# Simulation Lab 1
# How to Use Xilinx ISE® Design Suite Project Navigator

National
Science
Foundation

Funded in part, by a grant from the
National Science Foundation
DUE 1003736 and 1068182

## Acknowledgements

Developed by Craig Kief, Alonzo Vera, Alexandria Haddad, and Quinlan Cao, at the Configurable Space Microsystems Innovations & Applications Center (COSMIAC). Based on original tutorial developed by Bassam Matar, Engineering Faculty at Chandler-Gilbert Community College, Chandler, Arizona. *Funded by the National Science Foundation (NSF).*

## Lab Summary

This is a voluntary lab for those of you who lack experience in the use of Xilinx ISE®. We will walk you through a small version of a lab session and produce a fictitious lab report. In the process you will use all of the main steps that you will use in later lab work.

This lab teaches design entry, simulation, and prototyping using tools provided by Xilinx ISE® for this purpose. We will show how a simple design circuit of a **2**-input AND gate can be directly entered into Xilinx ISE® for synthesis, post synthesis simulation, and timing analysis. We will show the implementation of more complex designs in future labs by running them through the design flow illustrated in this lab.

## Lab Goal

The goal of this lab is to learn how to use the Xilinx ISE® software by implementing inputs and outputs for a simple 2-input AND gate.

## Learning Objectives

1. Use the Xilinx ISE® Schematic Editor to create a 2-input AND gate project using the free ISE® WebPACK™.
2. Compile and simulate the 2-input AND design.
3. Compile and simulate the 4-input XOR design.

## Grading Criteria

Your grade is determined by your instructor.

## Time Required

2-3 hours

## Lab Preparation

- Read this document completely before you start on this experiment.
- Print out the laboratory experiment procedure that follows.

## Equipment and Materials

Access to Xilinx software

| Software needed | Quantity |
|---|---|
| Download the ISE® WebPACK™ software from the Xilinx website, www.xilinx.com. | 1 |

## Additional References

Current Xilinx ISE Software manuals found on Xilinx web site: www.xilinx.com.

## Lab Procedure 1: Download and Install ISE® WebPACK™

In this lab, you will first load and license the current version of the free ISE® WebPACK™. Next, you will create a schematic of a small circuit of limited functionality. If you already have a copy of Xilinx ISE® or the WebPACK™ you may skip these steps.

You might be using a different version of ISE than the one used in this lab. However, there should be minimal changes in the instructions.

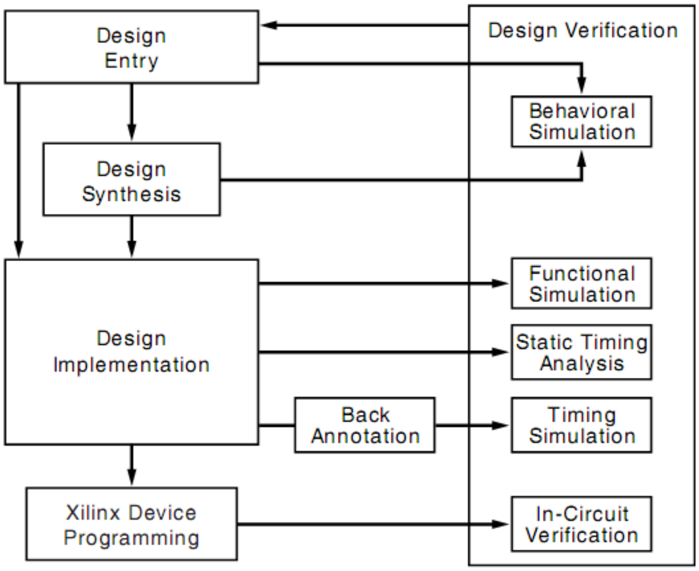
### Installing the ISE® WebPACK™ Software

Step 1:
Register with Xilinx at www.xilinx.com. Navigate to the http://www.xilinx.com/products/design-tools/ise-design-suite/ise-webpack.htm page and download the free ISE® WebPACK™.

Step 2:
Install ISE® WebPACK™.
System Requirements:

Microsoft Windows XP Professional7, Vista or Linux.
Step 3:
Have Fun!

## Lab Procedure 2: Design a 2-input AND Circuit



**Xilinx Design Process overview:**

**Step 1: Design Entry**

- Two design methods:

1. HDL (Verilog or VHDL)
   or
2. Schematic drawings.

*For the simulation part of our class, we will use the schematic method and VHDL.*

**Step 2: Design Synthesis**

- Translate VHDL and schematic files into an industry standard format EDIF file.

**Step 3: Design Implementation**

- Translate Map, Place and Route. This process will generate a configuration file (.BIT) for FPGA programming.

**Step 4: Xilinx Device Programming**

- Download JED file into FPGA

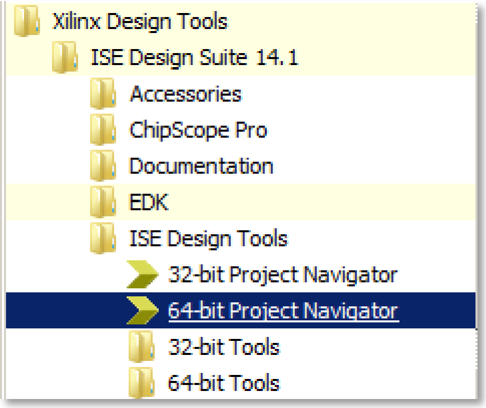
In this portion of the lab, you will use Xilinx ISE software to build a simple 2-input AND circuit. This lab focuses on Steps 1 and 2 of the Xilinx Design Process. We will implement Steps 3 and 4 in upcoming labs.

**NOTE**: At the conclusion of the lab, you will be asked to comment on why certain steps are required. Be sure to take notes on these questions as they appear in the lab procedure.

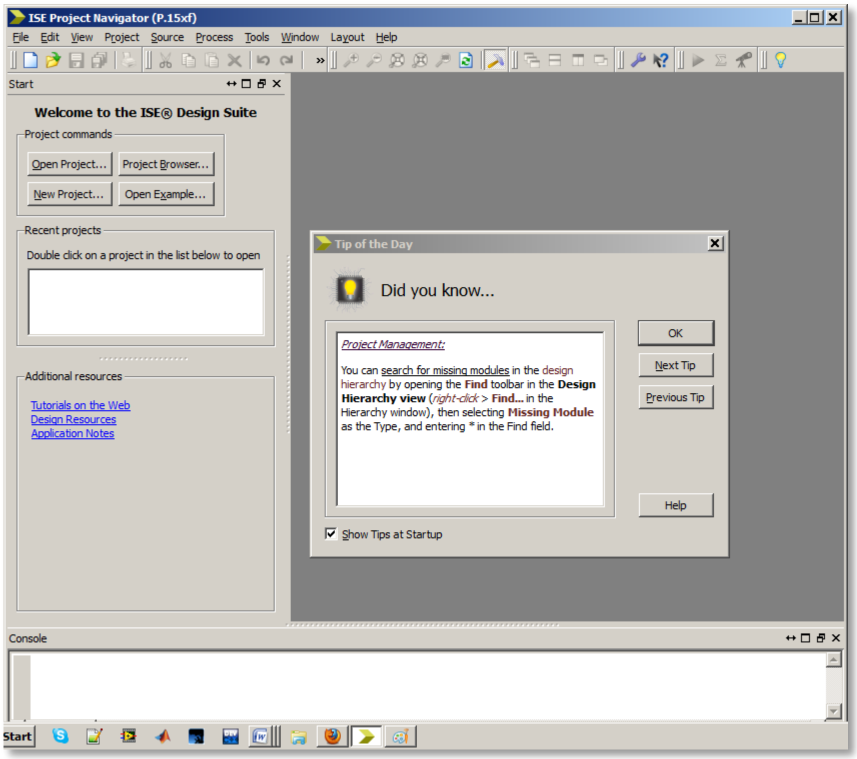
### Step 1: Design Entry

1. Open **Xilinx ISE Design Tool Project Navigator.**

   Your system might have slightly different Start Menu options.



2. The ISE Project Navigator window opens, with the Tip of the Day displayed. Click **OK** to close the Tip of the Day.


ISE Project Navigator (P.15xf)
File Edit View Project Source Process Tools Window Layout Help
Start
Welcome to the ISE® Design Suite
Project commands
Open Project... Project Browser...
New Project... Open Example...
Recent projects
Double click on a project in the list below to open
Additional resources
Tutorials on the Web
Design Resources
Application Notes
Tip of the Day
Did you know...
OK
Next Tip
Previous Tip
Help
Show Tips at Startup
Console

*Project Management:*

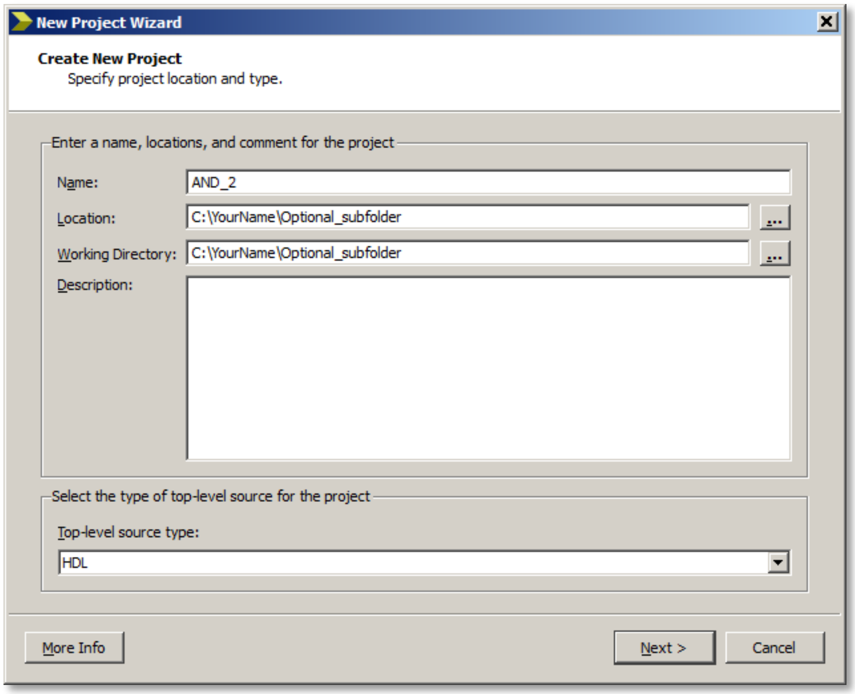
You can search for missing modules in the design hierarchy by opening the **Find** toolbar in the **Design Hierarchy view** (*right-click* > **Find...** in the Hierarchy window), then selecting **Missing Module** as the Type, and entering * in the Find field.

☑ Show Tips at Startup

3. Start a new project by selecting **File → New Project** from the menu. The New Project Wizard starts.

   a. Type **AND_2** in the Name text box.
   b. Select a location on your computer to save your project files by clicking the ellipsis (**...**) button to the right of the **Location** text box.
   c. Under **Top-level source type**, select **HDL**.
   d. Click **Next**.

The Project Settings dialog box opens.



**Create New Project**

Specify project location and type.

Enter a name, locations, and comment for the project

Name: AND_2

Location: C:\YourName\Optional_subfolder

Working Directory: C:\YourName\Optional_subfolder

Description:

Select the type of top-level source for the project

Top-level source type: HDL

**NOTE**: File names must start with a letter. Use underscores ( _ ) for readability. Do not use hyphens (-); although the file name will work, the entity name will not. More on this later.

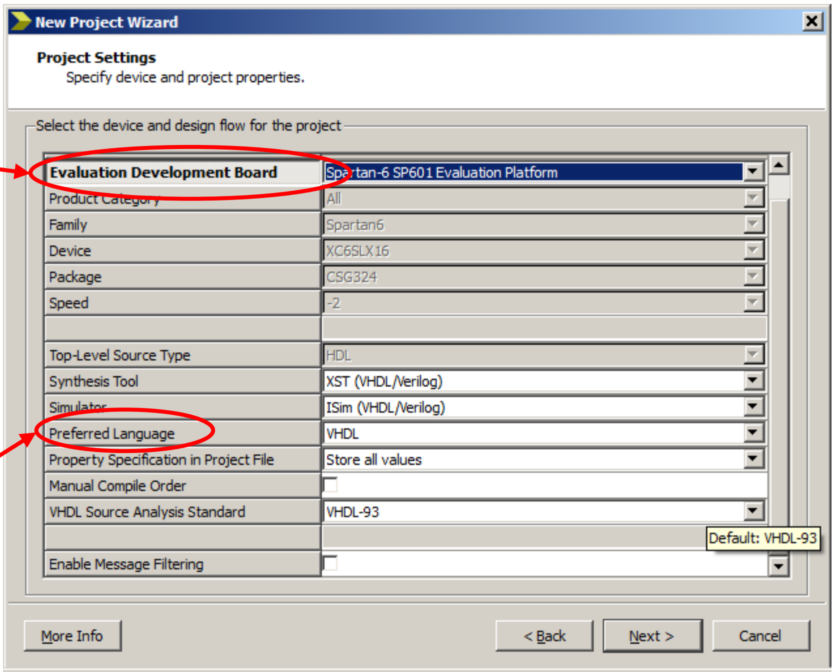
4. Select **Spartan-6 SP601 Evaluation Platform** from the **Evaluation Development Board** drop down menu.

   The **Product Category**, **Family**, **Device**, **Package**, and **Speed** should all automatically populate (top half of the screen). You will need to set some options in the lower half of the dialog box.

5. Select **VHDL** from the **Preferred Language** drop down menu.

   The Project Settings should resemble the figure to the right.

6. Click **Next**.



New Project Wizard

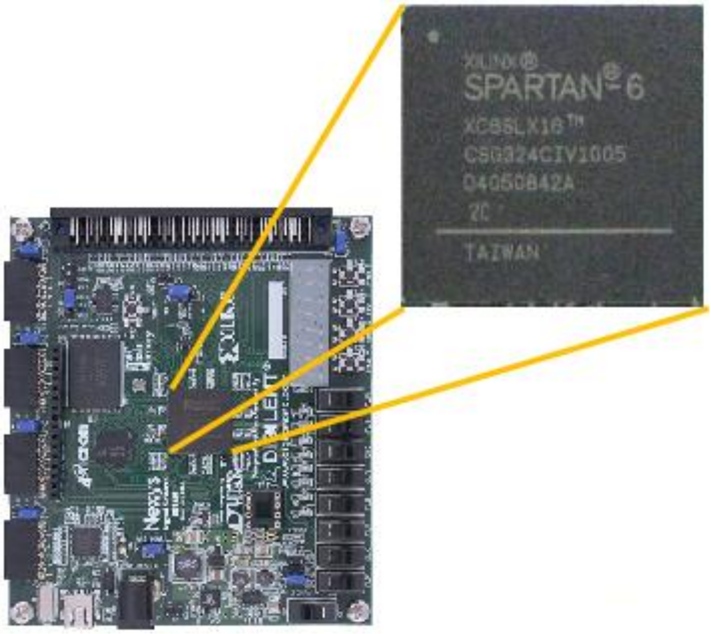
**Project Settings**

Specify device and project properties.

Select the device and design flow for the project

| | |
|---|---|
| **Evaluation Development Board** | Spartan-6 SP601 Evaluation Platform |
| Product Category | All |
| Family | Spartan6 |
| Device | XC6SLX16 |
| Package | CSG324 |
| Speed | -2 |
| | |
| Top-Level Source Type | HDL |
| Synthesis Tool | XST (VHDL/Verilog) |
| Simulator | ISim (VHDL/Verilog) |
| Preferred Language | VHDL |
| Property Specification in Project File | Store all values |
| Manual Compile Order | ☐ |
| VHDL Source Analysis Standard | VHDL-93 |
| | Default: VHDL-93 |
| Enable Message Filtering | ☐ |

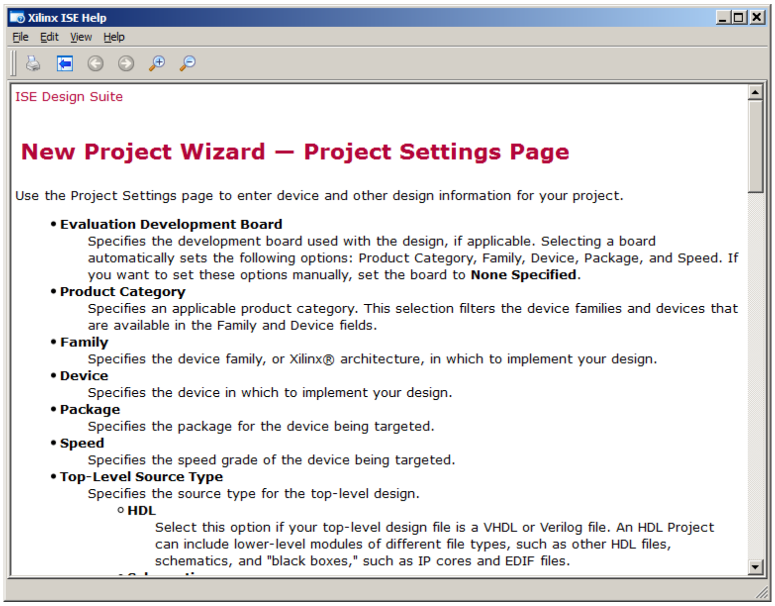
**NOTE**: The options specified are for the **Spartan 6 LX FPGA**. Your board might be different than the board used for these instructions. Board specifications are printed on the FPGA chip in the middle of the board as shown in the figure to the right. The board information is also listed on the box it came in.



**NOTE**: You can also leave the **Evaluation Development Board** drop down set to **None Specified**, and change each of the options manually.

**NOTE**: Press **F1** or select **Help → Help Topics** from the menu to access context sensitive **Help** about any ISE option.

The **Help** menu has many other support options available also, e.g., **Xilinx on the Web Tutorials**. Selecting this option opens the Xilinx support page on a separate workspace tab inside the ISE window.



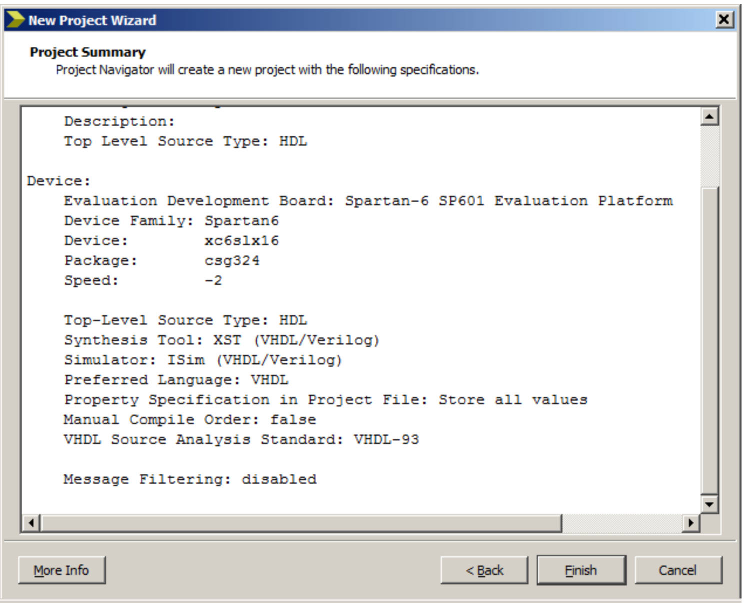
ISE Design Suite

**New Project Wizard — Project Settings Page**

Use the Project Settings page to enter device and other design information for your project.

- **Evaluation Development Board**
  Specifies the development board used with the design, if applicable. Selecting a board automatically sets the following options: Product Category, Family, Device, Package, and Speed. If you want to set these options manually, set the board to **None Specified**.
- **Product Category**
  Specifies an applicable product category. This selection filters the device families and devices that are available in the Family and Device fields.
- **Family**
  Specifies the device family, or Xilinx® architecture, in which to implement your design.
- **Device**
  Specifies the device in which to implement your design.
- **Package**
  Specifies the package for the device being targeted.
- **Speed**
  Specifies the speed grade of the device being targeted.
- **Top-Level Source Type**
  Specifies the source type for the top-level design.
  - **HDL**
    Select this option if your top-level design file is a VHDL or Verilog file. An HDL Project can include lower-level modules of different file types, such as other HDL files, schematics, and "black boxes," such as IP cores and EDIF files.

7. The Project Summary displays. Verify the file type and name are correct then click **Finish** to complete the New Project creation process.

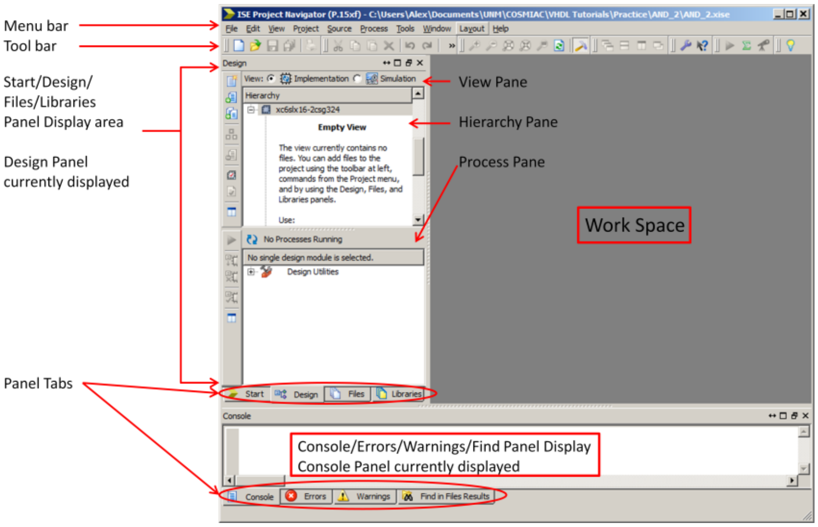


**Project Summary**
Project Navigator will create a new project with the following specifications.

```
    Description:
    Top Level Source Type: HDL

Device:
    Evaluation Development Board: Spartan-6 SP601 Evaluation Platform
    Device Family: Spartan6
    Device:        xc6slx16
    Package:       csg324
    Speed:         -2

    Top-Level Source Type: HDL
    Synthesis Tool: XST (VHDL/Verilog)
    Simulator: ISim (VHDL/Verilog)
    Preferred Language: VHDL
    Property Specification in Project File: Store all values
    Manual Compile Order: false
    VHDL Source Analysis Standard: VHDL-93

    Message Filtering: disabled
```

8. The Project Wizard window closes and a new project is created. The Project Navigator window provides all the options you need to create and/or include project files.

   Take a minute to familiarize yourself with the Project Navigator window **Menu**, **Toolbar**, **Panels**, and **Work Space** area.


Menu bar
Tool bar
Start/Design/
Files/Libraries
Panel Display area
Design Panel
currently displayed
Panel Tabs
View Pane
Hierarchy Pane
Process Pane
Work Space
Console/Errors/Warnings/Find Panel Display
Console Panel currently displayed
File Edit View Project Source Process Tools Window Layout Help
Design
View: Implementation Simulation
Hierarchy
xc6slx16-2csg324
Design Utilities
Start Design Files Libraries
Console
Console Errors Warnings Find in Files Results

ISE Project Navigator (P.15xf) - C:\Users\Alex\Documents\UNM\COSMIAC\VHDL Tutorials\Practice\AND_2\AND_2.xise

Empty View

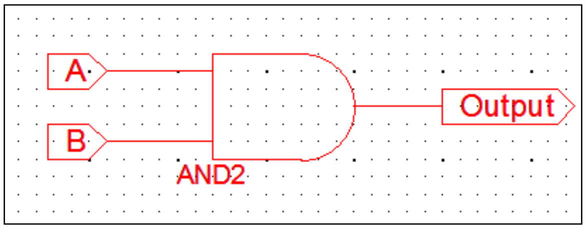
The view currently contains no files. You can add files to the project using the toolbar at left, commands from the Project menu, and by using the Design, Files, and Libraries panels.

Use:

No Processes Running

No single design module is selected.

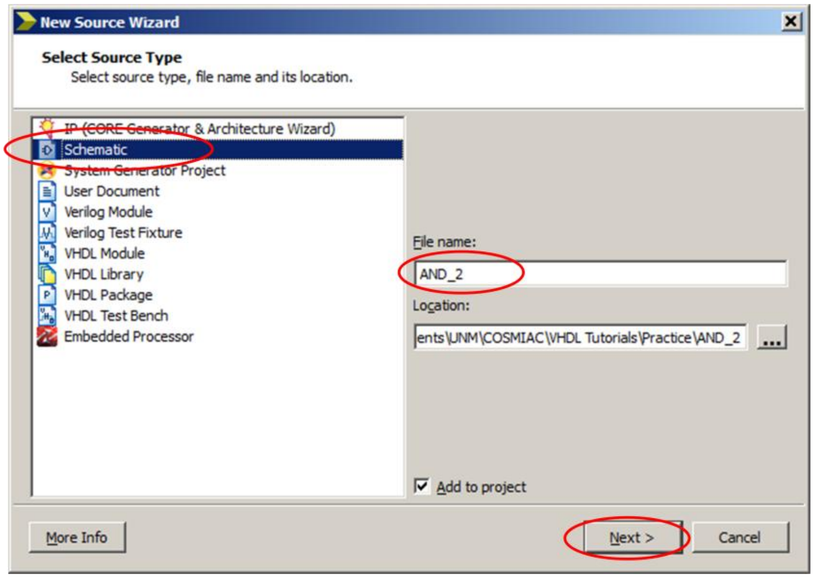
9. We will be building a simple 2-input AND gate as shown on the right. First we will use the schematic option, and then we will create a VHDL code file. This provides an introduction to the VHDL code and syntax.



10. Add a New Source Schematic to your project by selecting **Project→New source** from the menu. The **New Source Wizard** dialog box opens.

11. Select **Schematic** from the list of sources and type **AND_2** for the file name and click the **Next** button. The **Source Summary** dialog box opens. Click **Finish**.

**NOTE**: AND-2 is just a name for your schematic. You can name the schematic file any name you desire.



**Select Source Type**

Select source type, file name and its location.

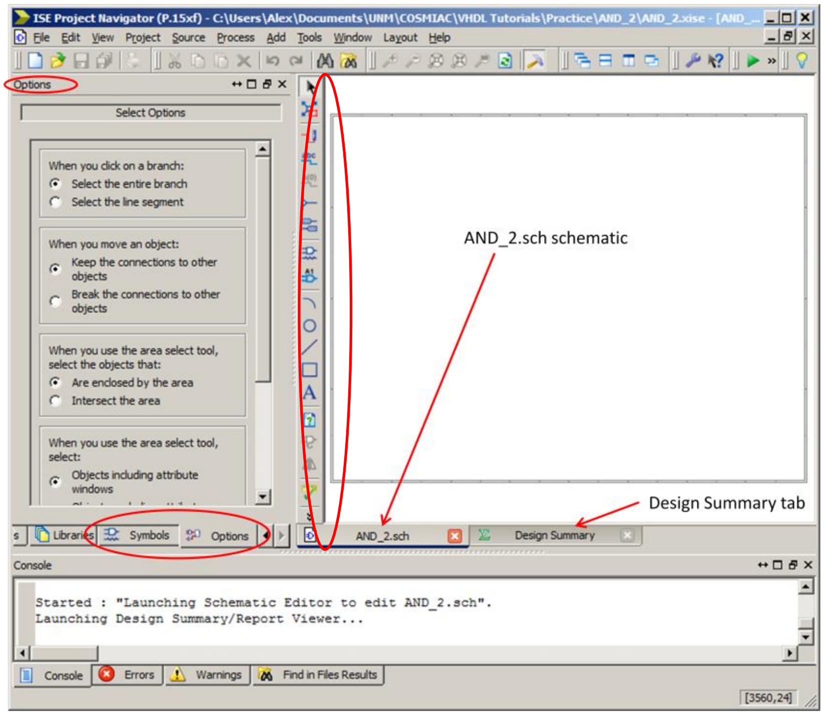
File name: AND_2

Location: ents\UNM\COSMIAC\VHDL Tutorials\Practice\AND_2

☑ Add to project

12. The new **AND_2.sch schematic** is displayed in the workspace. You might also see a workspace tab for the **Design Summary**.

    The left panel display area now shows the **Options panel** and additional panel tabs for **Options and Symbols.** Note there is also a left pointing arrow to navigate the panel tabs. Also notice the **schematic toolbar** on the left of the workspace.


ISE Project Navigator (P.15xf) - C:\Users\Alex\Documents\UNM\COSMIAC\VHDL Tutorials\Practice\AND_2\AND_2.xise - [AND_...
File Edit View Project Source Process Add Tools Window Layout Help
Options
Select Options
Libraries
Symbols
Options
AND_2.sch
Design Summary
Console
Console
Errors
Warnings
Find in Files Results
[3560,24]

When you click on a branch:
● Select the entire branch
○ Select the line segment

When you move an object:
● Keep the connections to other objects
○ Break the connections to other objects

When you use the area select tool, select the objects that:
● Are enclosed by the area
○ Intersect the area

When you use the area select tool, select:
● Objects including attribute windows

AND_2.sch schematic

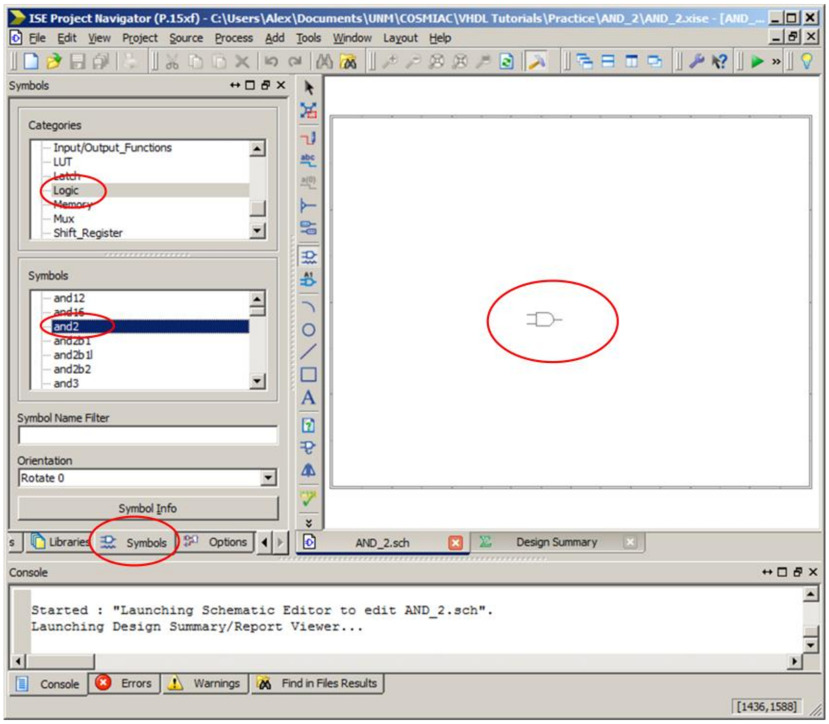
Design Summary tab

```
Started : "Launching Schematic Editor to edit AND_2.sch".
Launching Design Summary/Report Viewer...
```

13. Click the **Symbols** panel tab to display the Symbols panel.

14. Under **Categories** select **Logic**.

15. Under **Symbols** select **and2**, and then click the mouse on the schematic workspace to place the and2 symbol.

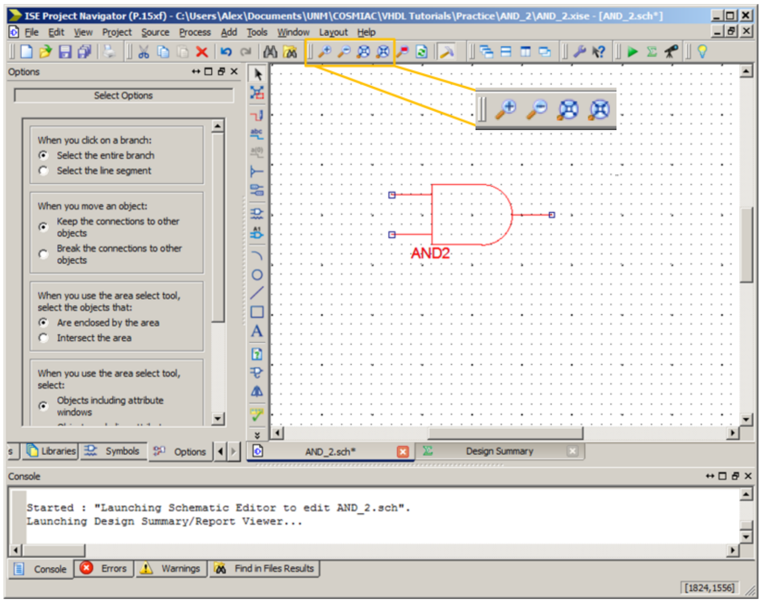


```
Started : "Launching Schematic Editor to edit AND_2.sch".
Launching Design Summary/Report Viewer...
```

### Using the symbols tools

All the symbols tools are either on or off. Selecting a tool, either from the symbols panel, menu, or the toolbar turns the tool on. The mouse pointer will reflect the symbol that is currently on. Clicking on the workspace will place the current symbol at that location. The tool remains on, and multiple symbol placements can be made, until the tool is turned off by pressing the ESC key. When a tool is active the options panel provides context sensitive options for that tool.

16. Zoom in or out, using the **Zoom** buttons on the toolbar, to resize your view.



When you click on a branch:
- ● Select the entire branch
- ○ Select the line segment

When you move an object:
- ● Keep the connections to other objects
- ○ Break the connections to other objects

When you use the area select tool, select the objects that:
- ● Are enclosed by the area
- ○ Intersect the area

When you use the area select tool, select:
- ● Objects including attribute windows

```
Started : "Launching Schematic Editor to edit AND_2.sch".
Launching Design Summary/Report Viewer...
```

17. Add wires to the AND gate to enable future connection of pins and gates to the schematic.

    Click the **Add Wires** button on the **toolbar** to turn on the Add Wires tool.

    Add a wire to each input and the output of the AND gate by **Clicking** once on the end of the AND gate to start the wire then **double-clicking** to end the wire. Repeat for each wire.

    Press the esc key to turn off the Add Wires tool.

**NOTE:** For this simple schematic it would not normally be necessary to add wires. This step is for practice purposes only.

### Adding I/O Markers

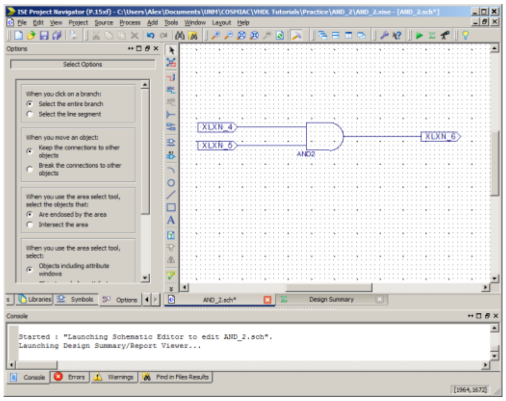
An I/O marker is an input, output, or bidirectional signal. This establishes net polarity (direction of signal flow) and shows that the net is externally accessible. Without pins, the device is meaningless. All primary inputs and outputs must be marked with I/O markers. You can add an I/O Marker from the menu or the toolbar:

- Select **Add → I/O Marker** from the menu.
- Click the **Add I/O Marker** toolbar button .

18. Click the **Add I/O Markers** button on the **toolbar** to turn on the Add I/O Markers tool.

    Add an I/O marker by placing the mouse pointer over the end of a wire and **clicking once**. Click on each wire end to add an I/O marker.

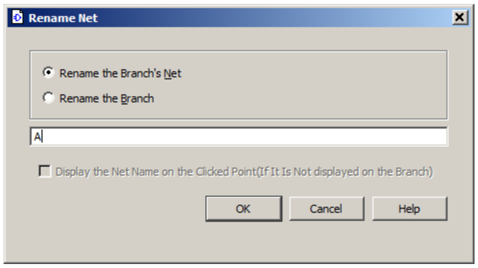
    Press the esc key to turn off the Add I/O Markers tool.


ISE Project Navigator (P.15xf) - C:\Users\Alex\Documents\UNM\COSMIAC\VHDL Tutorials\Practice\AND_2\AND_2.xise - [AND_2.sch*]
File Edit View Project Source Process Add Tools Window Layout Help
Options
Select Options
When you click on a branch:
Select the entire branch
Select the line segment
When you move an object:
Keep the connections to other objects
Break the connections to other objects
When you use the area select tool, select the objects that:
Are enclosed by the area
Intersect the area
When you use the area select tool, select:
Objects including attribute windows
XLXN_4
XLXN_5
AND2
XLXN_6
Libraries
Symbols
Options
AND_2.sch*
Design Summary
Console
Errors
Warnings
Find in Files Results
[1964,1672]

```
Started : "Launching Schematic Editor to edit AND_2.sch".
Launching Design Summary/Report Viewer...
```

19. Change the name of the I/O marker by **right-clicking** on the marker and selecting **Rename Port** from the shortcut menu.

    Type the new name for the port and click **OK**.

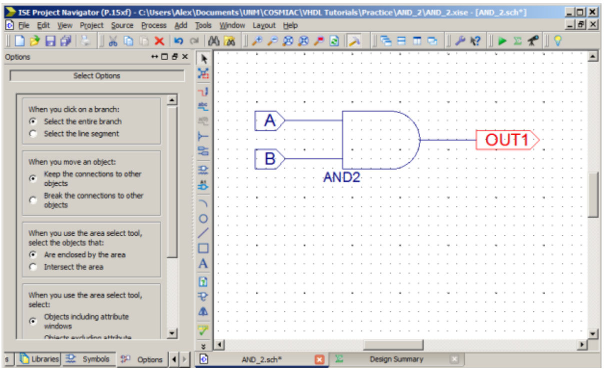


● Rename the Branch's Net
○ Rename the Branch

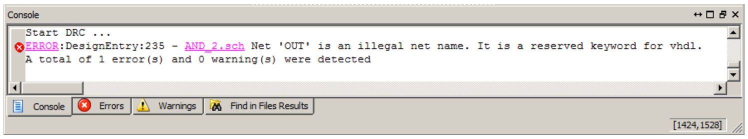
A

□ Display the Net Name on the Clicked Point(If It Is Not displayed on the Branch)

20. Once the schematic is complete, test for wiring errors by selecting **Tools → Check Schematic** from the menu.

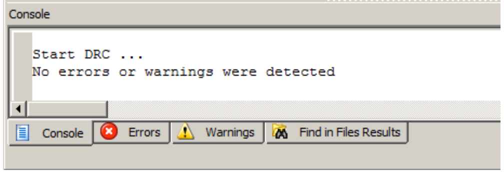

ISE Project Navigator (P.15xf) - C:\Users\Alex\Documents\UNM\COSMIAC\VHDL Tutorials\Practice\AND_2\AND_2.xise - [AND_2.sch*]
File Edit View Project Source Process Add Tools Window Layout Help
Options
Select Options
When you click on a branch:
Select the entire branch
Select the line segment
When you move an object:
Keep the connections to other objects
Break the connections to other objects
When you use the area select tool, select the objects that:
Are enclosed by the area
Intersect the area
When you use the area select tool, select:
Objects including attribute windows
A
B
AND2
OUT1
Libraries
Symbols
Options
AND_2.sch*
Design Summary

21. The Console Panel will display any errors or warnings. If you have errors, fix them and run **Check Schematic** again.



```
Start DRC ...
ERROR:DesignEntry:235 - AND_2.sch Net 'OUT' is an illegal net name. It is a reserved keyword for vhdl.
A total of 1 error(s) and 0 warning(s) were detected
```

22. **Save your file.**



```
Start DRC ...
No errors or warnings were detected
```

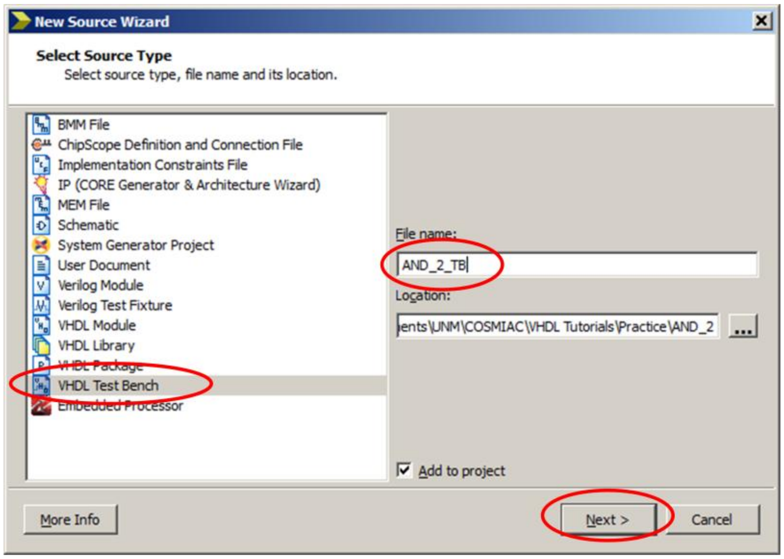
## Step 2: Design Simulation

1. Create a test bench for your file. **Right-click** in the Hierarchy Pane and select **New Source** from the short cut menu.

2. Select **VHDL Test Bench** as the source type.

3. Enter **AND_2_TB** as the file name.

**NOTE**: It is good practice to name your test bench file with the same name as the original file and append **_TB** to the end of the file name.



New Source Wizard

Select Source Type

Select source type, file name and its location.

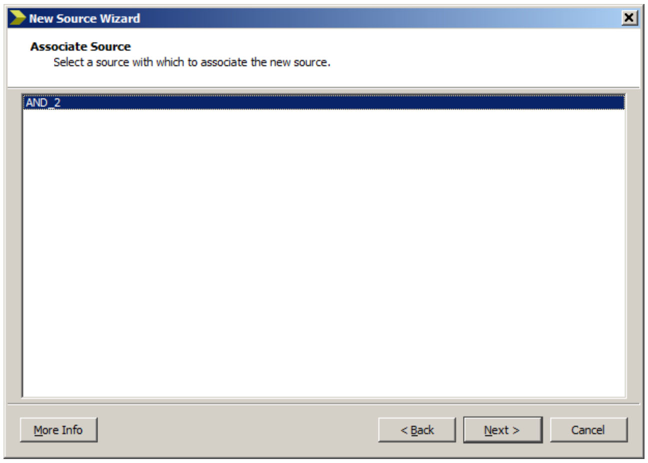
File name: AND_2_TB

Location: ents\UNM\COSMIAC\VHDL Tutorials\Practice\AND_2

☑ Add to project

4. Click the **Next** button. The Associate Source dialog box displays with **AND_2** selected. Click the Next Button.

**NOTE**: When you have a project with multiple .vhd files, every file is listed in the Associate Source dialog box. Before you click **Next**, make sure you have the correct file selected for the test bench you are creating.



New Source Wizard

Associate Source

Select a source with which to associate the new source.

AND_2

5. The New Source Summary dialog box displays.

6. Click the **Finish** button.

7. The test bench file will display in the workspace. **Make the changes highlighted on the right.**

**NOTE**: The PROCESS section of the simulation sets the values of A and B from 0 to 1 and then waits for a small period of time (25 and 50 nanoseconds, respectively).

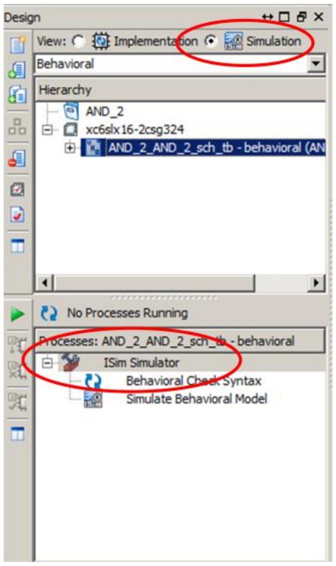
8. Click the **Save All** button on the toolbar.

```
-- initialize inputs to 0

   SIGNAL A  :         STD_LOGIC:='0';
   SIGNAL B  :         STD_LOGIC:='0';
   SIGNAL OUTPUT :          STD_LOGIC;

-- *** Test Bench - User Defined Section ***
~~-- tb : PROCESS~~
~~-- BEGIN~~
~~--    WAIT; -- will wait forever~~
~~-- END PROCESS;~~

-- offset timing of inputs simulate all input options.
-- output = 1 only when A = B = 1
         A_process : PROCESS
         BEGIN
                  A <= NOT A;
                  WAIT FOR 25 ns;
         END PROCESS;

         B_process: PROCESS
         BEGIN
                  B <= NOT B;
                  WAIT FOR 50 ns;
         END PROCESS;

-- *** End Test Bench - User Defined Section ***
```

9. On the **View Pane**, select **Simulation**. The Simulation Hierarchy and Process options display.

10. **Expand** the ISim Simulator tool if it isn't already expanded.

11. **Double-click** the **Simulate Behavioral Model** tool. Click **Yes** to save any changes.



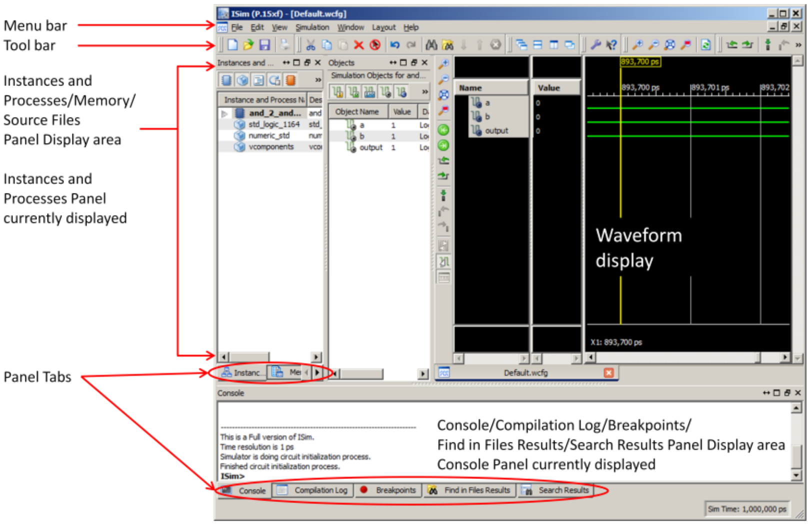
Design

View: ○ Implementation ● Simulation

Behavioral

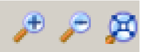
Hierarchy
AND_2
xc6slx16-2csg324
AND_2_AND_2_sch_tb - behavioral (AN
No Processes Running
Processes: AND_2_AND_2_sch_tb - behavioral
ISim Simulator
Behavioral Check Syntax
Simulate Behavioral Model

12. The Simulation Elaboration will begin to run. When it is finished the **ISim** window will open displaying the simulation.


Menu bar
Tool bar
Instances and Processes/Memory/ Source Files Panel Display area
Instances and Processes Panel currently displayed
Panel Tabs
Waveform display
Console/Compilation Log/Breakpoints/ Find in Files Results/Search Results Panel Display area
Console Panel currently displayed
ISim (P.15xf) - [Default.wcfg]
File Edit View Simulation Window Layout Help
Instances and ...
Objects
Simulation Objects for and...
Instance and Process N...
and_2_and...
std_logic_1164
numeric_std
vcomponents
Object Name
Value
a 1
b 1
output 1
Name
Value
a 0
b 0
output 0
893,700 ps
893,701 ps
893,702
X1: 893,700 ps
Instanc...
Me
Default.wcfg
Console
Compilation Log
Breakpoints
Find in Files Results
Search Results
Sim Time: 1,000,000 ps

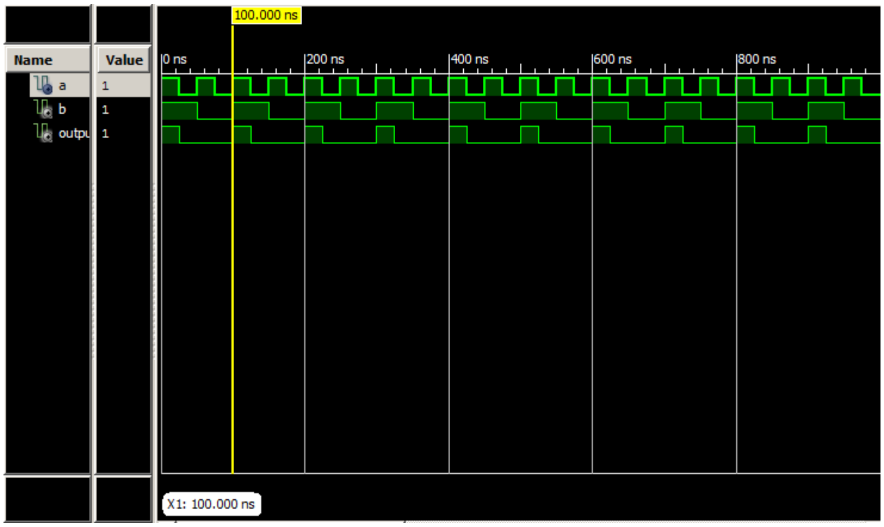
```
This is a Full version of ISim.
Time resolution is 1 ps
Simulator is doing circuit initialization process.
Finished circuit initialization process.
ISim>
```

13. Use the **Zoom to Full View** and the **Zoom In** buttons to display the simulation waveform clearer.



Review the waveform to ensure the testbench is working as designed.

The waveform should match the following logic table for this circuit, A•B=C.

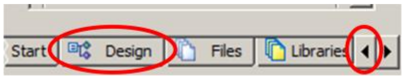


| A | B | = | C |
|---|---|---|---|
| 0 | 0 |  | 0 |
| 0 | 1 |  | 0 |
| 1 | 0 |  | 0 |
| 1 | 1 |  | 1 |

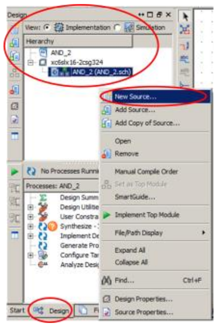
When you are finished, **close** the ISim application and return to ISE.

## Lab Procedure 3: Create VHDL file for 2-input AND circuit

1. **Click** the panel tab’s left triangle button to display the **Design** tab.

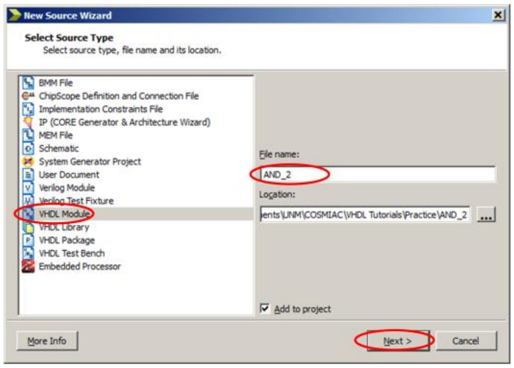


2. **Right-click** in the Design Panel and Click **New Source...** from the shortcut menu. The New Source Wizard displays.



3. Select **VHDL Module** from the source type list.

4. Type **AND_2** as the file name.


New Source Wizard

**Select Source Type**

Select source type, file name and its location.

File name: AND_2

Location: ents\UNM\COSMIAC\VHDL Tutorials\Practice\AND_2

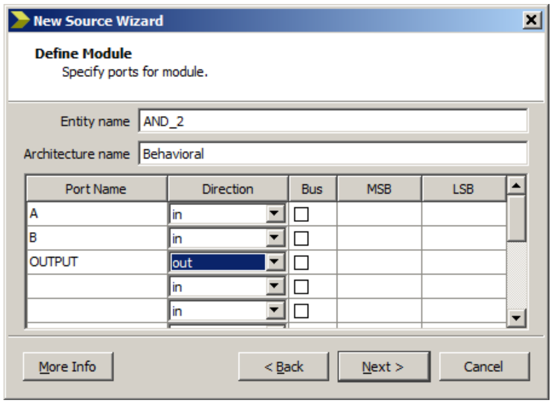
☑ Add to project

5. Click the **Next** button. The Define Module dialog displays.

6. Type the **port names** and select the **direction** of each port as shown in the figure to the right.

7. Click the **Next** button. The Summary dialog displays.

8. Click the **Finish** button to accept the New Source Wizards options.


New Source Wizard

**Define Module**

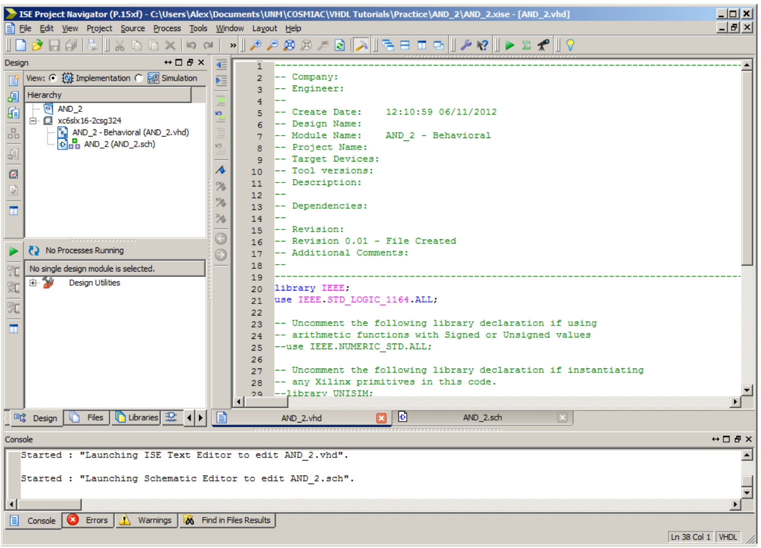
Specify ports for module.

Entity name: AND_2

Architecture name: Behavioral

| Port Name | Direction | Bus | MSB | LSB |
|---|---|---|---|---|
| A | in | □ | | |
| B | in | □ | | |
| OUTPUT | out | □ | | |
| | in | □ | | |
| | in | □ | | |

9. ISE will generate source code for you and it will display in the **workspace** area.


ISE Project Navigator (P.15xf) - C:\Users\Alex\Documents\UNM\COSMIAC\VHDL Tutorials\Practice\AND_2\AND_2.xise - [AND_2.vhd]

```
--------------------------------------------------------------------------------
-- Company:
-- Engineer:
--
-- Create Date:    12:10:59 06/11/2012
-- Design Name:
-- Module Name:    AND_2 - Behavioral
-- Project Name:
-- Target Devices:
-- Tool versions:
-- Description:
--
-- Dependencies:
--
-- Revision:
-- Revision 0.01 - File Created
-- Additional Comments:
--
--------------------------------------------------------------------------------
library IEEE;
use IEEE.STD_LOGIC_1164.ALL;

-- Uncomment the following library declaration if using
-- arithmetic functions with Signed or Unsigned values
--use IEEE.NUMERIC_STD.ALL;

-- Uncomment the following library declaration if instantiating
-- any Xilinx primitives in this code.
--library UNISIM;
```

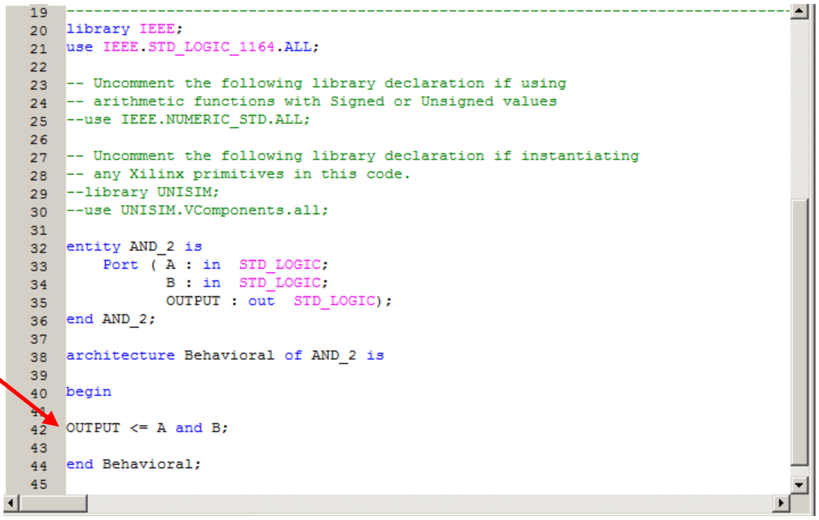
```
Started : "Launching ISE Text Editor to edit AND_2.vhd".

Started : "Launching Schematic Editor to edit AND_2.sch".
```

Basic VHDL code has three sections:  1) library, 2) entity, and 3) architecture. The library section holds all the libraries and packages needed for the code. All the code's inputs and outputs are declared in the entity section. The actual circuit's instruction code is in the architecture section.

10. Scroll down to the **architecture** section of the code and modify the code as highlighted:

```
begin

OUTPUT <= A and B;

end Behavioral;
```

11. Save your files by clicking on the **Save All** button.

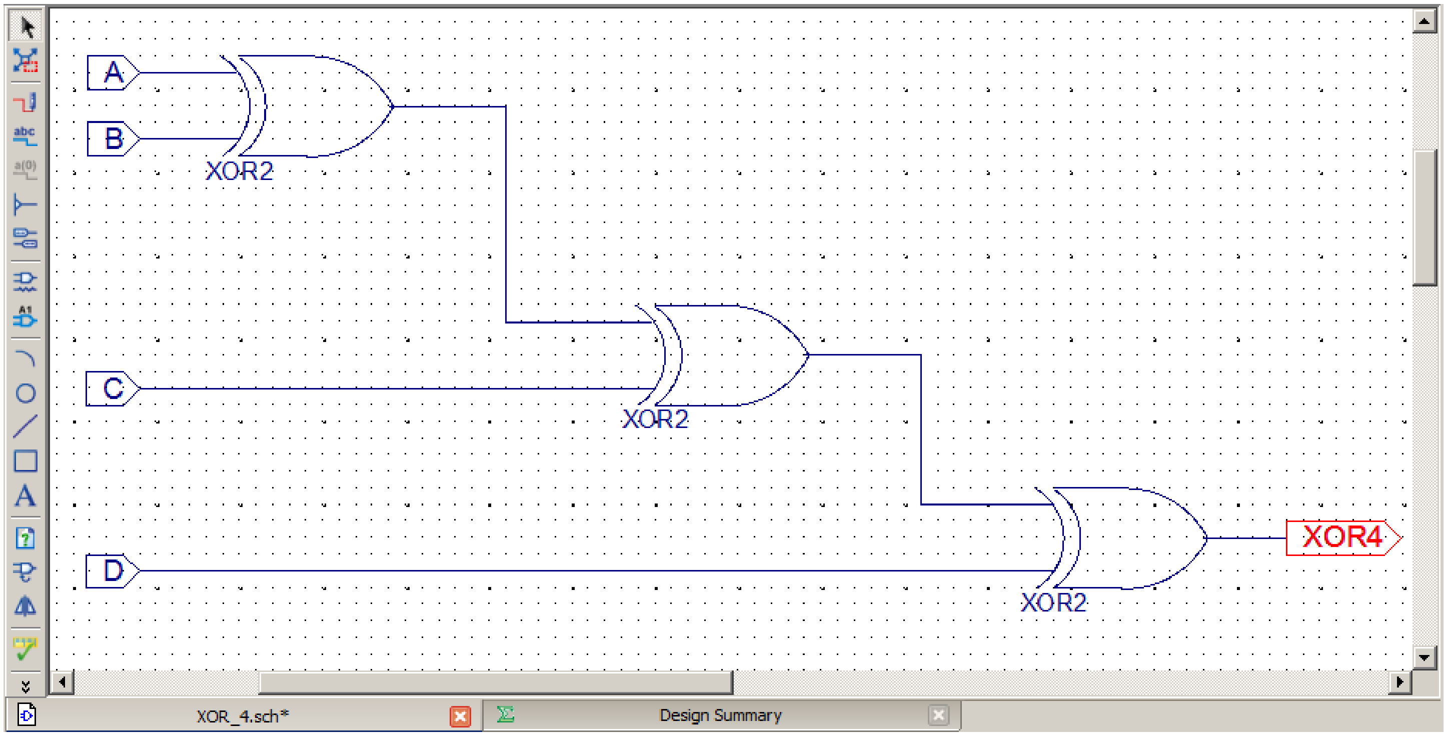


```
----------------------------------------------------------------------------------
library IEEE;
use IEEE.STD_LOGIC_1164.ALL;

-- Uncomment the following library declaration if using
-- arithmetic functions with Signed or Unsigned values
--use IEEE.NUMERIC_STD.ALL;

-- Uncomment the following library declaration if instantiating
-- any Xilinx primitives in this code.
--library UNISIM;
--use UNISIM.VComponents.all;

entity AND_2 is
    Port ( A : in  STD_LOGIC;
           B : in  STD_LOGIC;
           OUTPUT : out  STD_LOGIC);
end AND_2;

architecture Behavioral of AND_2 is

begin

OUTPUT <= A and B;

end Behavioral;
```

## Lab Procedure 4: Test your Knowledge

The more you practice, the more you will be proficient with the software.

Build the following circuit (4-input XOR) using three 2-input XOR gates.



Create a test bench. Make the following changes as highlighted, and then run the test bench.

```
SIGNAL D  :        STD_LOGIC:='0';
SIGNAL C  :        STD_LOGIC:='0';
SIGNAL A  :        STD_LOGIC:='0';
SIGNAL B  :        STD_LOGIC:='0';
SIGNAL XOR4     :        STD_LOGIC;

-- *** Test Bench - User Defined Section ***
~~   tb : PROCESS~~
~~   BEGIN~~
~~      WAIT; -- will wait forever~~
~~   END PROCESS;~~
        A_process : PROCESS
        BEGIN
                A <= NOT A;
                WAIT FOR 25 ns;
        END PROCESS;
        B_process: PROCESS
        BEGIN
                B <= NOT B;
                WAIT FOR 50 ns;
        END PROCESS;

        C_process: PROCESS
        BEGIN
                C <= NOT C;
                WAIT FOR 75 ns;
        END PROCESS;

        D_process: PROCESS
        BEGIN
                D <= NOT D;
                WAIT FOR 100 ns;
        END PROCESS;
-- *** End Test Bench - User Defined Section ***

END;
```