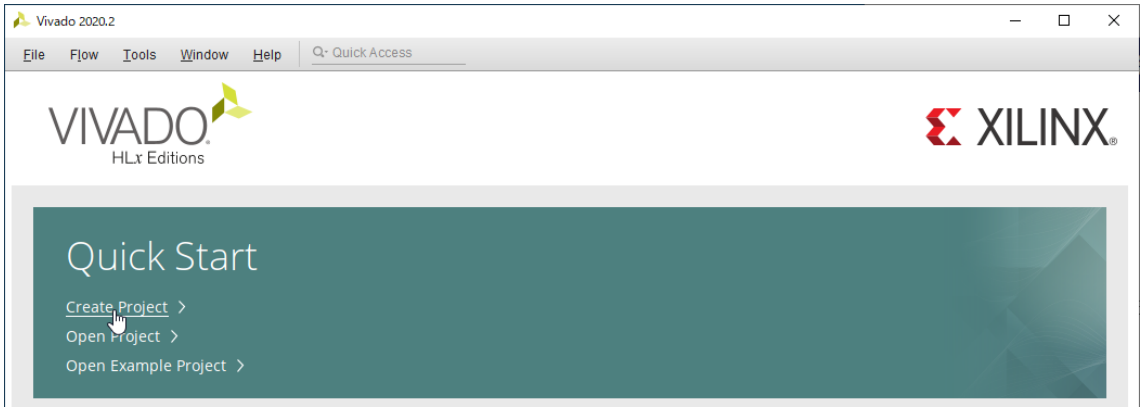
Vivado Design Suite Tutorial

Introduction

This tutorial guides you through the design flow using Xilinx Vivado software to create a simple digital circuit using Verilog HDL. You will go through the typical design flow targeting the Artix-7 based BASYS3 board.

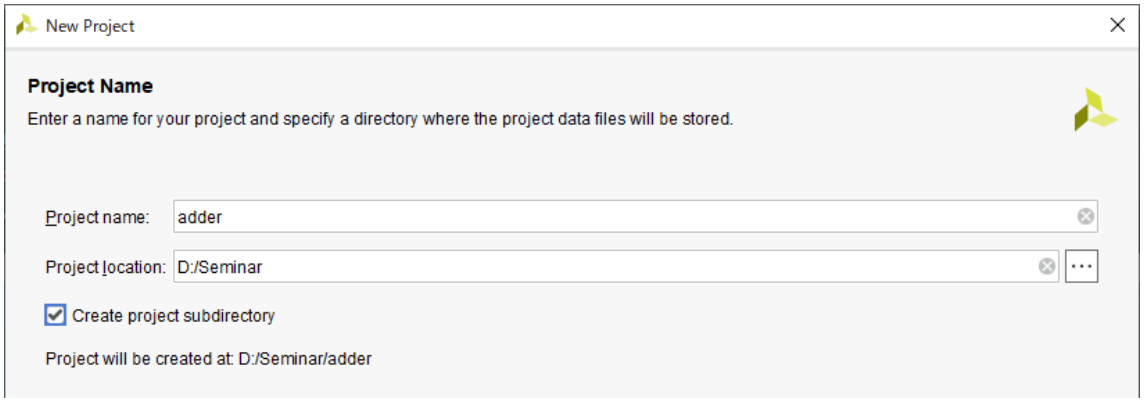
1. Creating a Vivado Project

1.1) Open up Vavado 2020.2 (not Vivado HLS) and click **Create Project** to start the wizard.

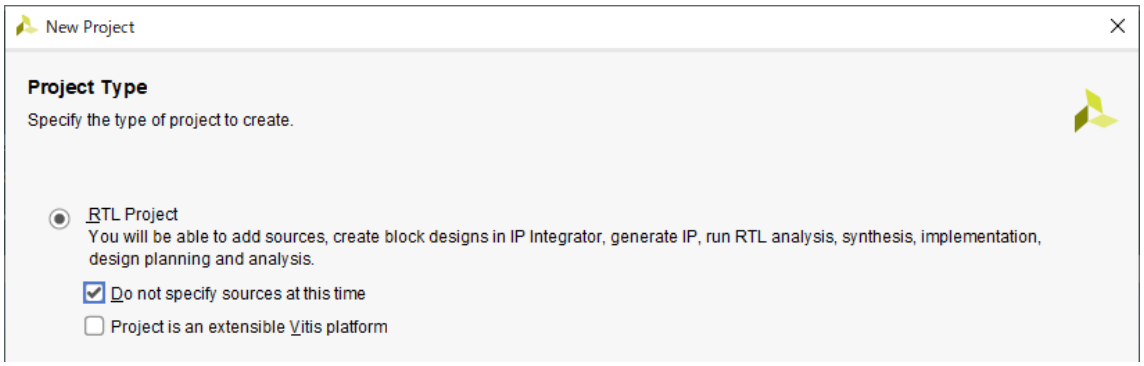


1.2) You will see the "Create a New Vivado Project" dialog box. Click **Next**.

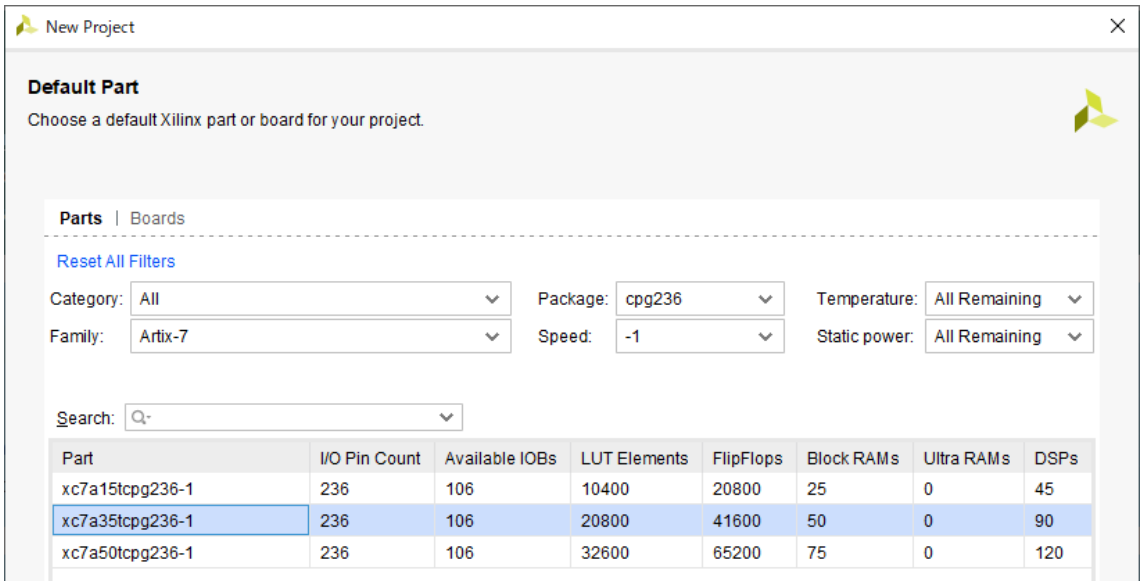
1.3) Enter **adder** in the "Project name" field. Click the browse button of the "Project location" field, browse to your project saving directory, and click **Select**. Make sure that the "Create Project Subdirectory" box is checked. Click **Next**.



1.4) Select **RTL Project** option in the "Project Type" form and check the "Do not specify sources at this time" box. Click **Next**.



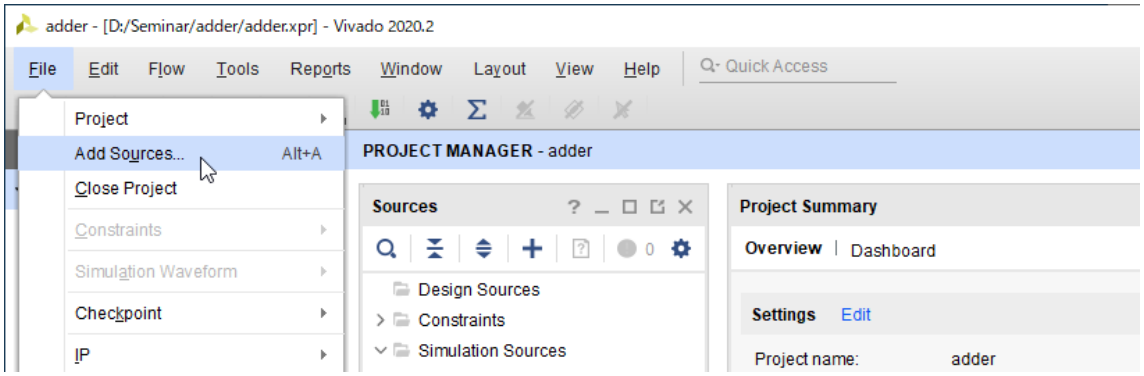
1.5) In the "Default Part" form, using the **Parts** option and various drop-down fields of the filter section, select the **xc7a35tcpg236-1** part. Click **Next**.



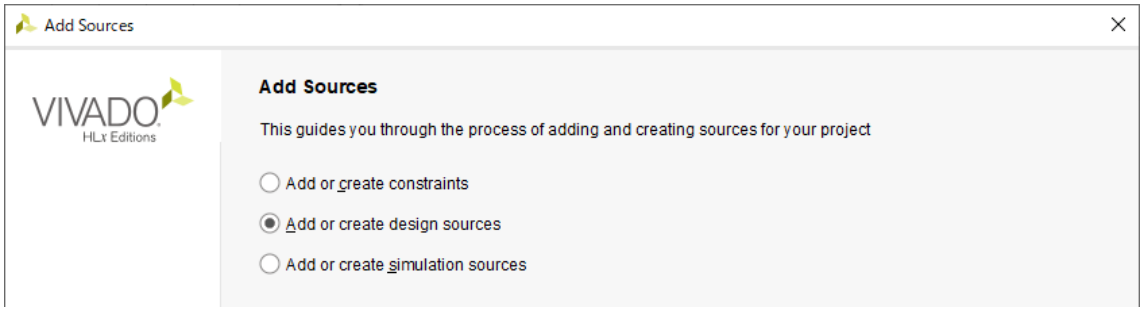
1.6) You will see "New Project Summary" form. Click **Finish** to create the project.

2. Creating a Source File

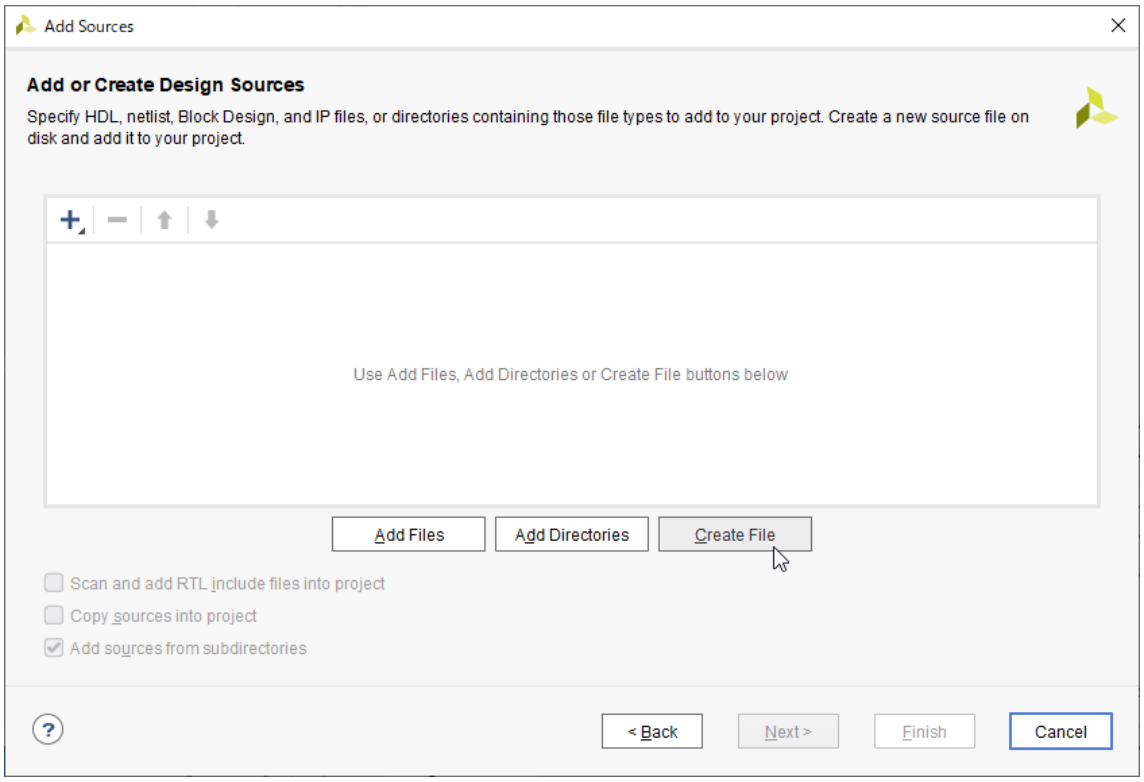
2.1) Select **Add Sources** from the File menu.



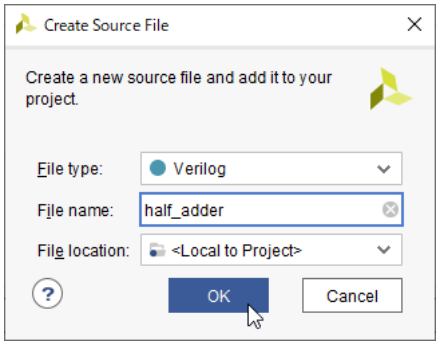
2.2) In the “Add Sources” form, select **Add or create design sources**. Click **Next**.



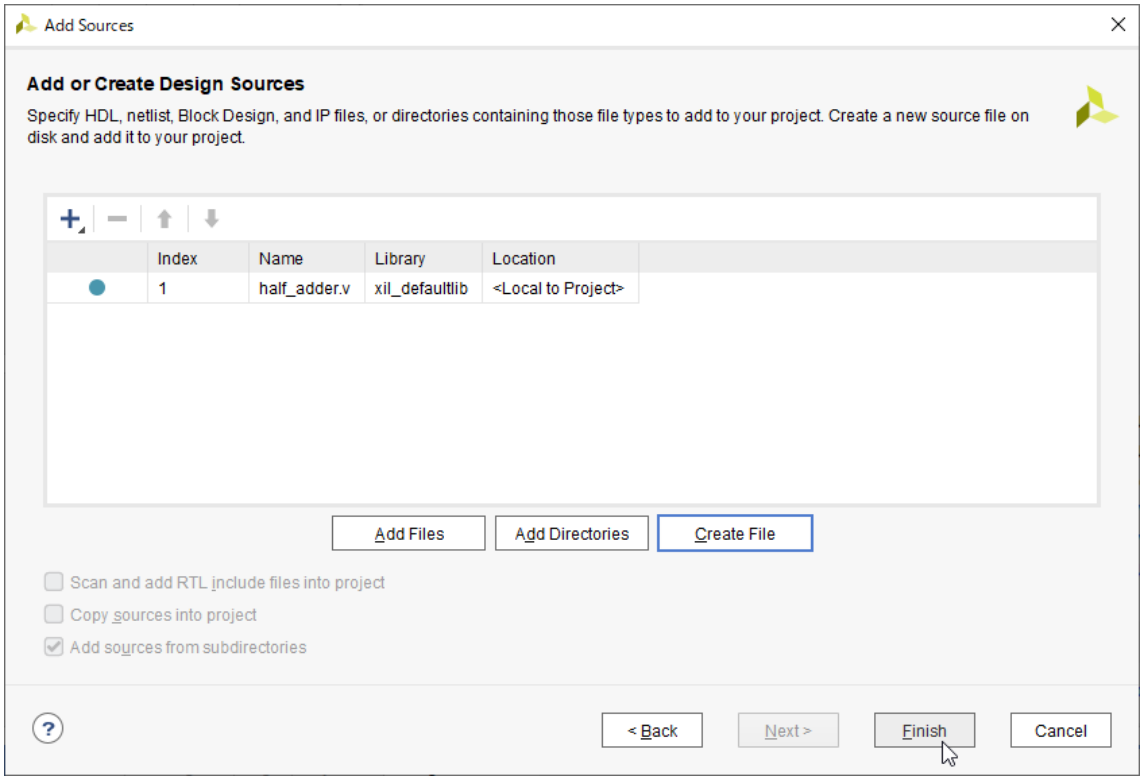
2.3) In the “Add or Create Design Sources” form, click **Create File** button.



2.4) In the “Create Source File” form, select **Verilog** in the “File type” field. Enter **half_adder** in the “File name” field. Click **OK**.



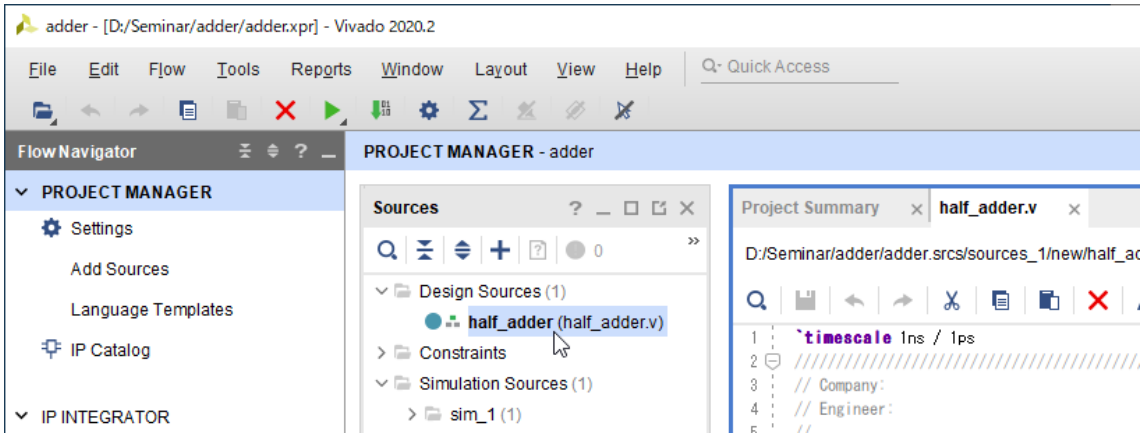
2.5) You will see the created file in the “Add or Create Design Sources” form. Click **Finish**.



2.6) Skip the “Module Definition” form by clicking **OK**.

2.7) You will see the “Define Module” dialog box. Click **Yes**.

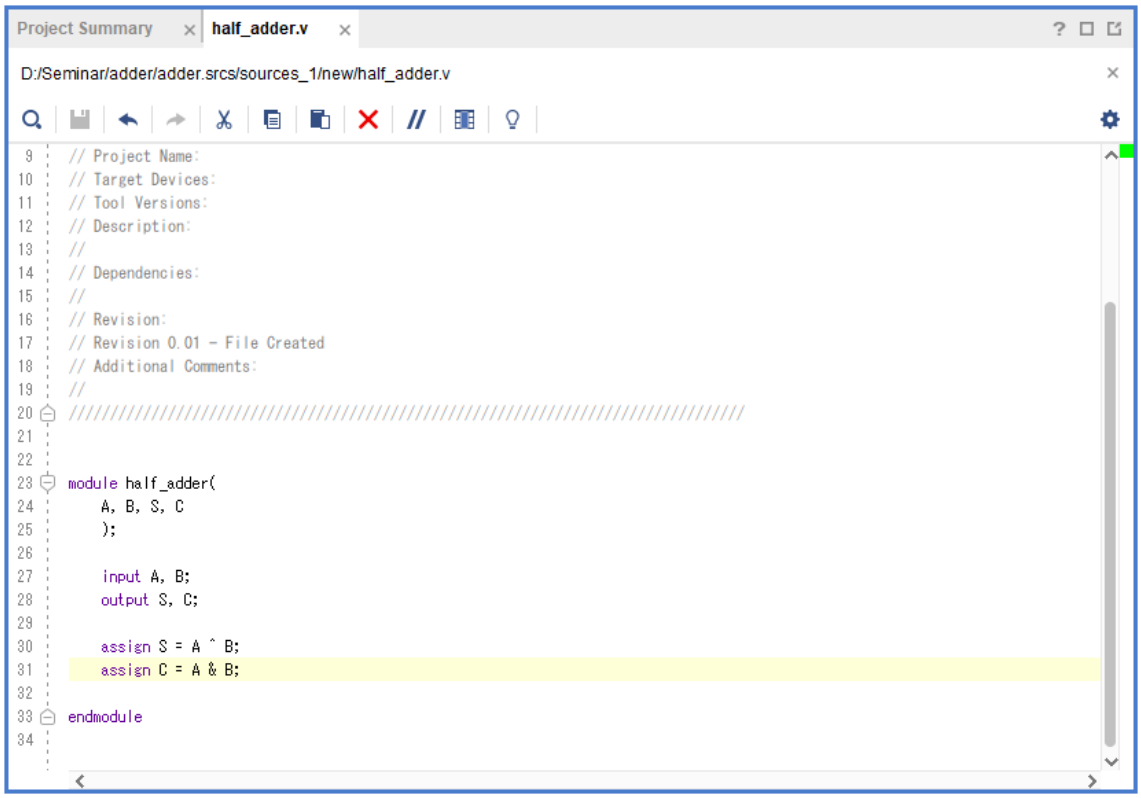
2.8) The created file will be added to the “Design Sources” folder in the “Sources” pane of the “PROJECT MANAGER”.



3. Verilog Coding

3.1) Double click on the **half_adder.v** file in the “Sources” pane to open it.

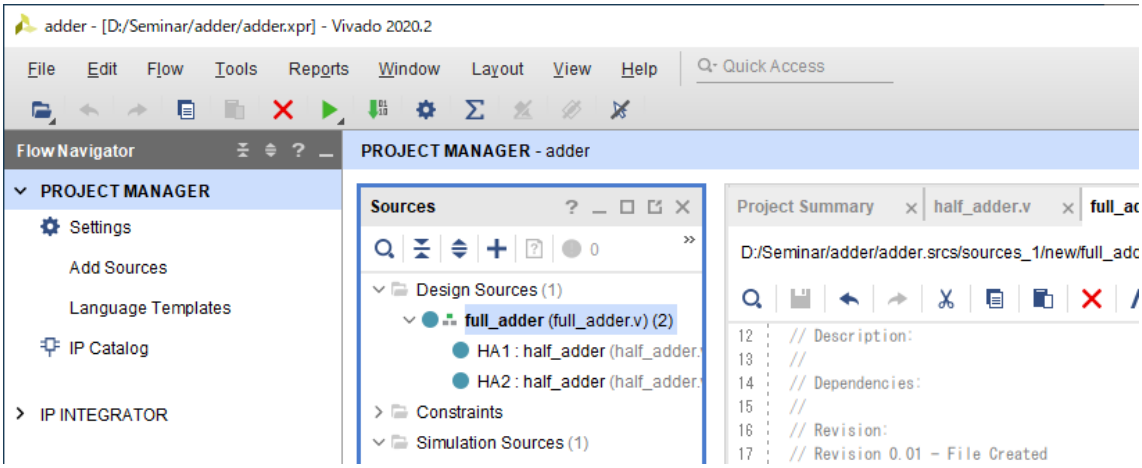
3.2) Write the half adder Verilog code into the text editor.



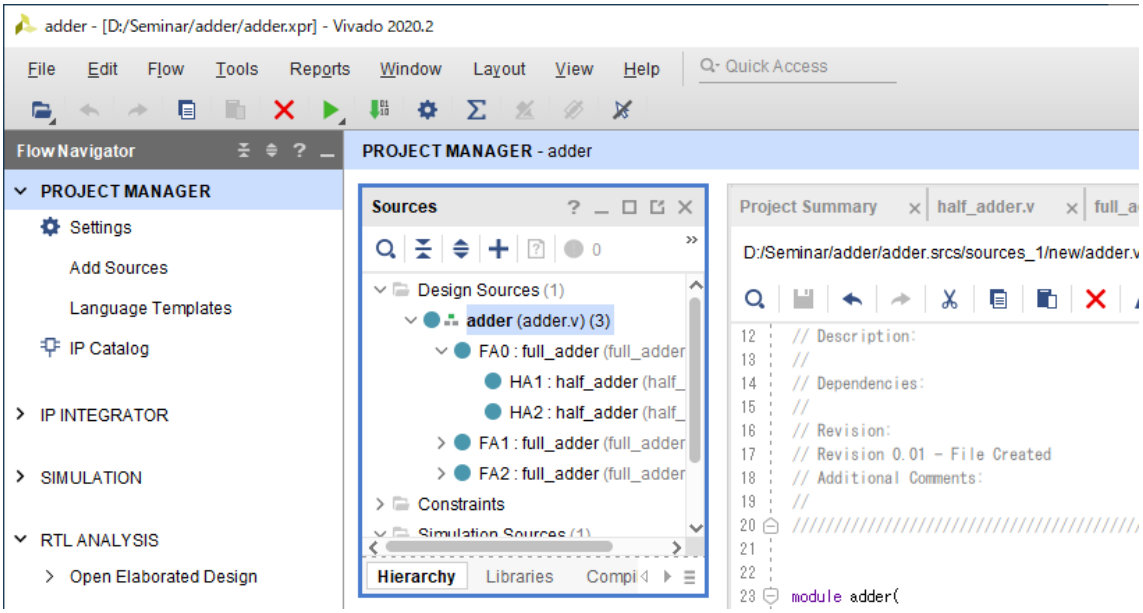
3.3) Ctrl + S to save the file.

3.4) Create and add the **full_adder.v** file into the project and write the full adder Verilog code.

3.5) Vivado automatically updates the hierarchy of the modules and instances and detects the top module of the project. The top module defines the hierarchy of the design for synthesis and implementation.

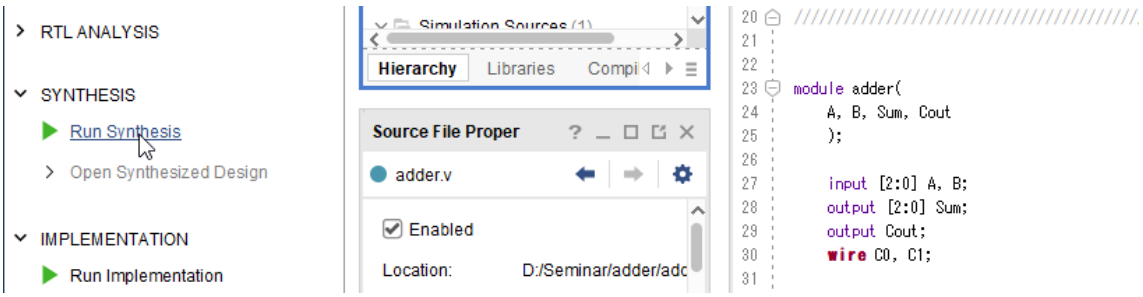


3.6) Create and add the **adder.v** file into the project and write the 3-bit adder Verilog code. It will be the top module of the project.

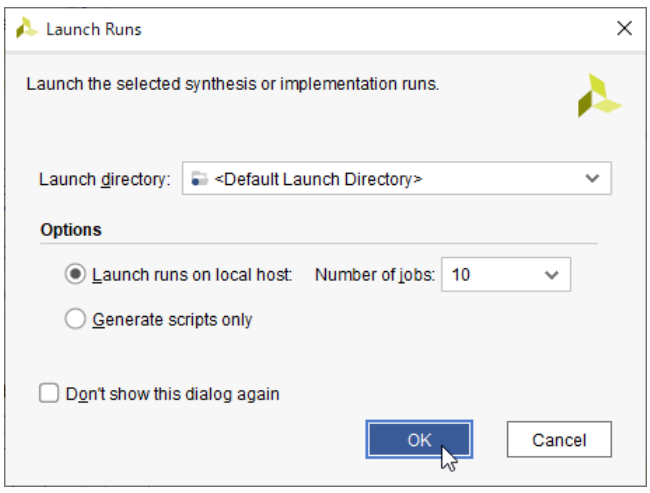


4. Synthesis

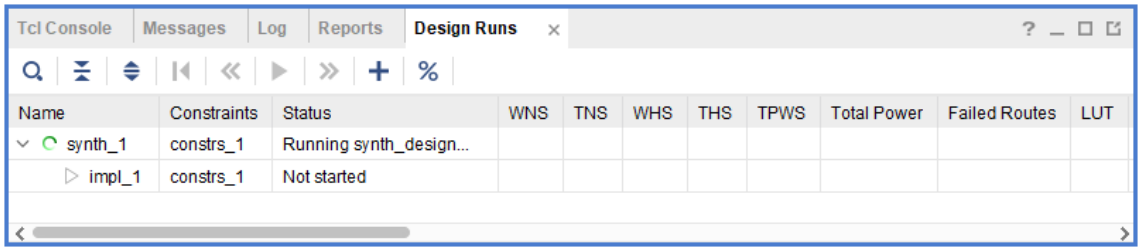
4.1) Click **Run Synthesis** in the "Flow Navigator".



4.2) You will see the "Launch Runs" form. Click **OK**.

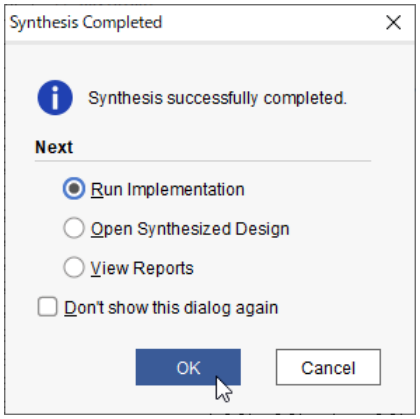


4.3) You can see the run status and information in the “Design Runs” tab.



5. Implementation

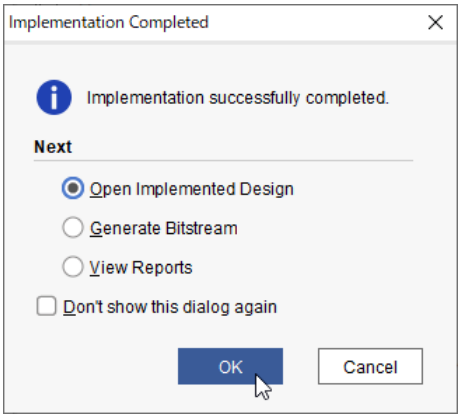
5.1) Upon completion of the synthesis process, you will see the “Synthesis Completed” dialogue. Select **Run Implementation** and click **OK**.



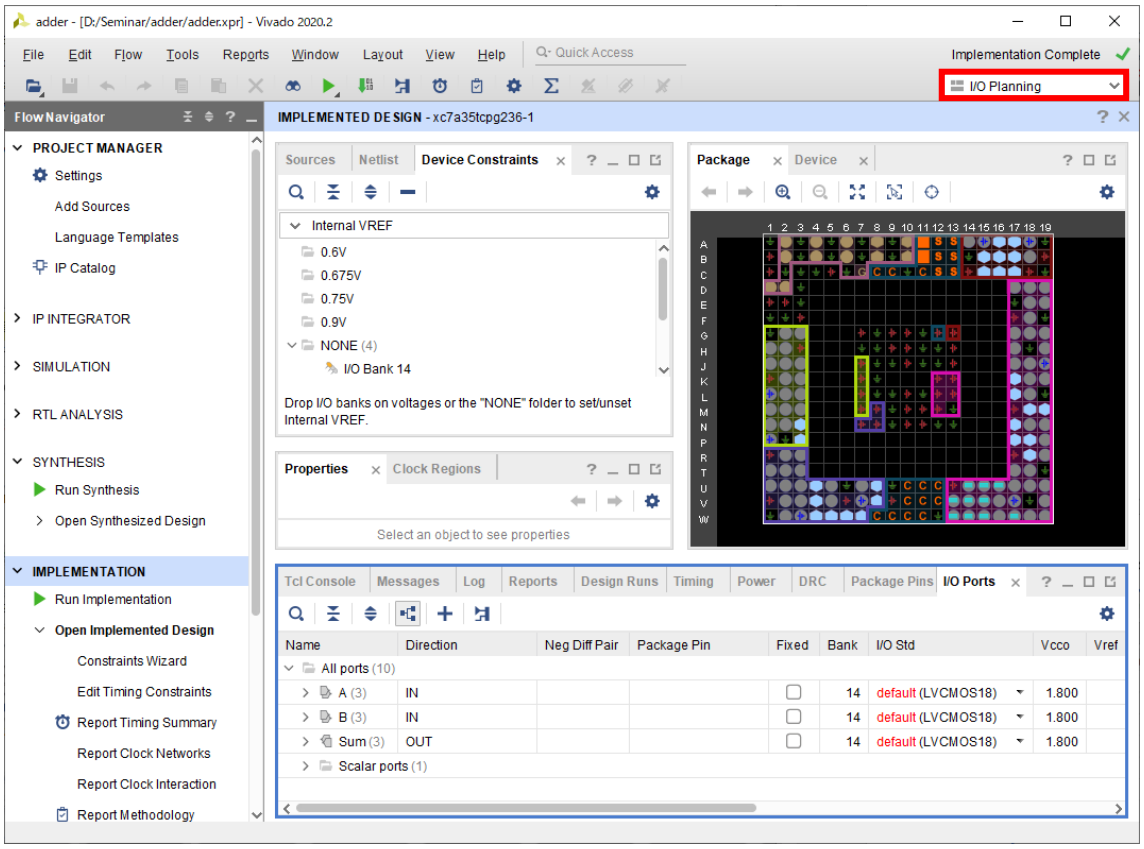
5.2) You will see the “Launch Runs” form. Click **OK**.

6. Pin Assignment

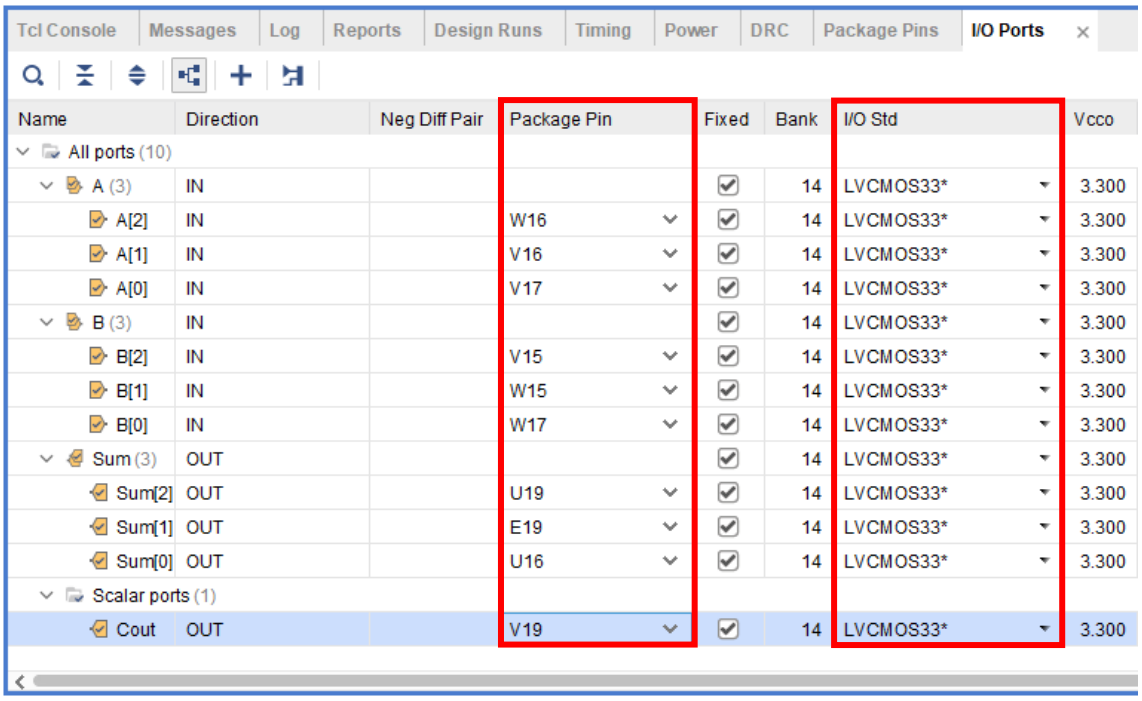
6.1) Upon completion of the implementation process, you will see the “Implementation Completed” dialogue. Select **Open Implemented Design** and click **OK**.

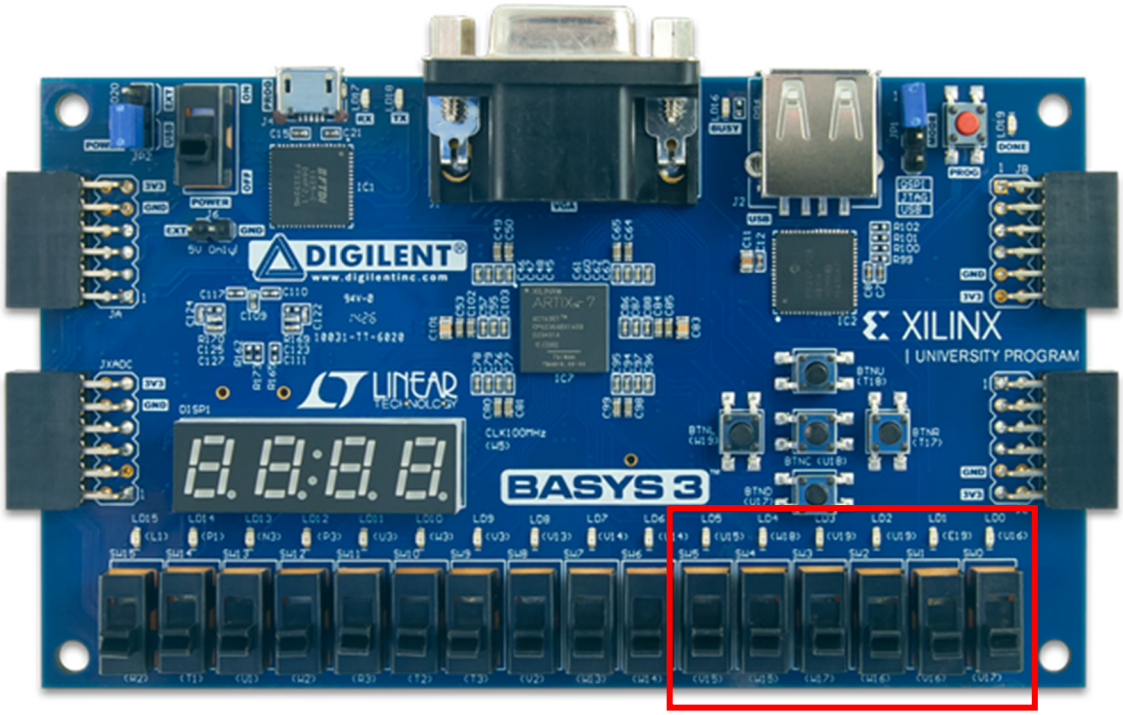


6.2) To launch the I/O Planning view layout, select **I/O planning** in the **Layout** menu.

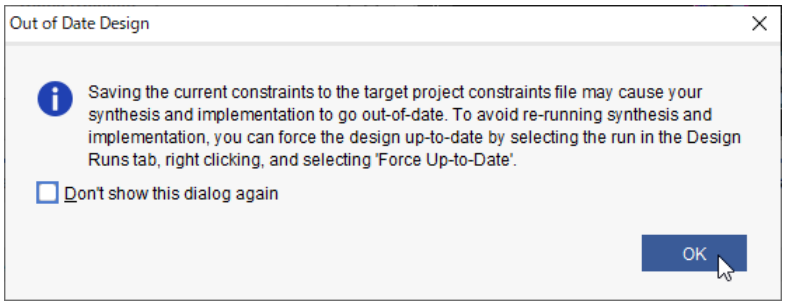


6.3) In the "I/O Ports" tab, set the **Package Pin** and **I/O Std** for the BASYS3 board.

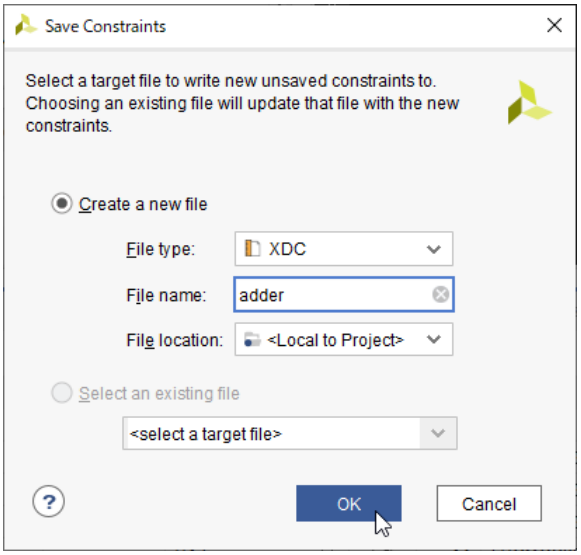




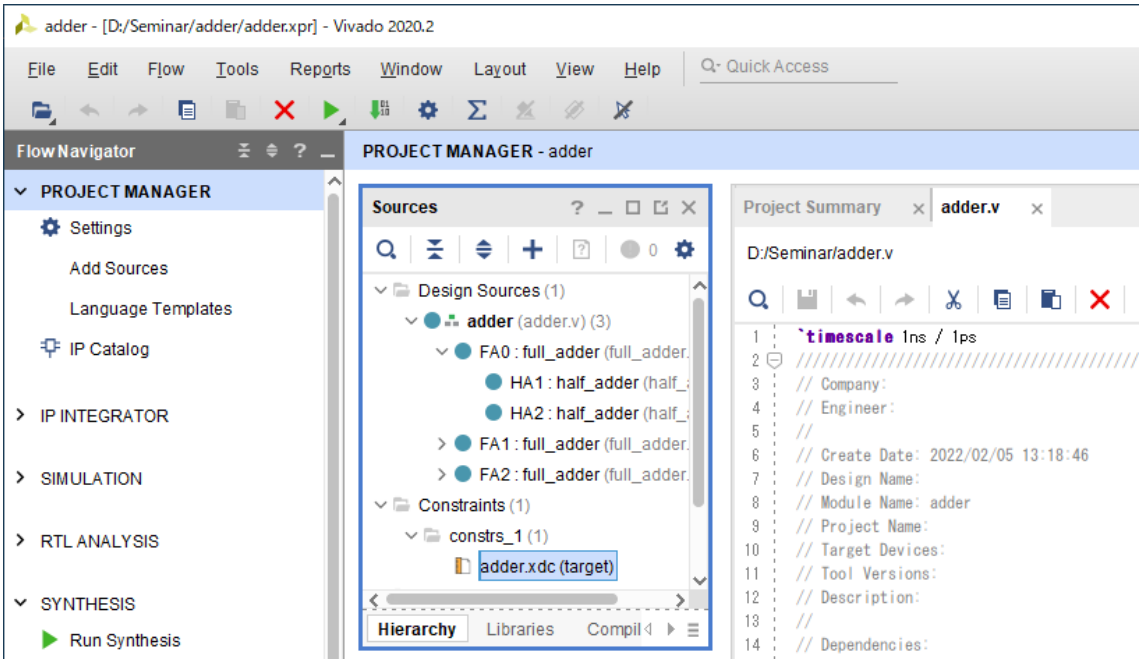
6.3) Ctrl + S to save the pin assignment. You will see the “Out of Date Design” dialog. Click **OK**.



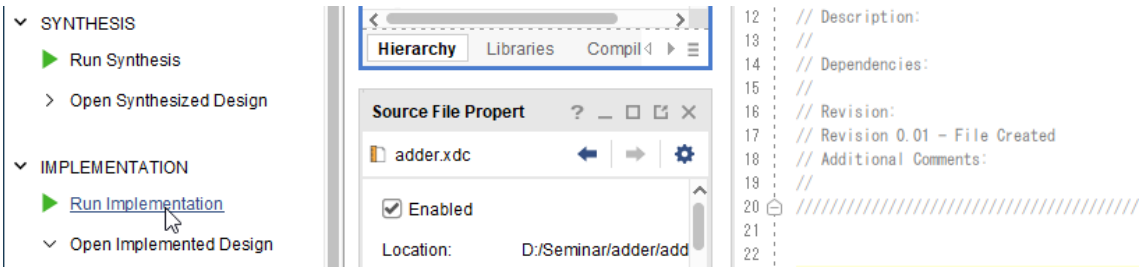
6.4) You will see the “Save Constraints” form. Enter **adder** in the “File name” field and click **OK**.



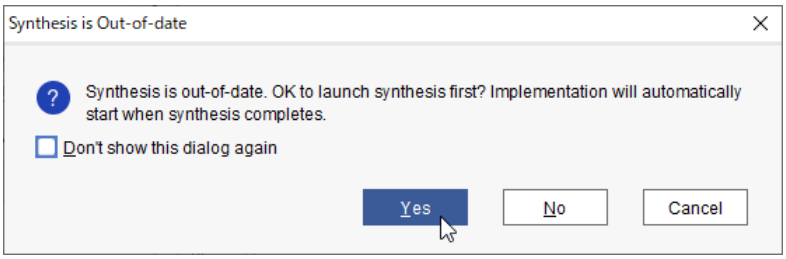
6.5) The constraint file **adder.xdc** will be added to the “Constraints” folder in the “Sources” pane of the “PROJECT MANAGER”.



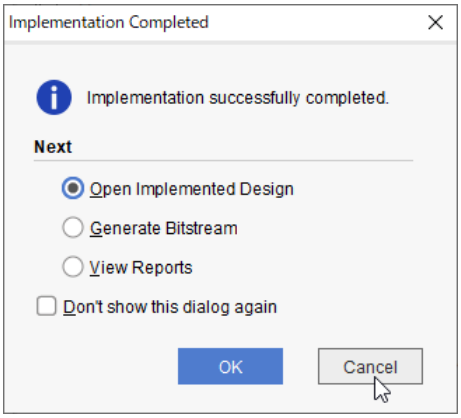
6.6) Click **Run Implementation** in the “Flow Navigator” to rearrange the layout.



6.7) You will see the “Synthesis is Out-of-date” dialogue. Click **Yes**.

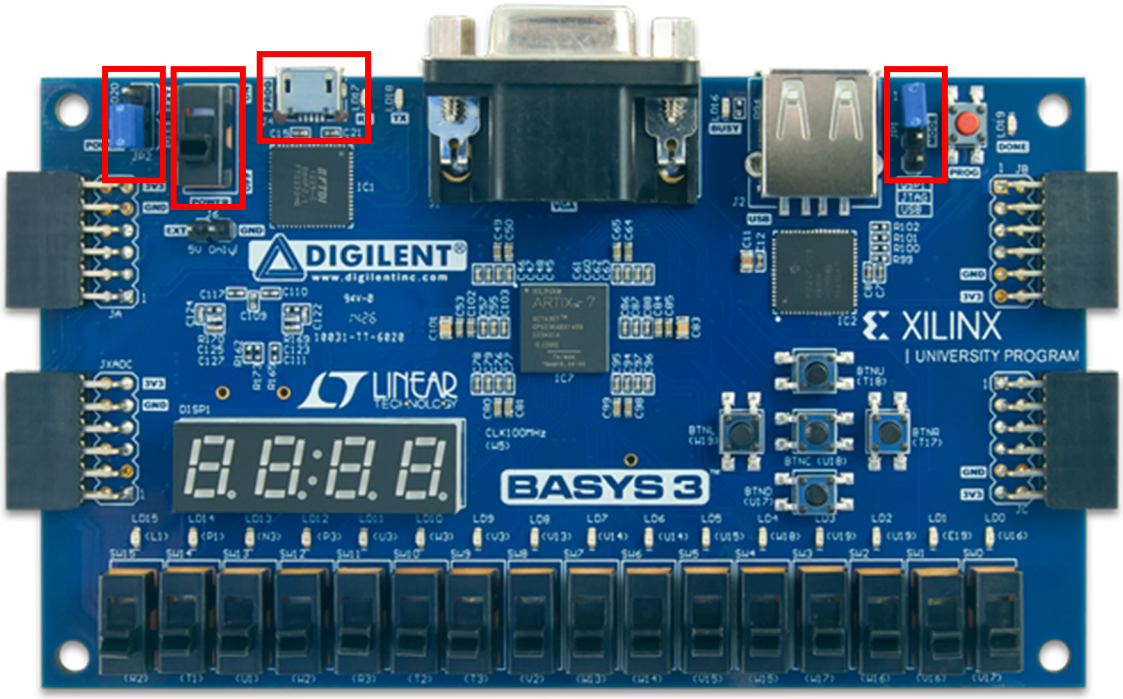


6.8) Upon completion of the implementation process, you will see the “Implementation Completed” dialogue. Click **Cancel** to ignore.

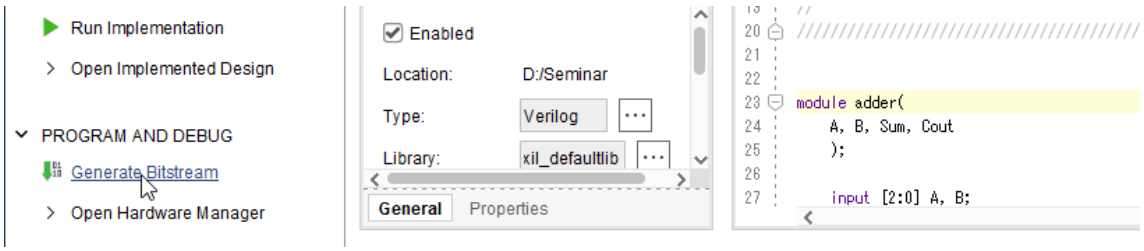


7. Programming FPGA

7.1) Make sure that **JP2** of the BASYS3 board is configured to use **USB** as power source. Furthermore, make sure **JP1** is configured to the **QSPI** mode. Insert a micro USB cable to **J4** and connect it to your PC. Finally, switch **SW16** to the **ON** position. You will see the demo design running.

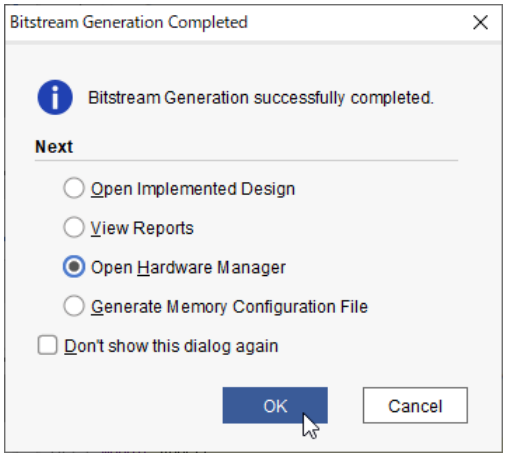


7.1) Click **Generating Bitstream** in the "PROGRAM AND DEBUG" section of the "Flow Navigator".

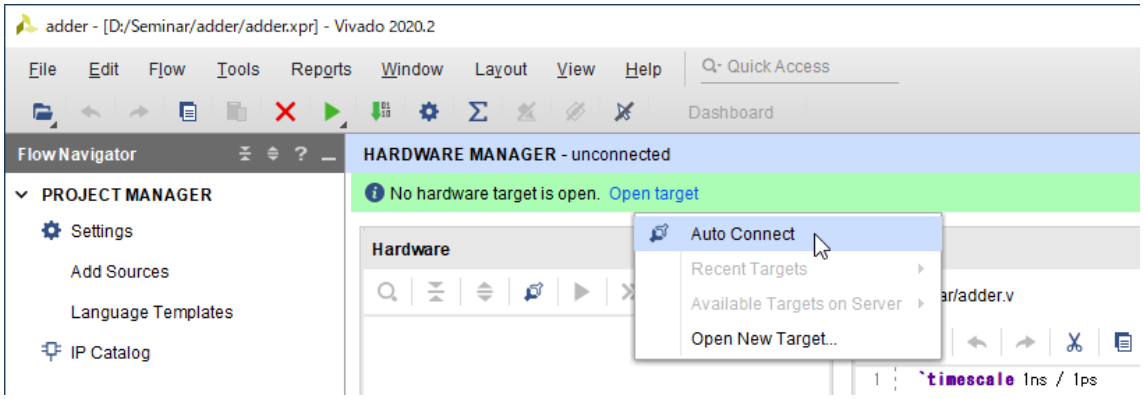


7.2) You will see the “Launch Runs” form. Click **OK**.

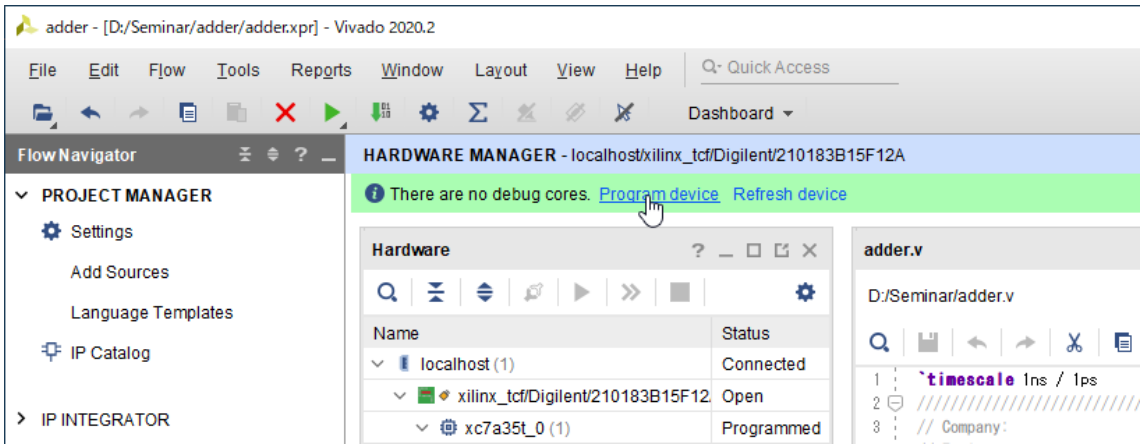
7.3) Upon completion of the bitstream generation, you will see the “Bitstream Generation Completed” dialogue. Select **Open Hardware Manager** and click **OK**.



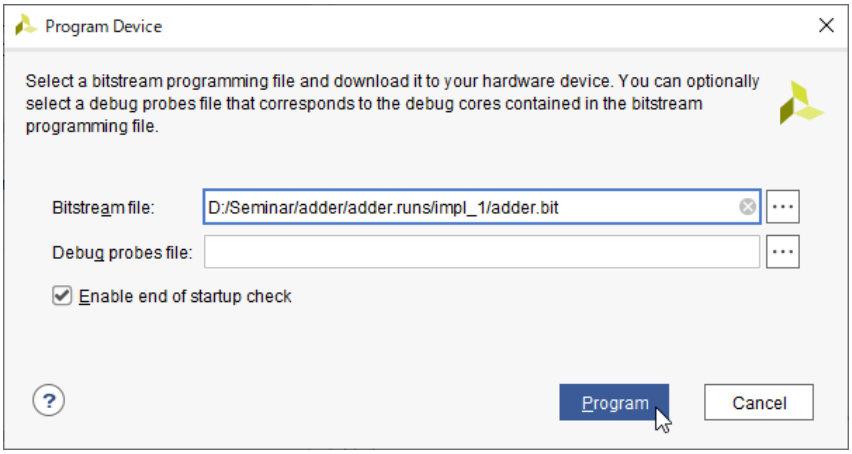
7.4) Click **Open target** in the “HARDWARE MANAGER” and select **Auto Connect**.



7.5) Click **Program device**.



7.6) You will see the “Program Device” form. Click **Program**.



7.7) The BASYS3 is now running your 3-bit adder!

7.8) After testing your design, press **PROG** button to refresh the device.