outputs and its bit/bus size. In ANDing example there are two inputs and output of single bit each. Also define clock signal for clocking operation.



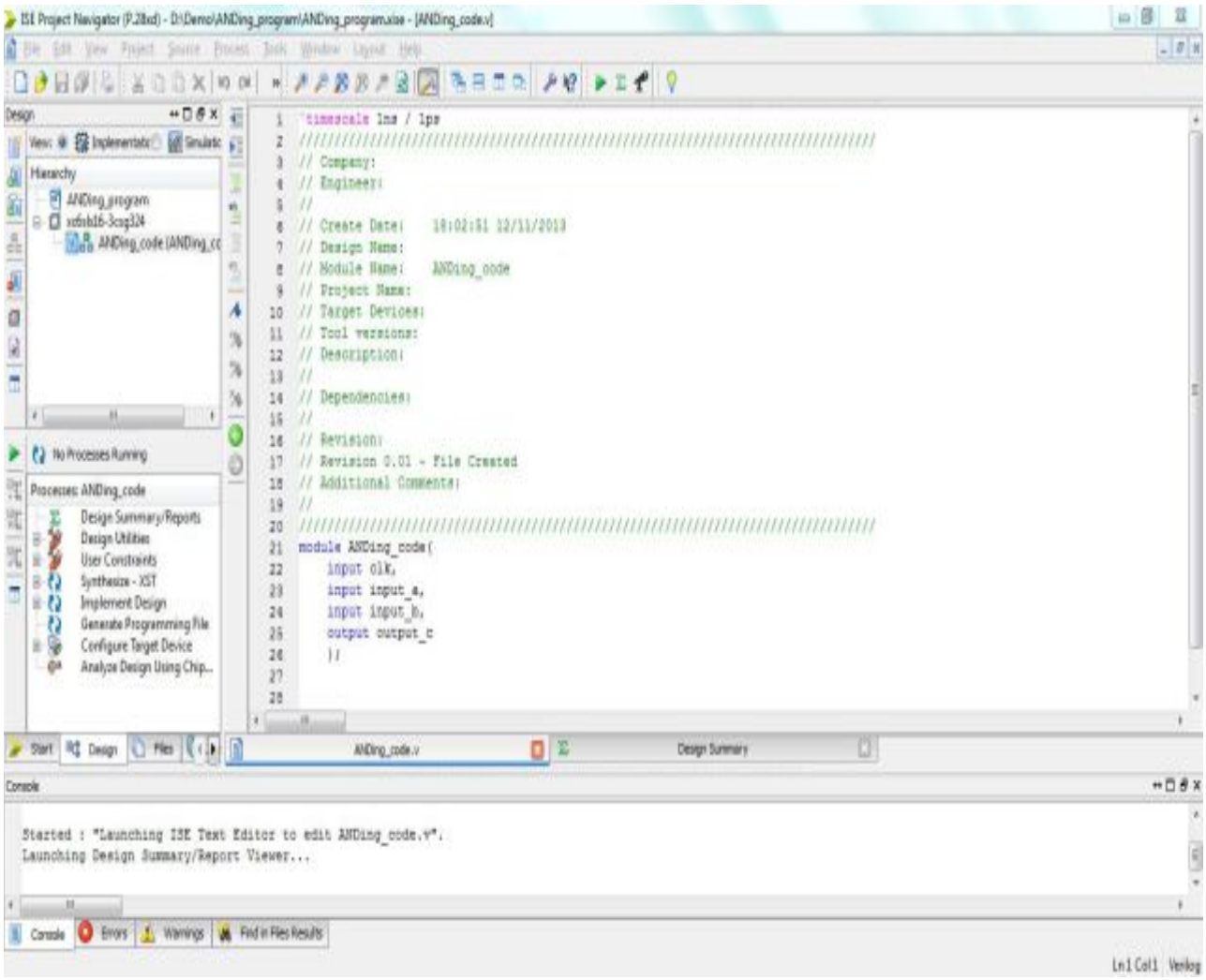
Click on **Next.**

Step 6: Summery Window



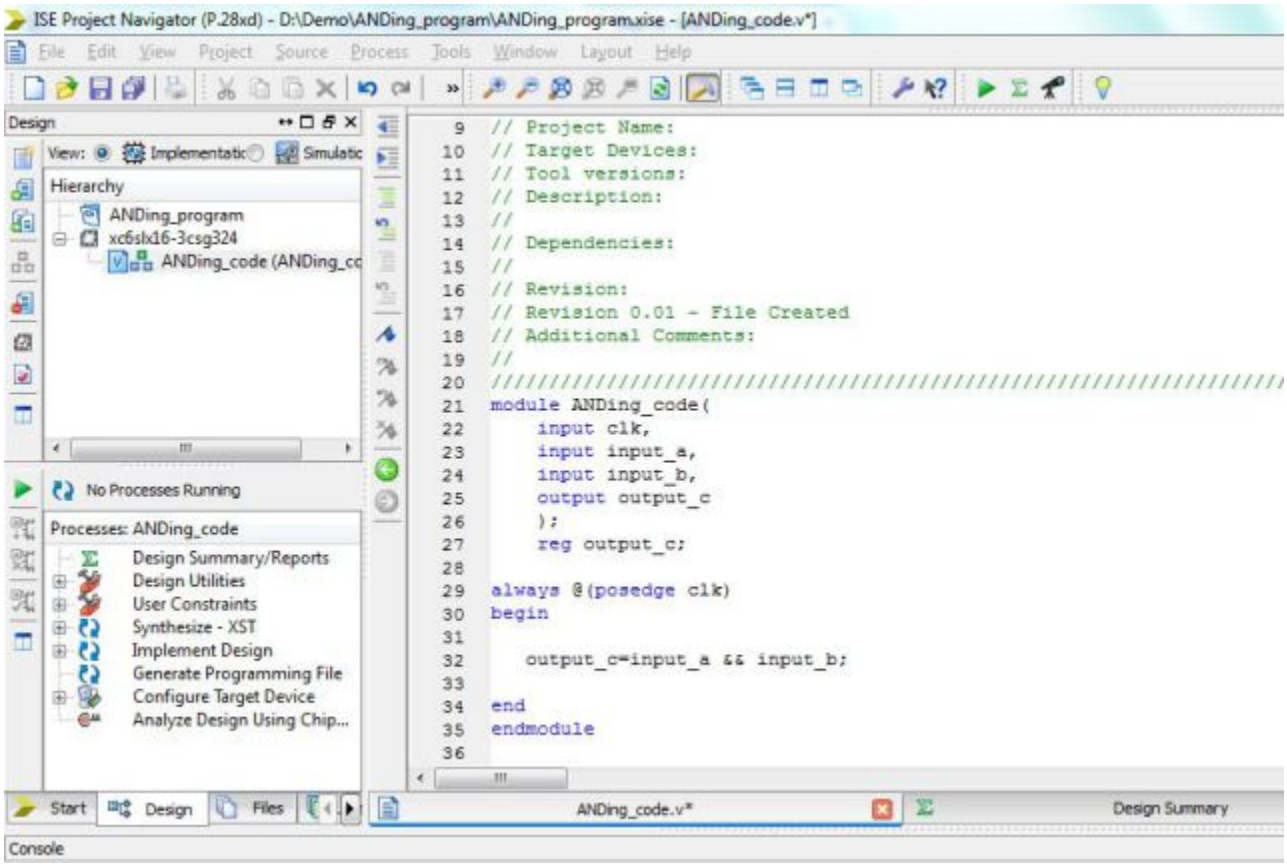
Click on **Finish.**

Step 7: The Project Navigator window looks like below window.

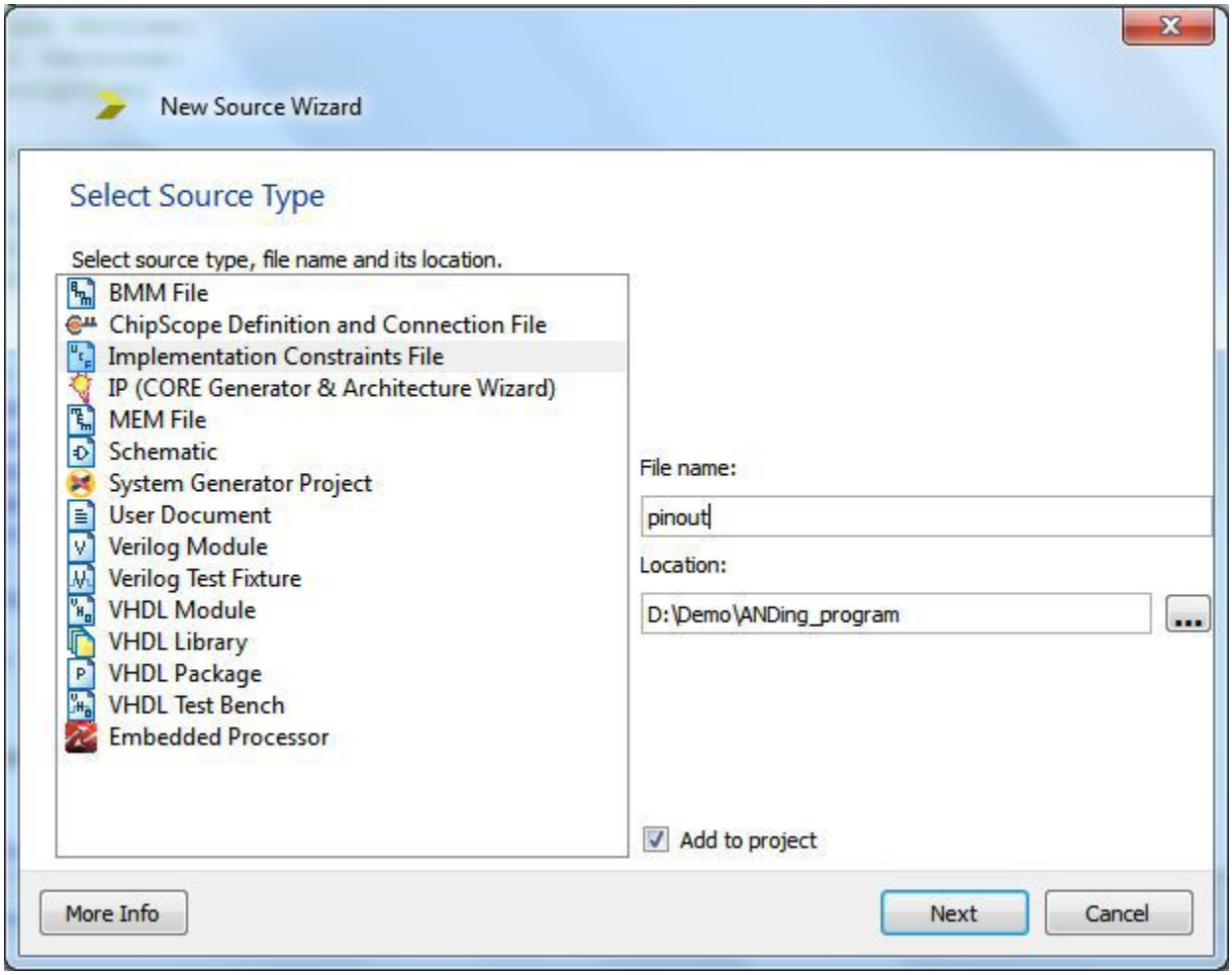


All the inputs and outputs are already defined in Define Module window so these inputs and outputs are seen in project navigator.

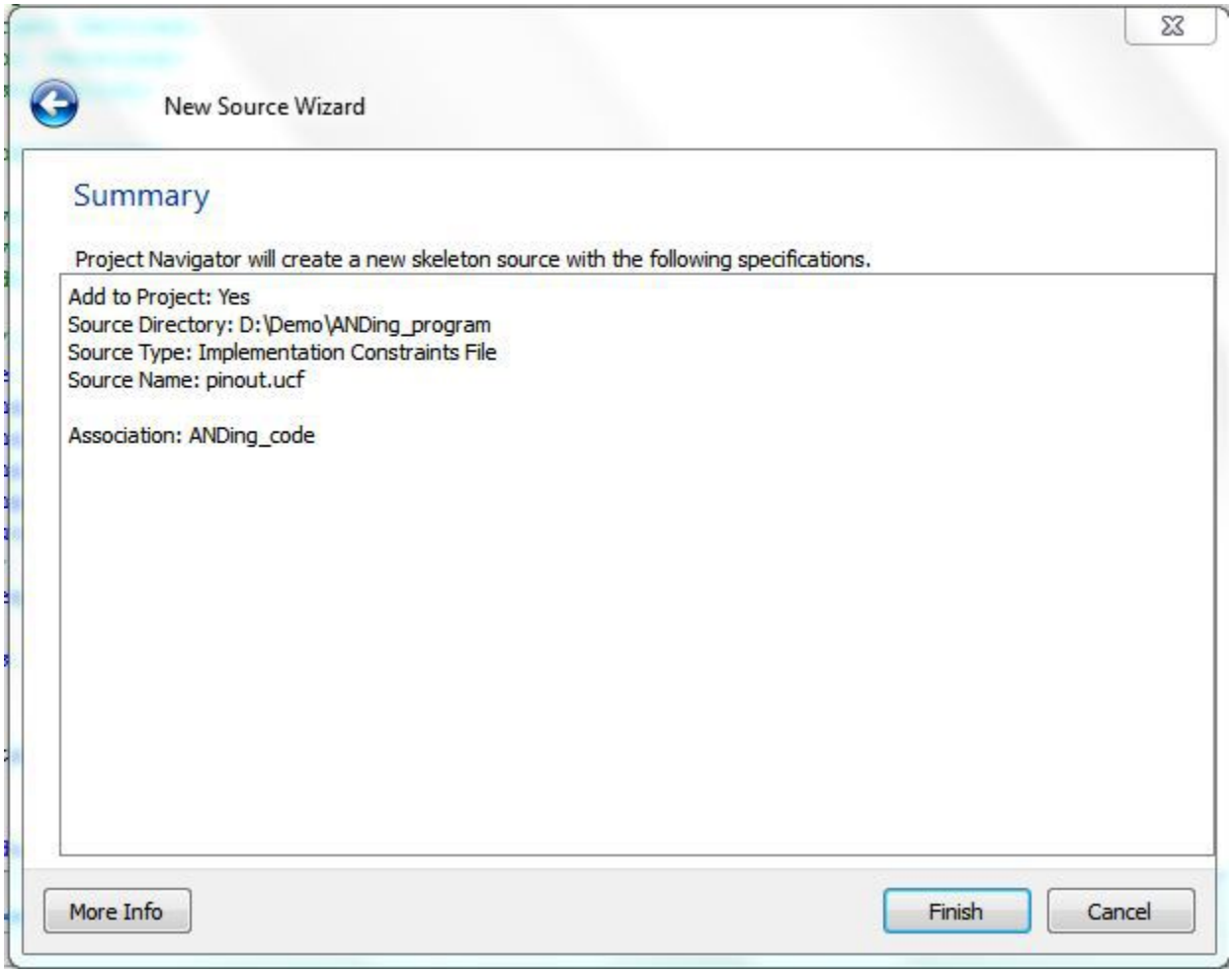
Step 8: Write a program for ANDing operator in module present in project navigator.



Step 9: Click on Project > New Source. Select Implementation Constraints file type and enter the file name (e.g. pinout).

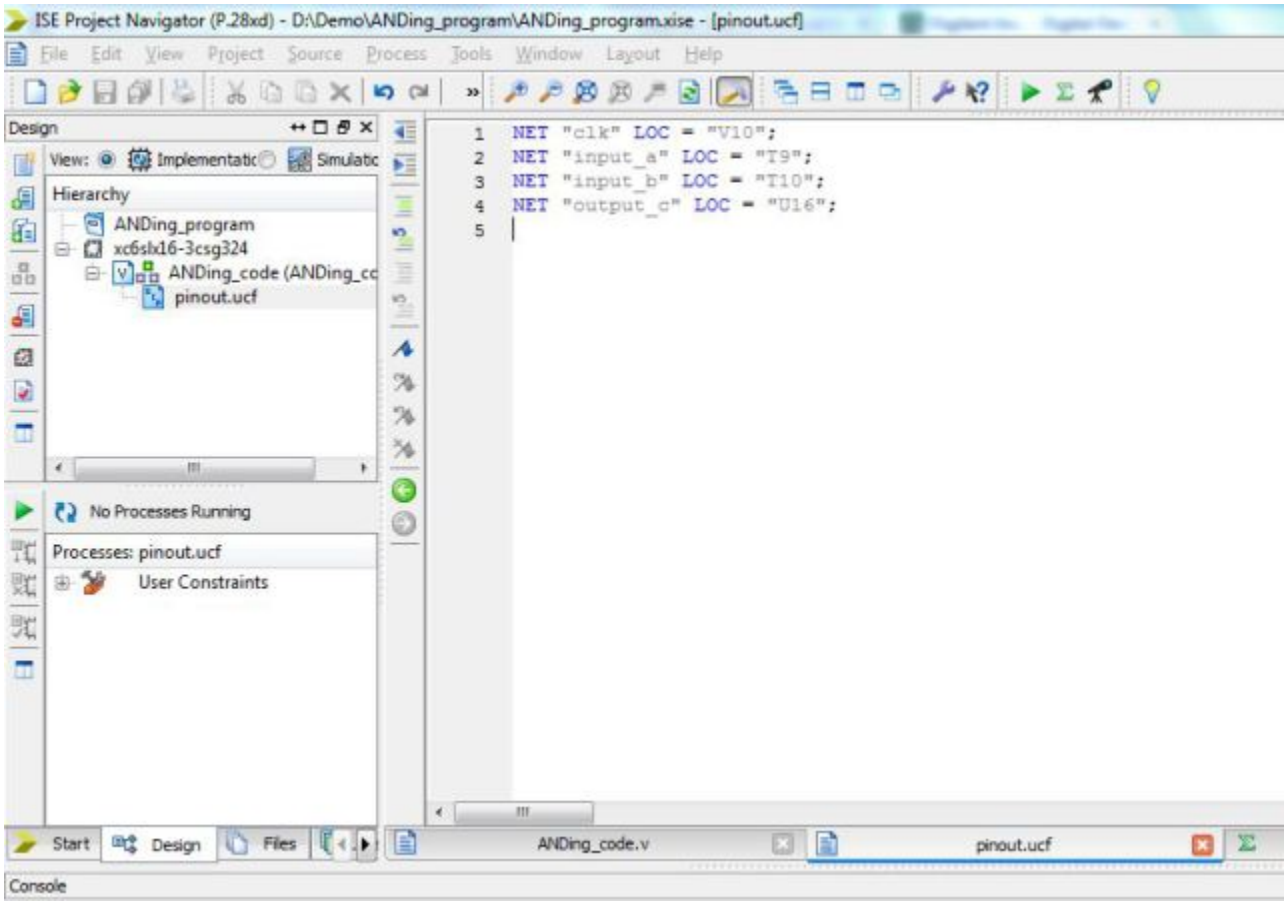


Click on Next.



Click on Finish.

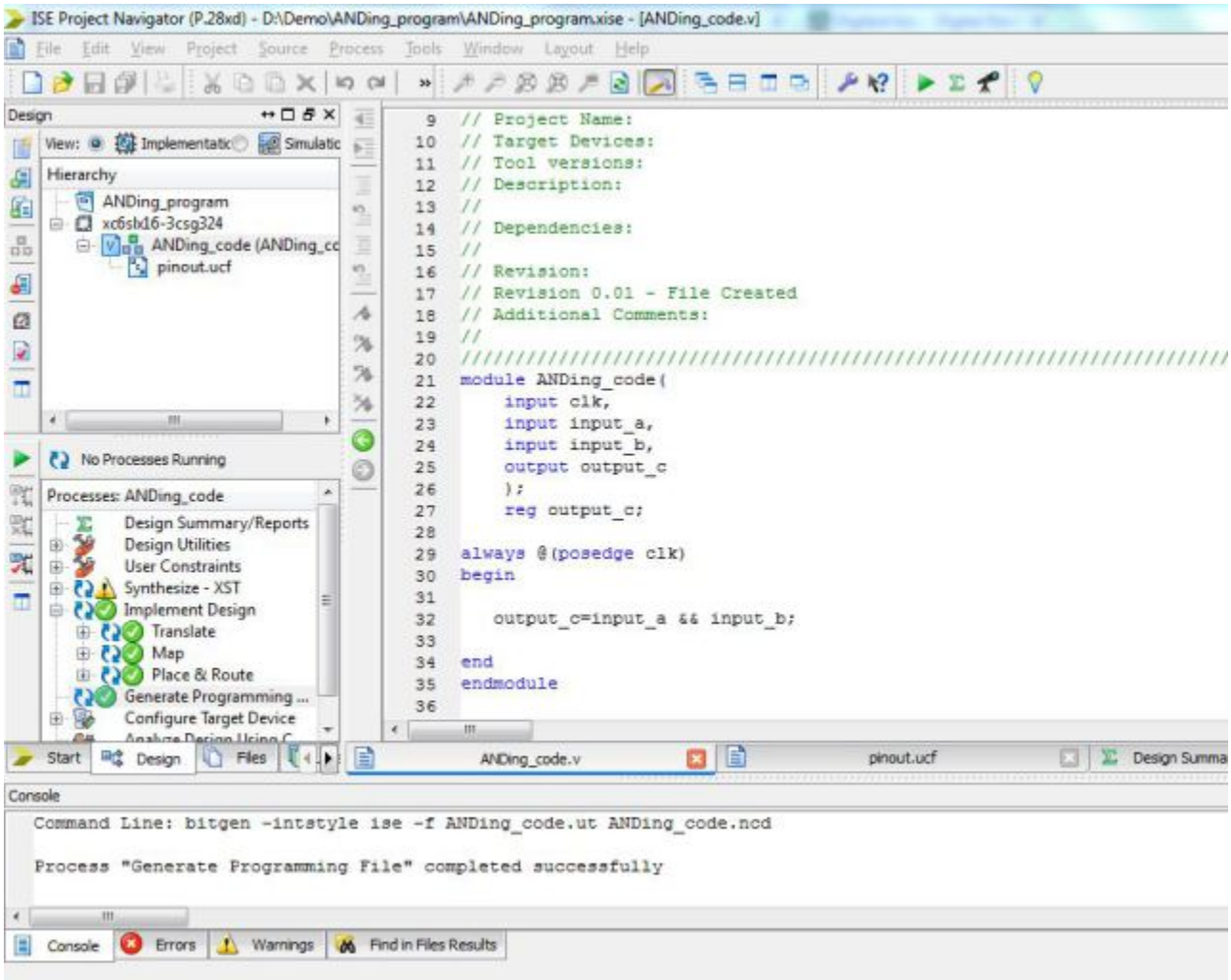
Step 10: Write the inputs, outputs and its pin location in proper format of **.ucf** file. (use datasheet of Xilinx board for pin location). Here two switchs SW0 and SW1 are used for input and one led LD0 is used for output.



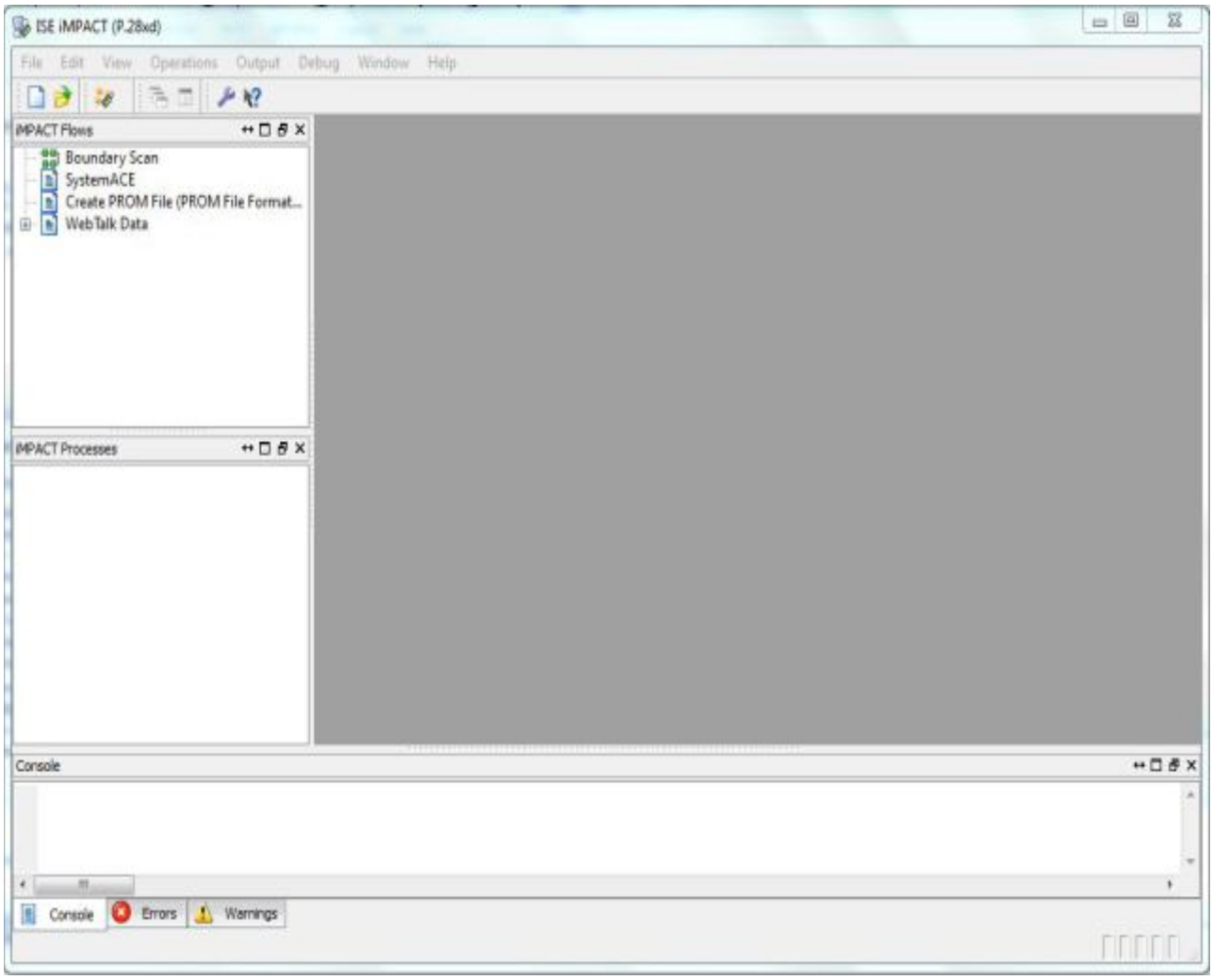
Step 11: Open main ANDing program and double click on Synthesize – XST. After successful completion of Synthesis, double click on Implement Design. Implement design consists of three parts-

- Translate
- Map
- Place and Route

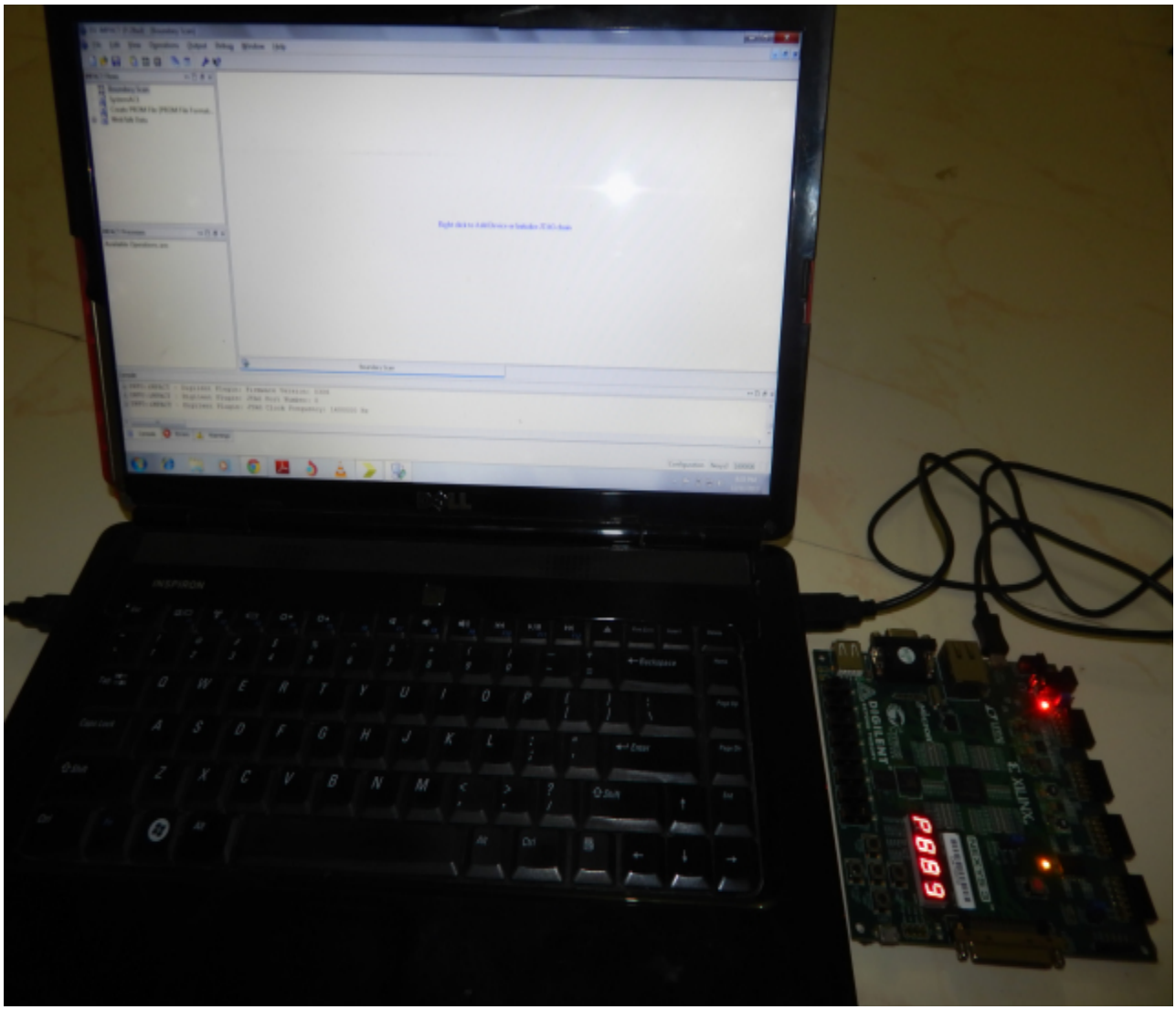
For detail information of Implement design Click Here. After successful completion of implement design double click on Programming File Generation. Programming File Generation produces a bitstream for Xilinx device configuration. Successful completion of all these process the window looks like.



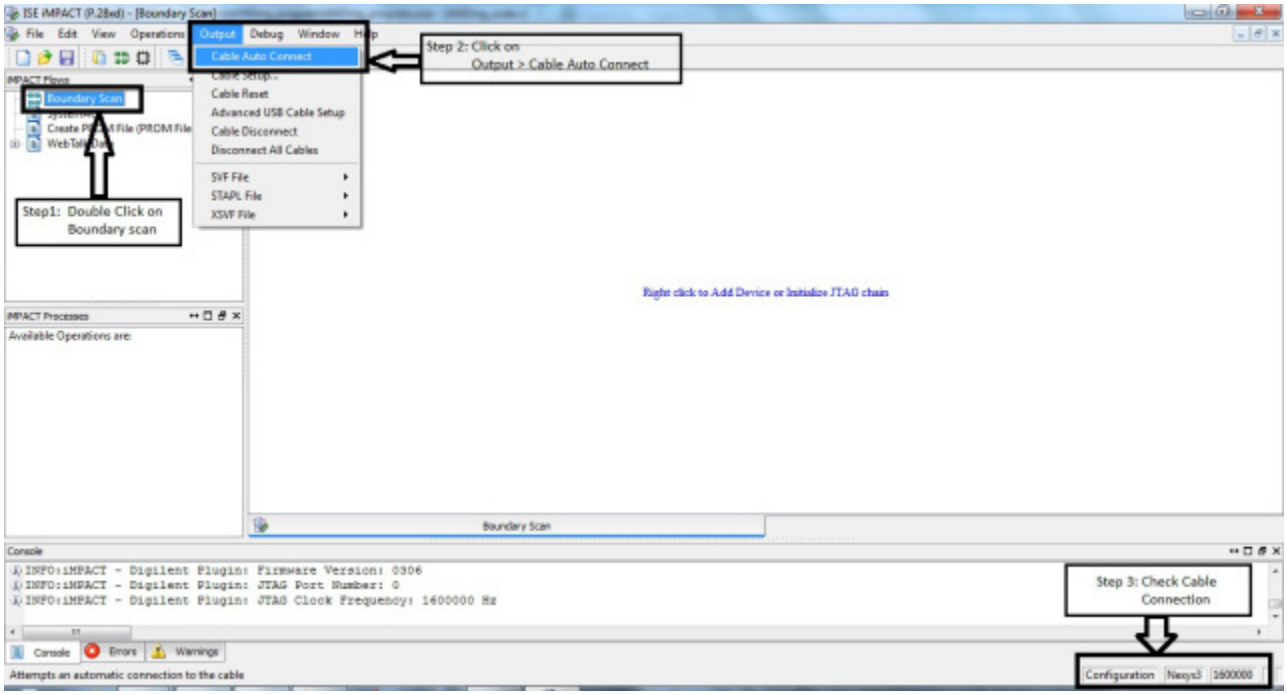
Step 12: Double click on **Configure Target Device** and a new **ISE iMPACT** window open.



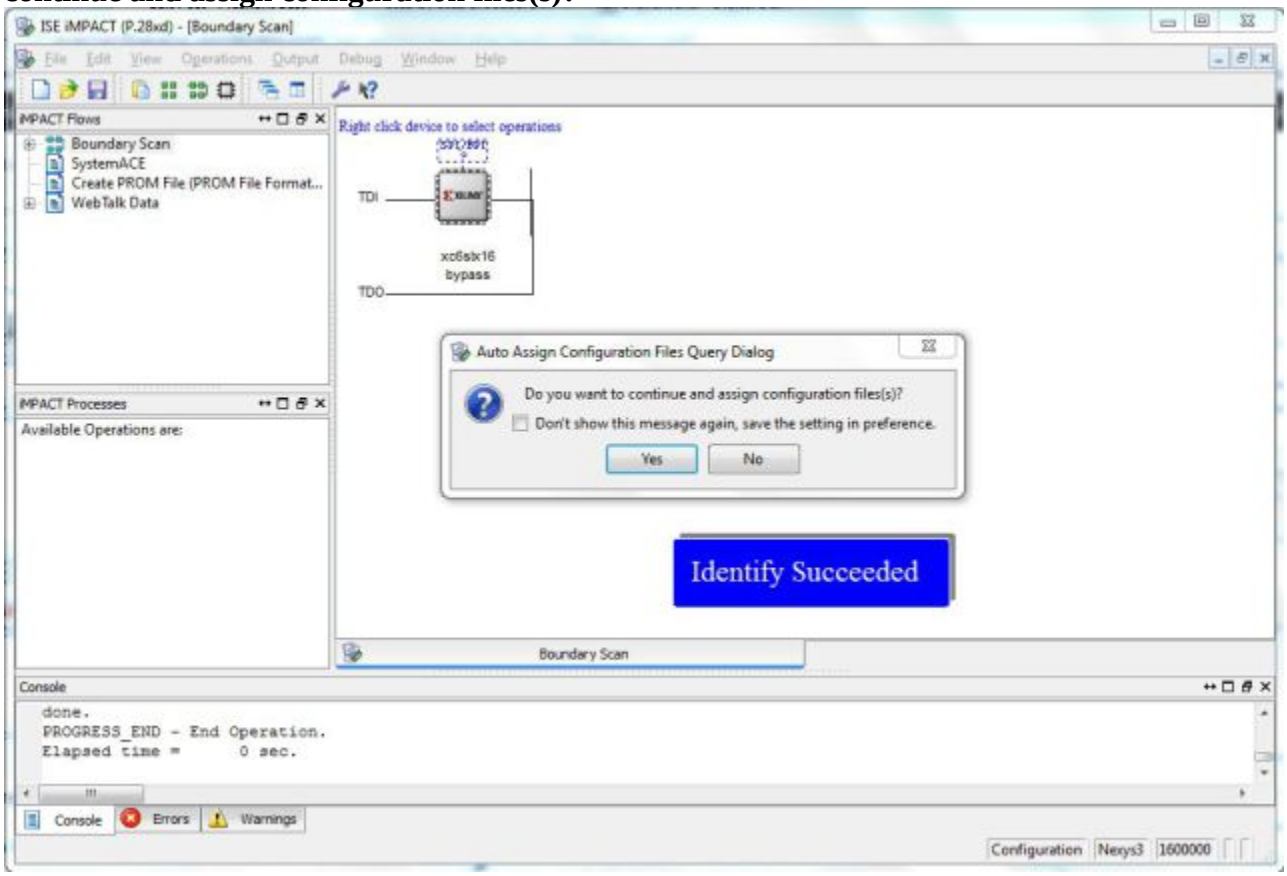
Step 13: Connect the Xilinx board to your PC/Laptop using USB cable.



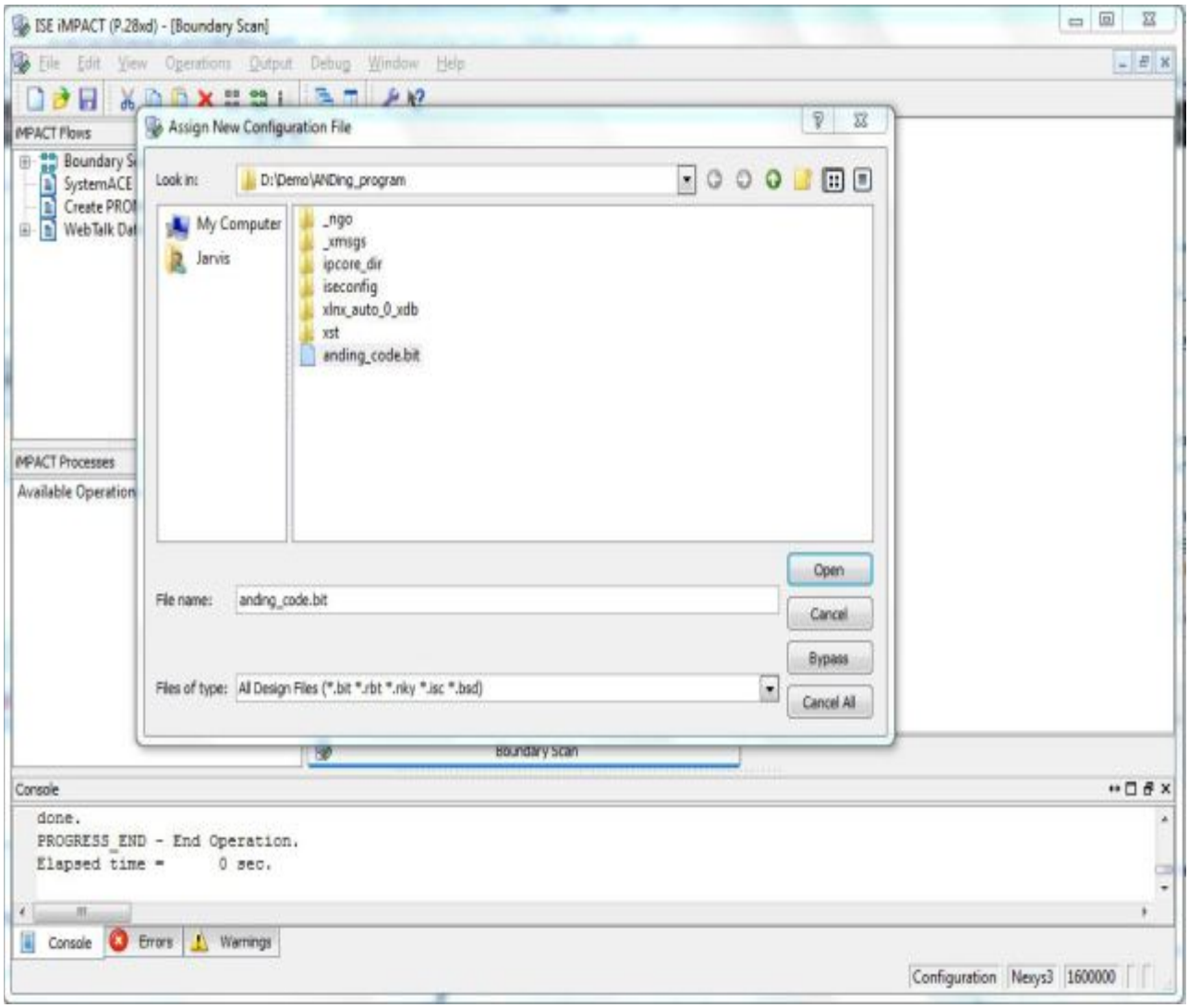
Step 14: Double click on Boundary scan. Check auto cable connection **Output > Cable Auto Connect** and if cable is connected then Window bottom part looks like step 3 shown in below.



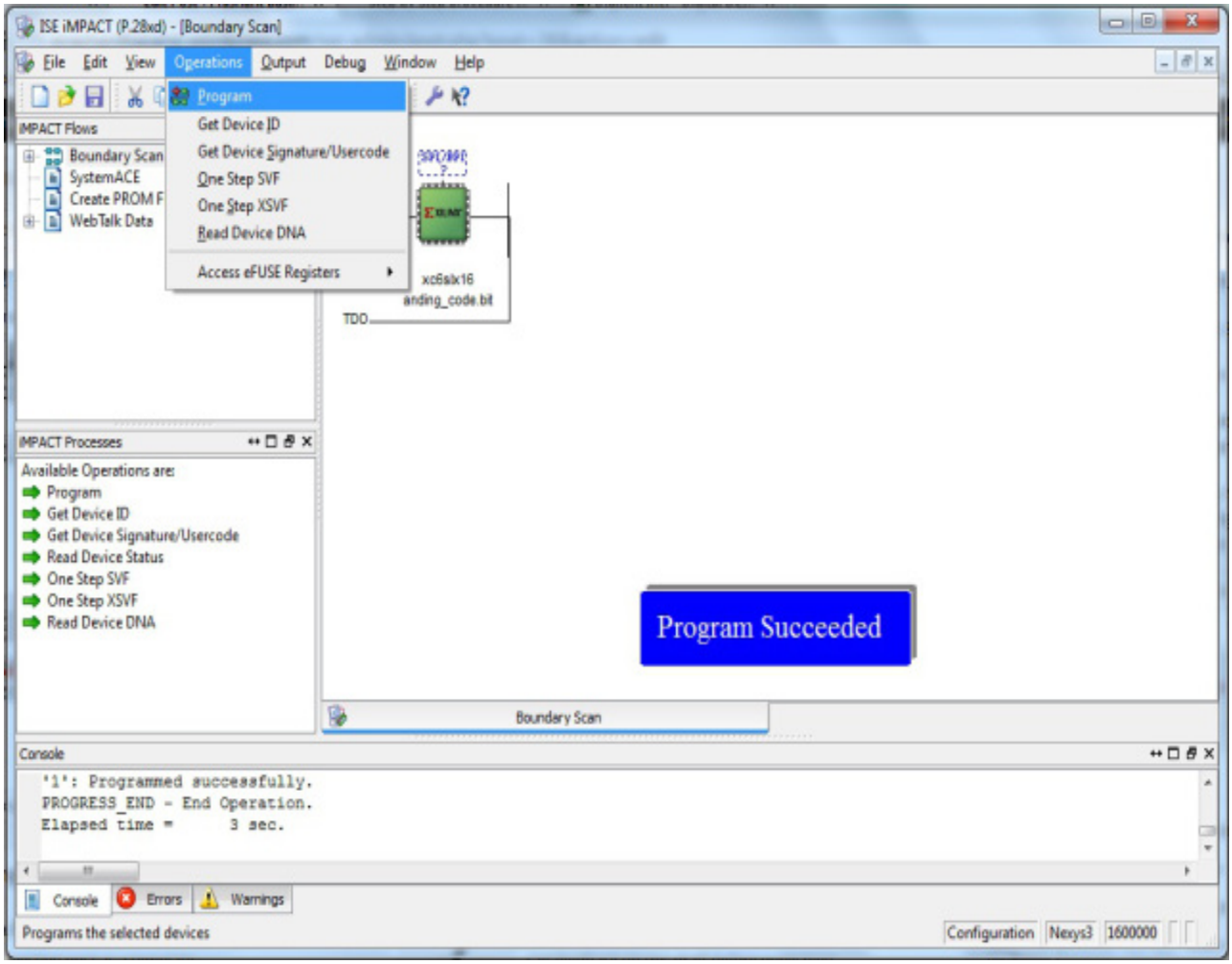
Step 15: Click on **File > Initialize Chain.** After that they ask for **"Do you want to continue and assign configuration files(s)?"**



Click on **Yes** and select the generated bit-stream file.



Step 16: Click on Open. After that they ask **"Do you want to attach an SPI or BPI PROM to this device?"** click on No tab. Click on **Operations > Program.** If Programming successful then they shows **Program Succeeded.**



Step 17: Check the output on hardware(Board). Here I am giving input through switches and output shows on LED. Output:

1. Switch1(OFF) AND Switch2(OFF) = LED(OFF)
2. Switch1(ON) AND Switch2(OFF) = LED(OFF)
3. Switch1(OFF) AND Switch2(ON) = LED(OFF)
4. Switch1(ON) AND Switch2(ON) = LED(ON)