USER MANUAL

Wireless Communication Between Two Embedded System

A : Vivado Design Suite Installation

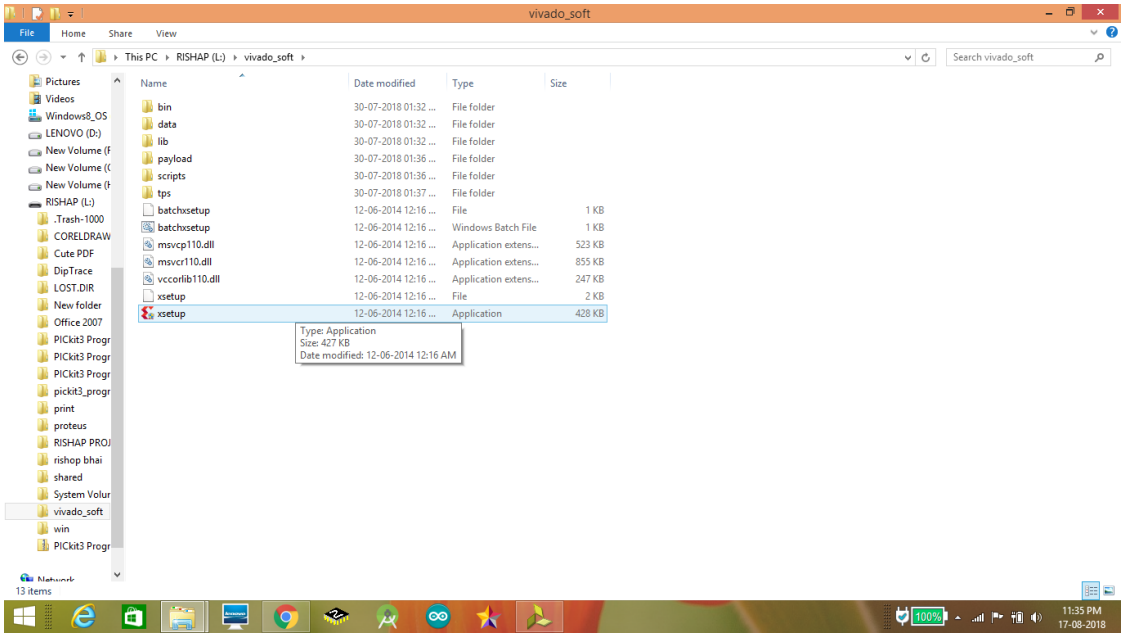
1: Download the Vivado design suite from the Xilinx website

(https://www.xilinx.com/support/download.html)

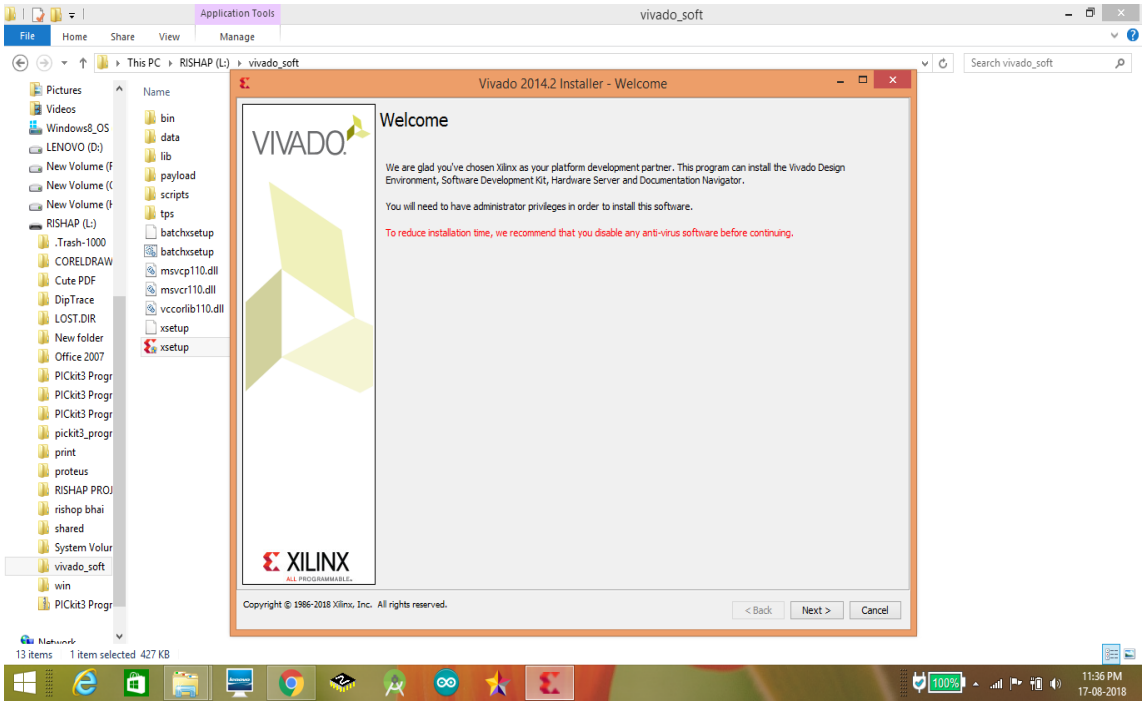
or

2: Use the DVD of Vivado design suite (provided with the ZEDboard)

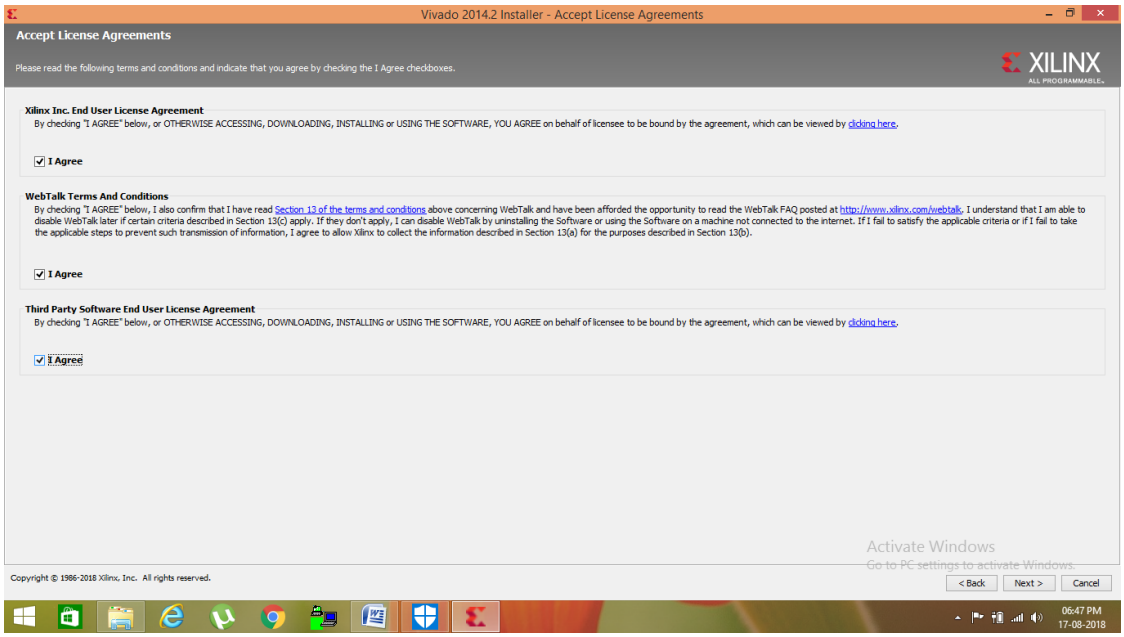
3: Click on Setup.exe for installation to run.



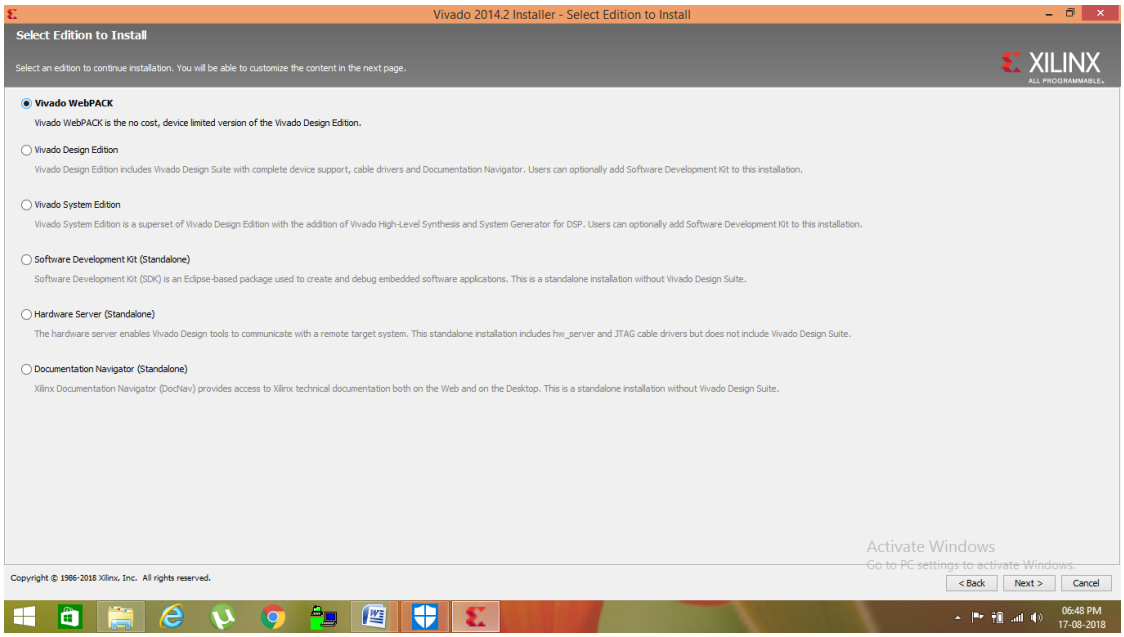
4: Close unwanted application to reduce installation time and click → Next.



5 : Check each box to accept the Licence Agreements and click → Next.

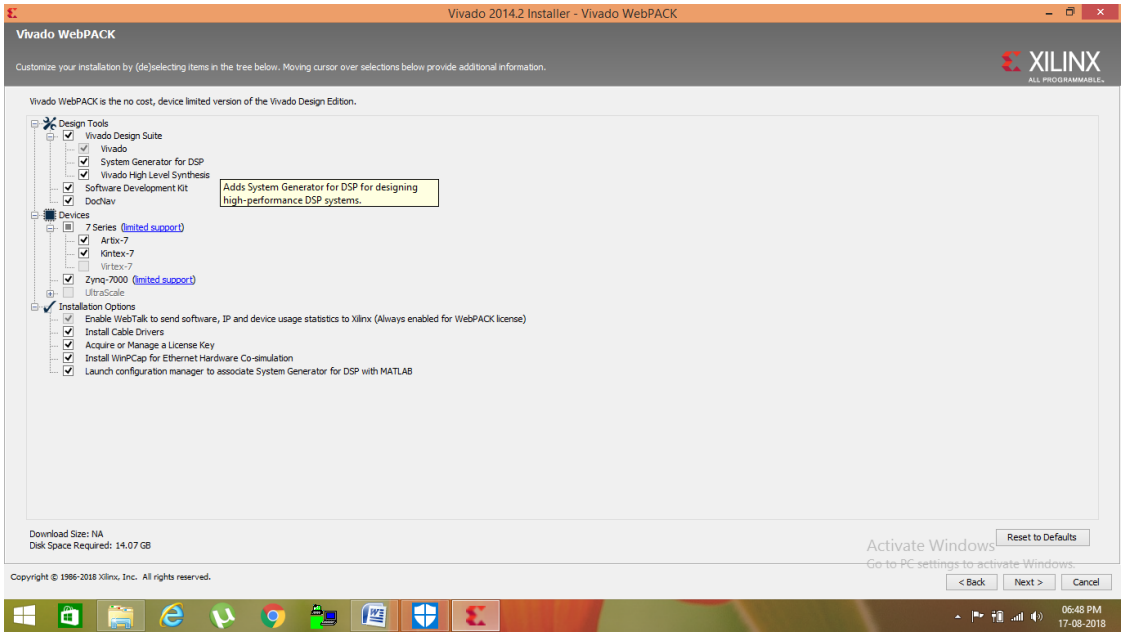


6 : Select Vivado WebPACK to use a free version of Vivado and click → Next.

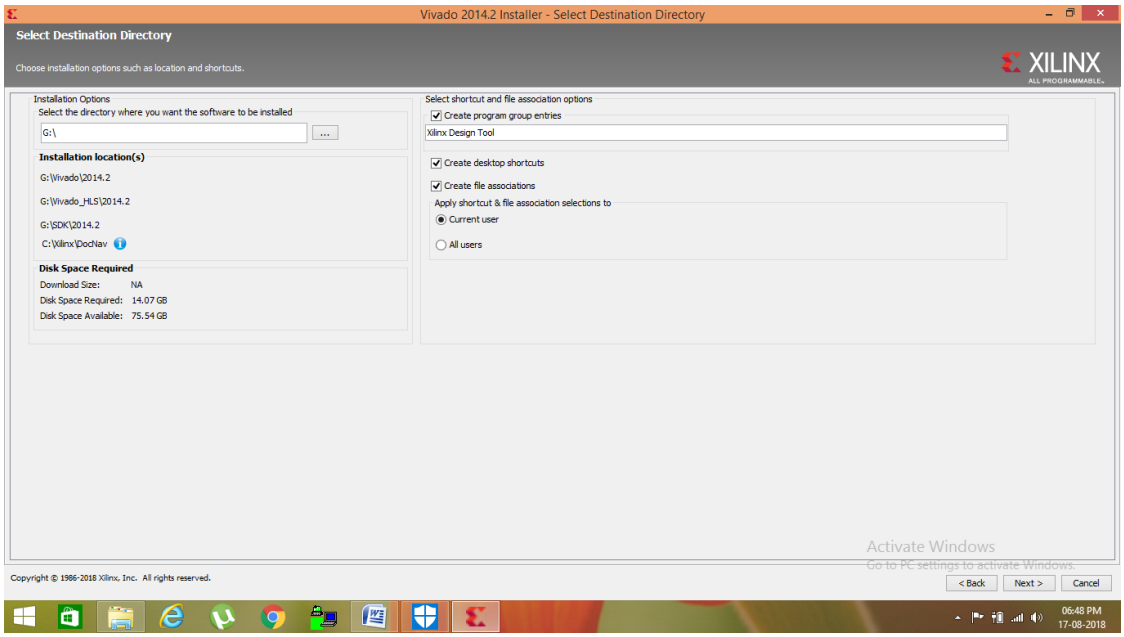


7 : Check the all the option available. (SDK and ZYNQ7000 is mandatory)

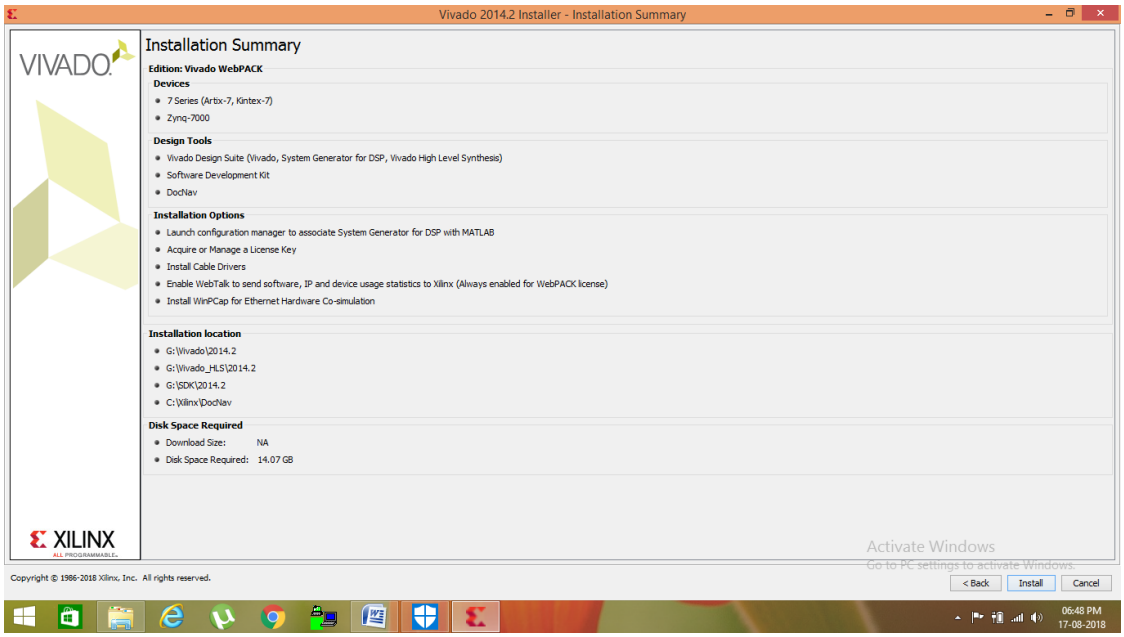
and click → Next.



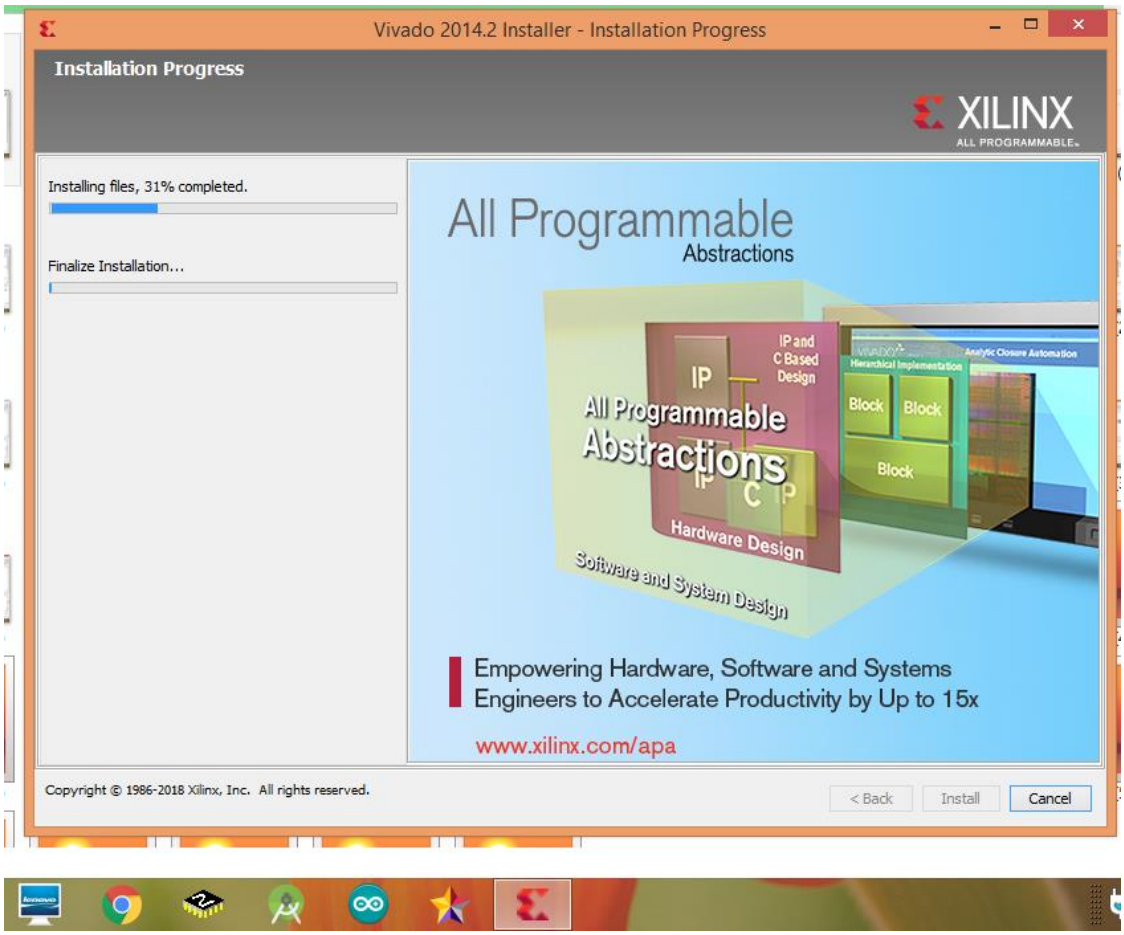
8 : Browse the installation path or keep it default. and click → Next.



9 : Check the summary as per selection to verify. and click → Install.

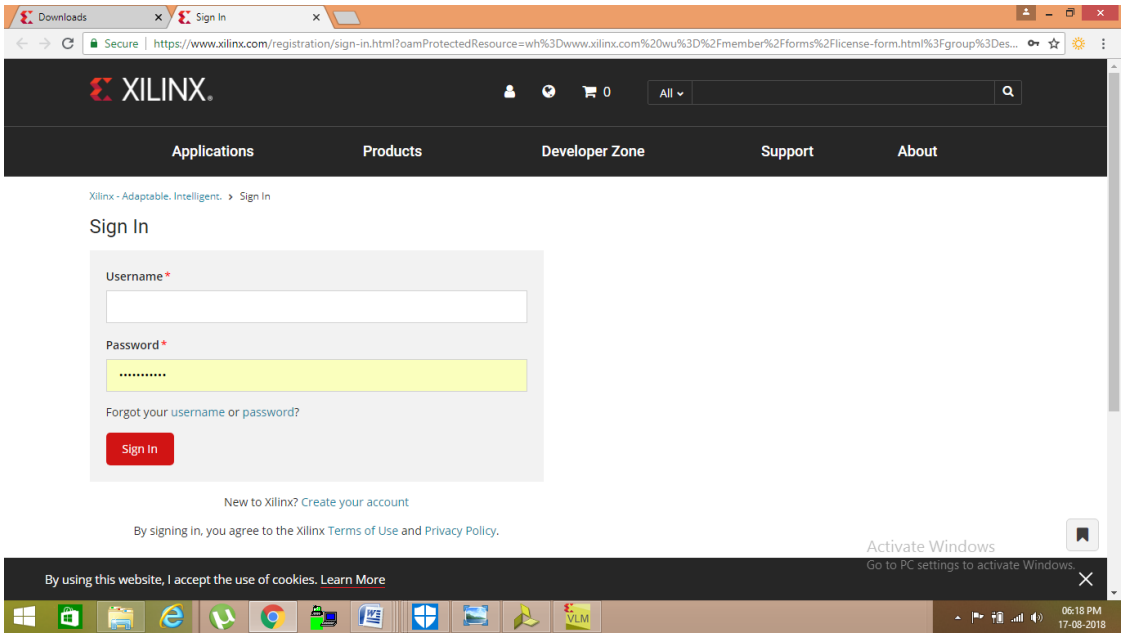


10 : Accept all prompts while installing for cable drivers, etc.

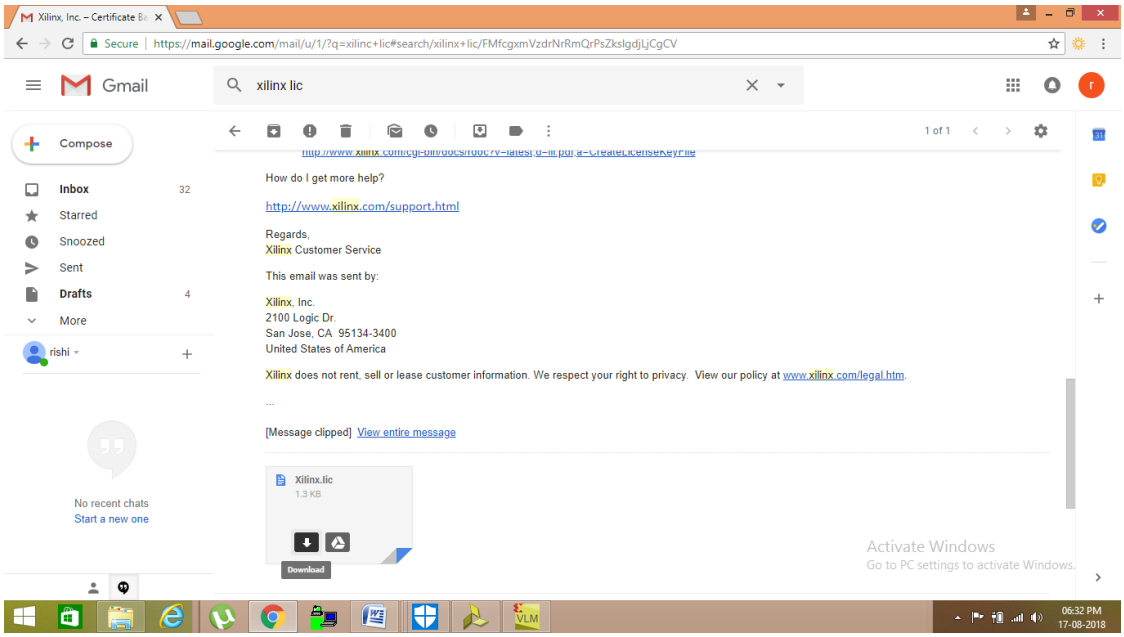


11 : TO obtain the licence. Click on →Obtain Licence and select → Get free licences. and click on → Connect Now

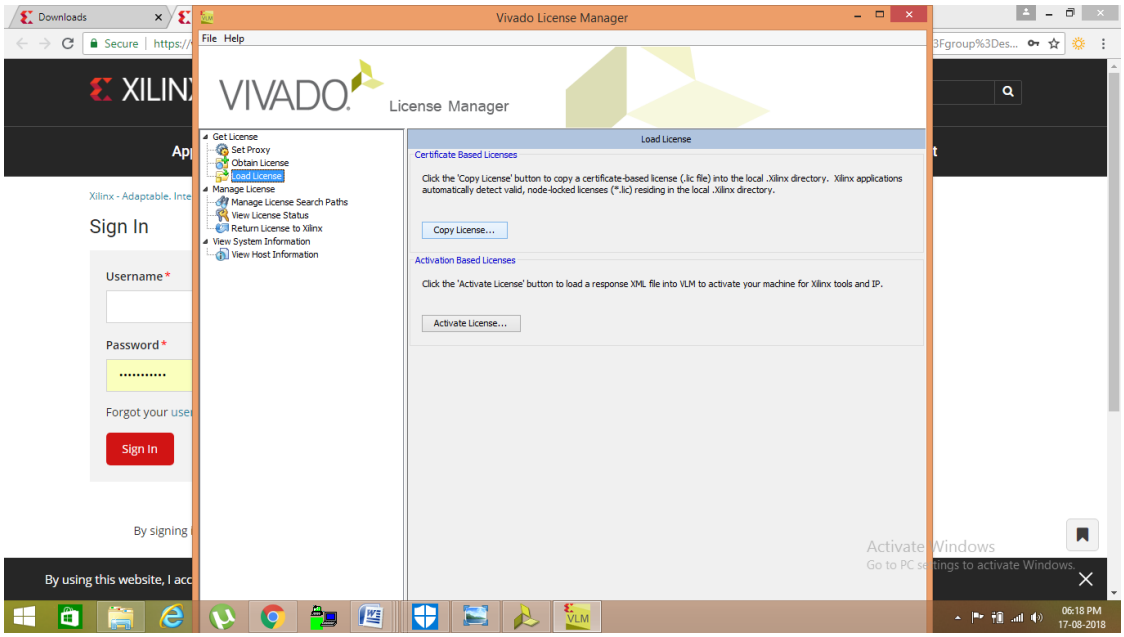
12 : Sign in or Create a account, to obtain a licence on the registered email ID



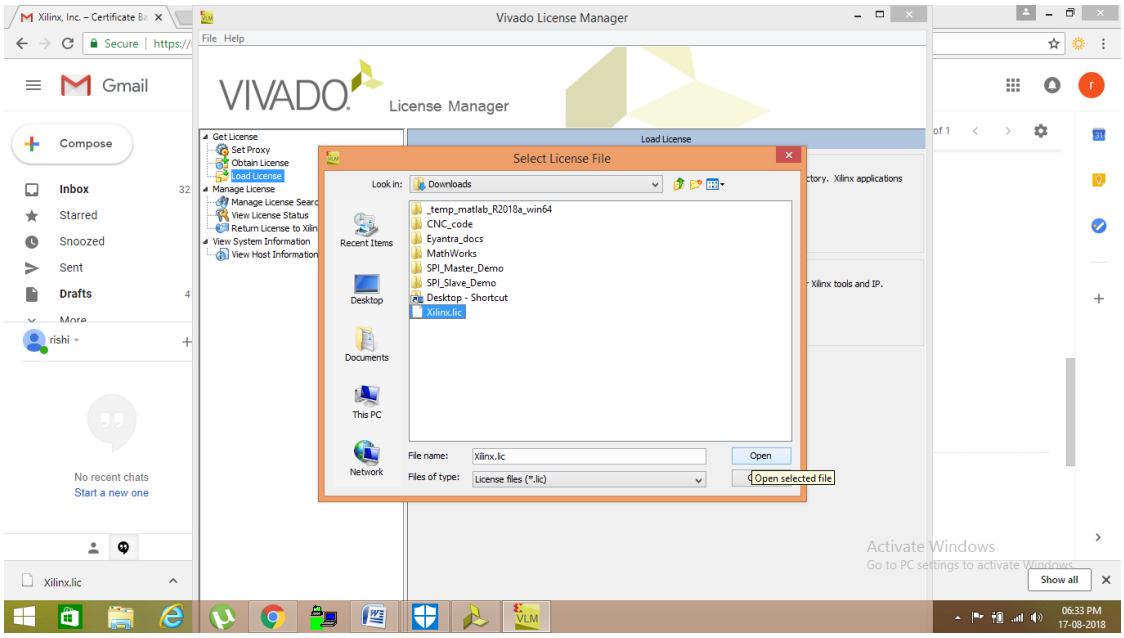
13 : Download the Xilinx.lic licence file from the email received by Xilinx.



14 : To load the licence. Click on →Load Licence and click on → Copy Licence.

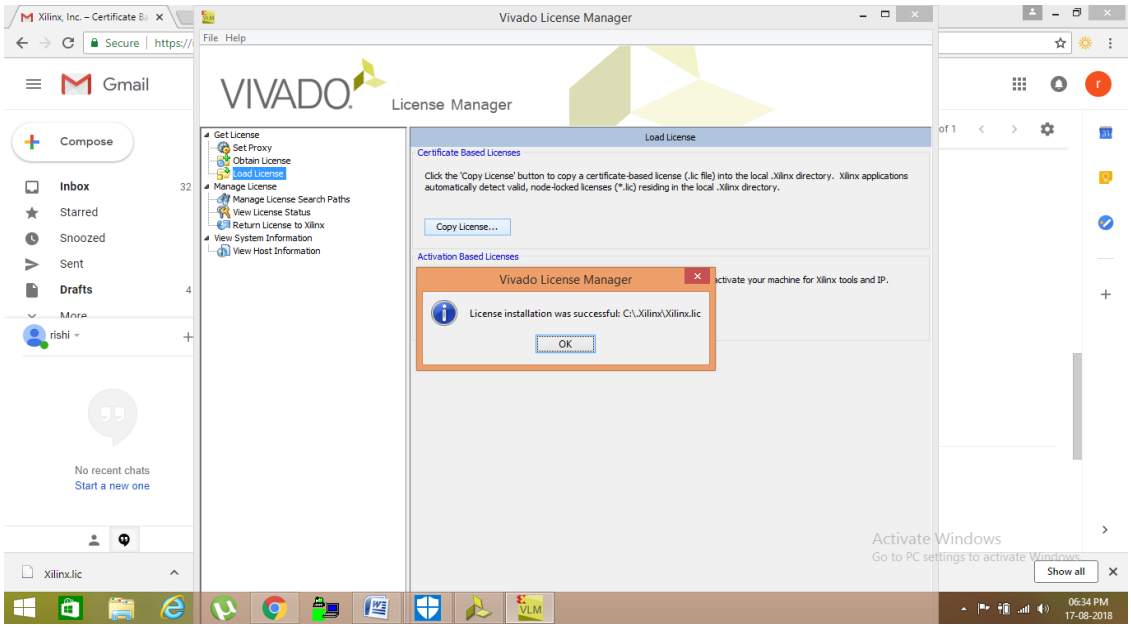


15 : Browse to the Downloads folder and select Xilinx.lic and open.



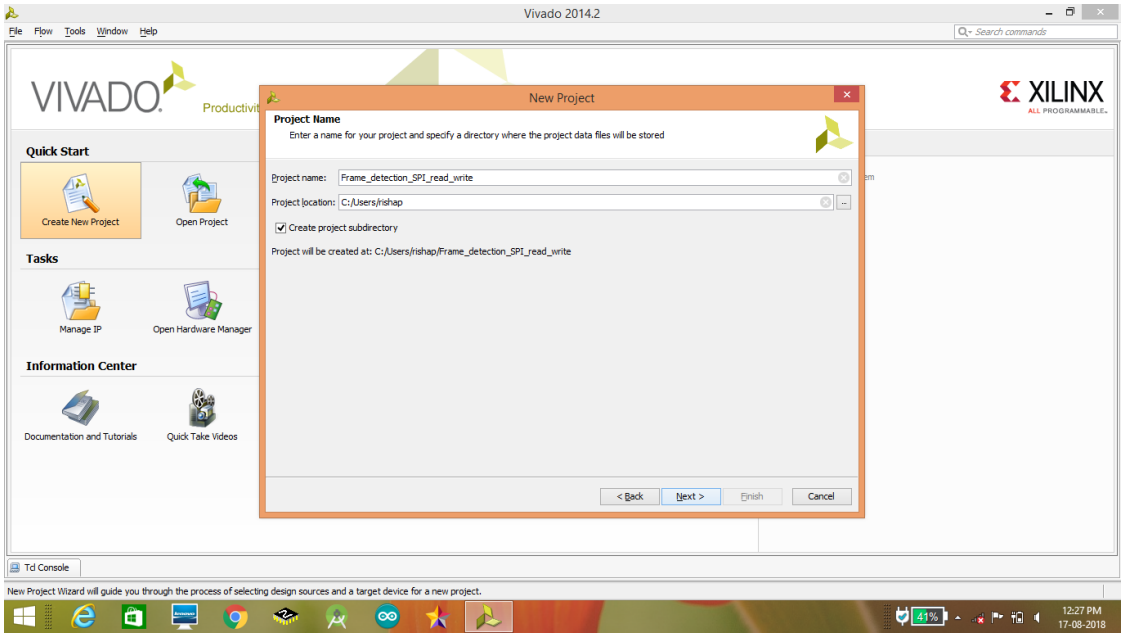
16 : Success message show licence installation was successful.

Now Vivado is ready to use.



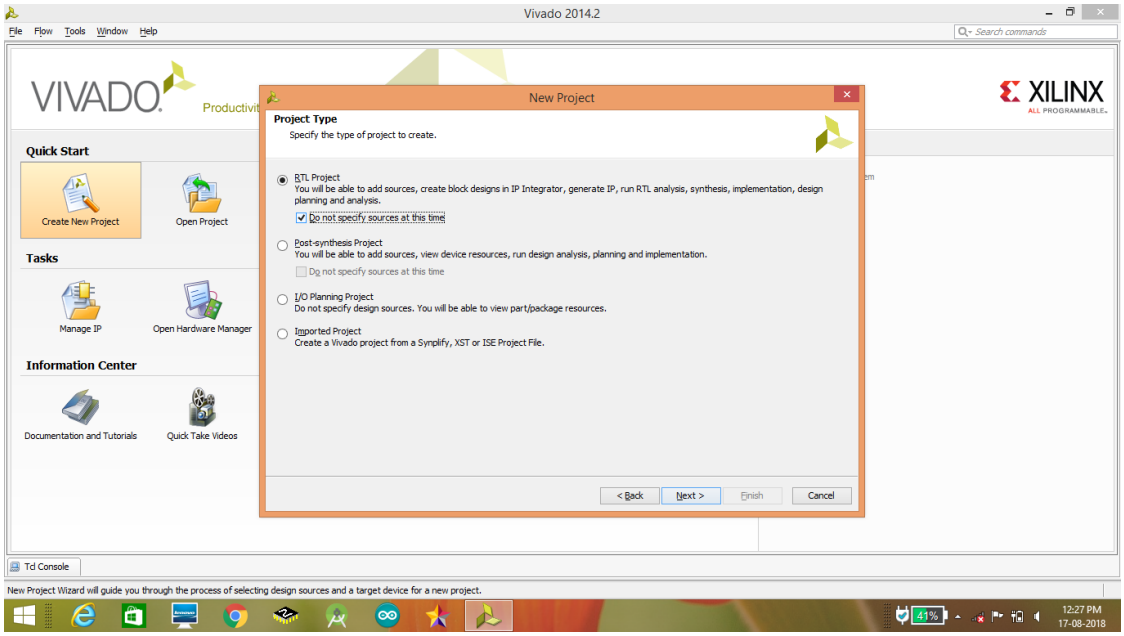
B : Steps to Create the Hardware Definition File in Vivado.

1 : Open Vivado and select → New project and Name the project.

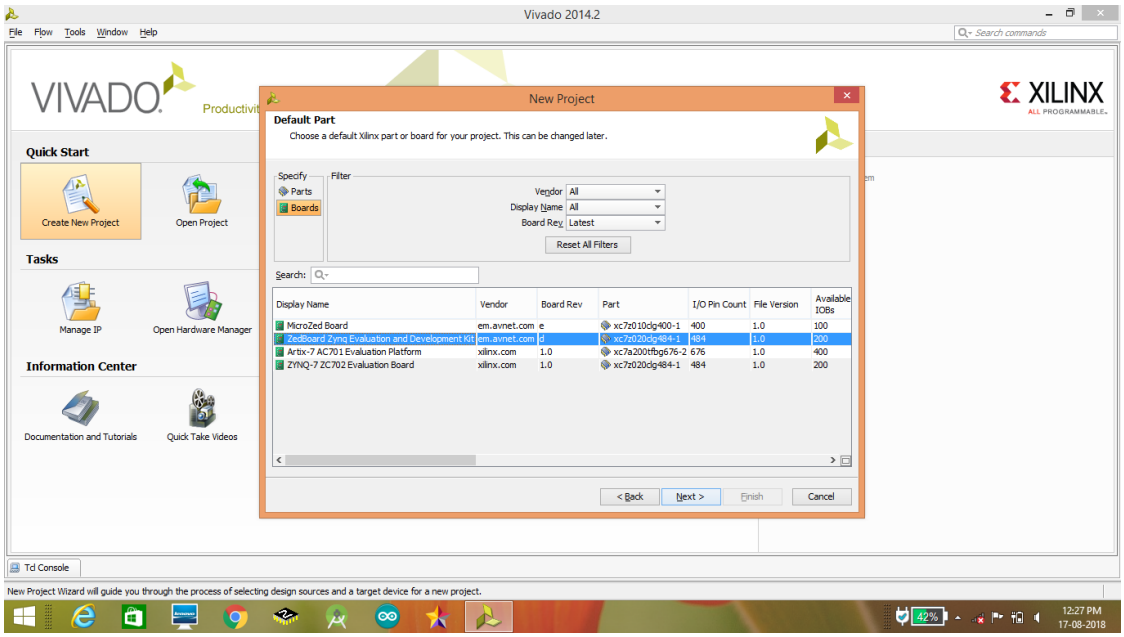


2 : Select→ RTL project and check→ Do not specify sources at this time

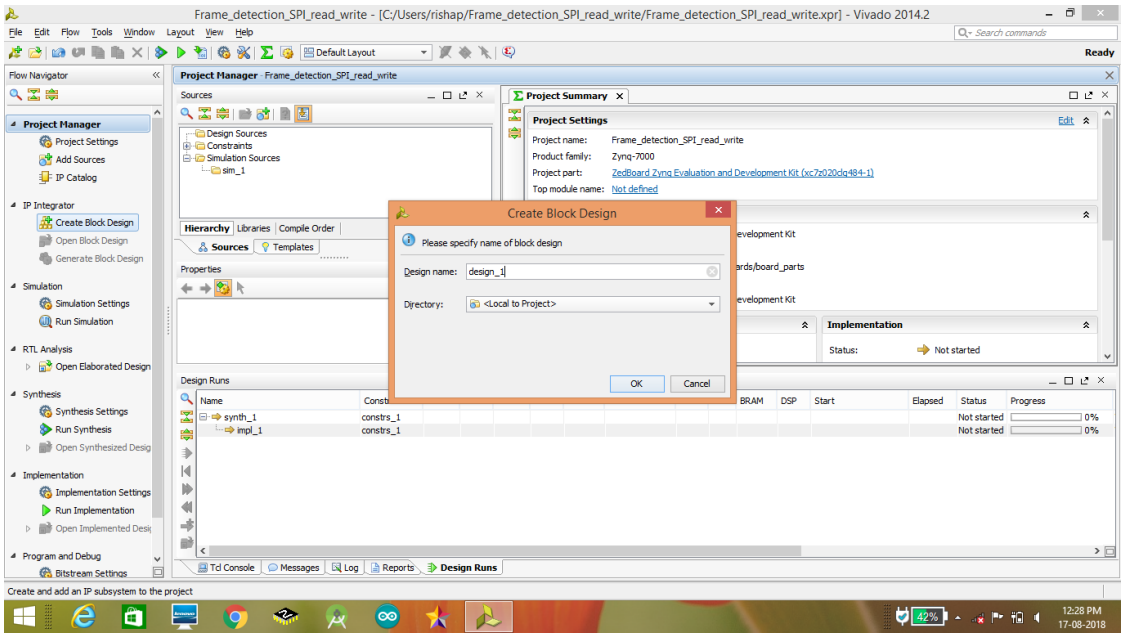
(Uncheck → To add a source files)



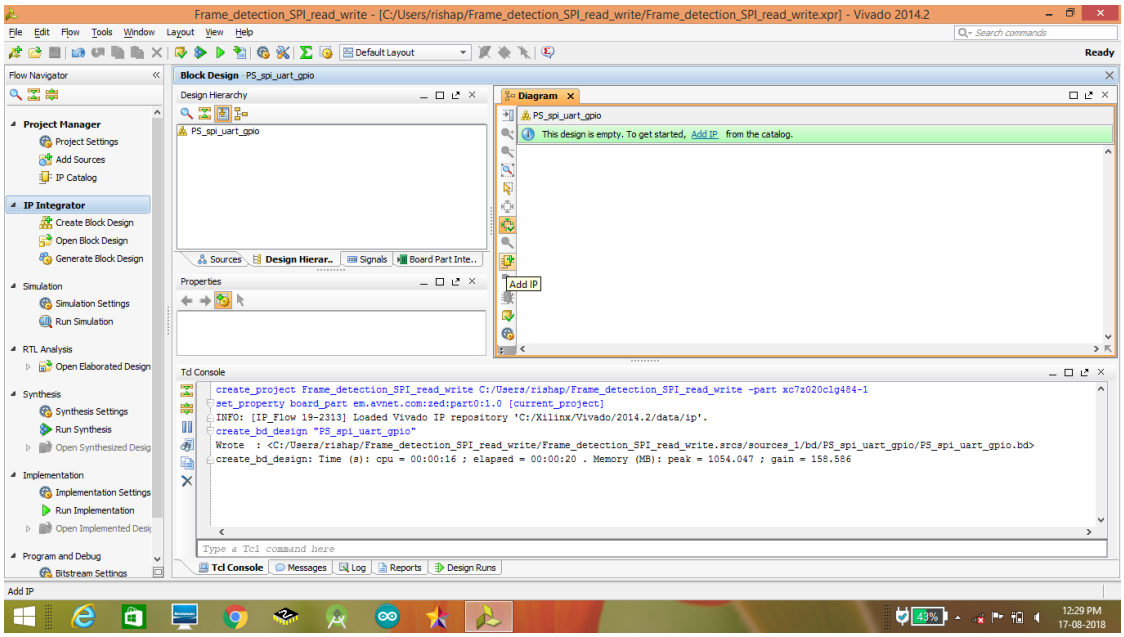
3: Select→ boards and select → Zedboard Zynq Evaluation and Dev kit



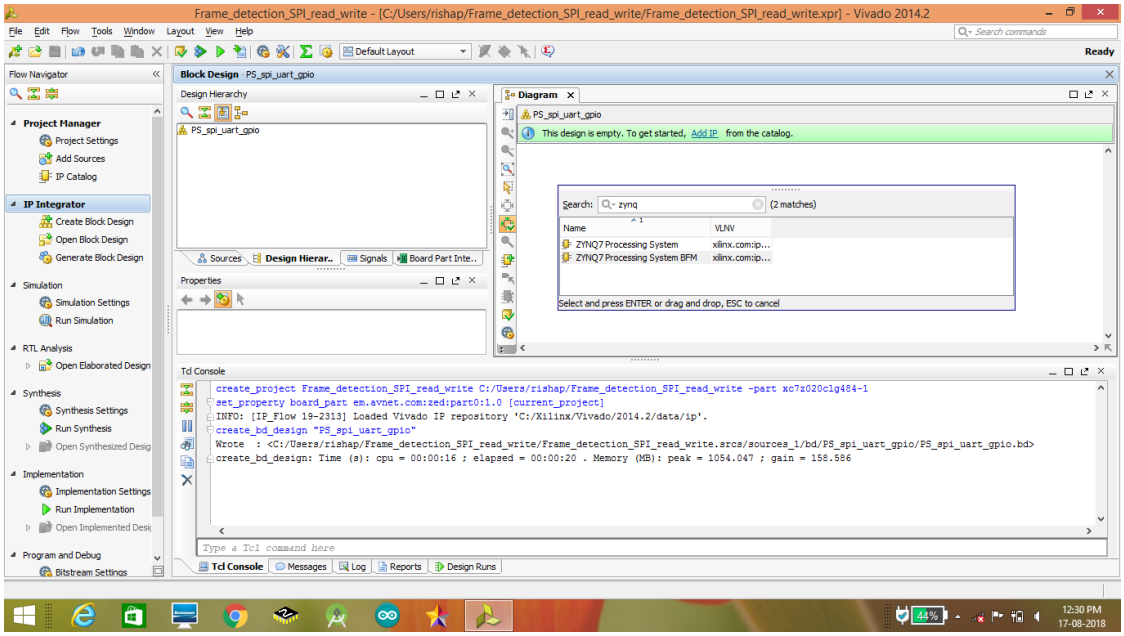
4 : Create a new block designed and name it.



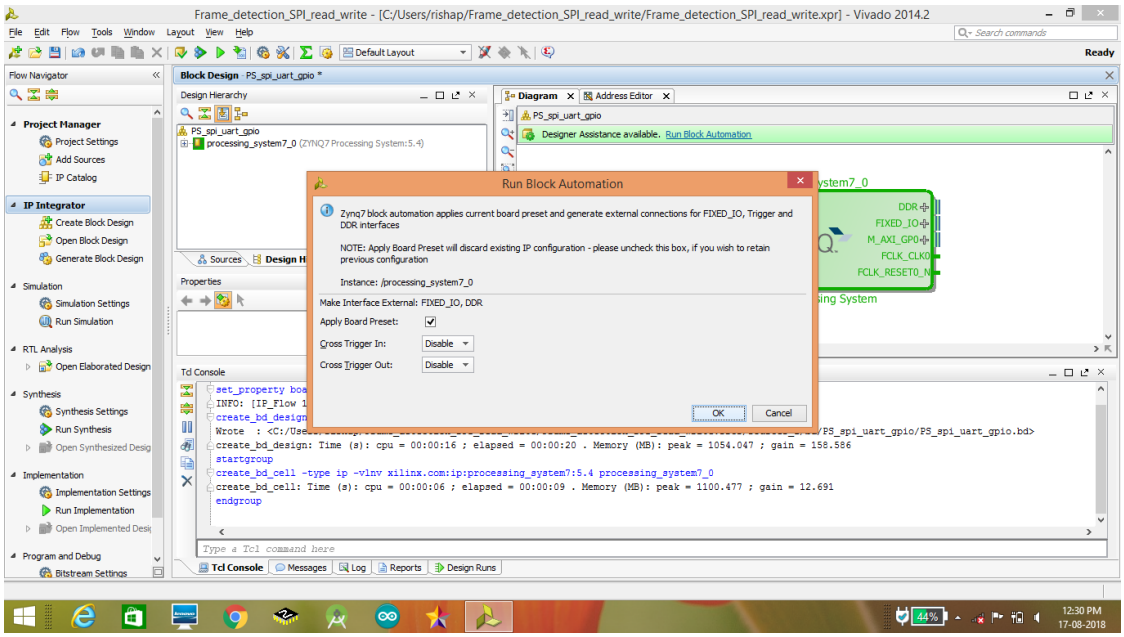
5 : Select Add IP



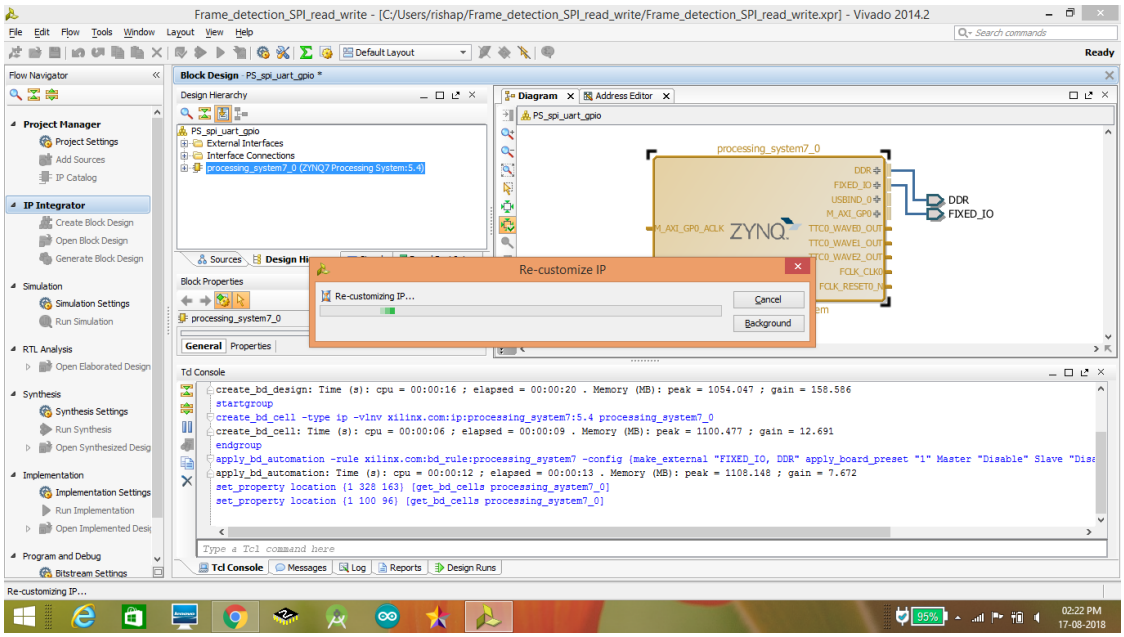
6 : Search for Zynq7 Processing System and double click on it to select.



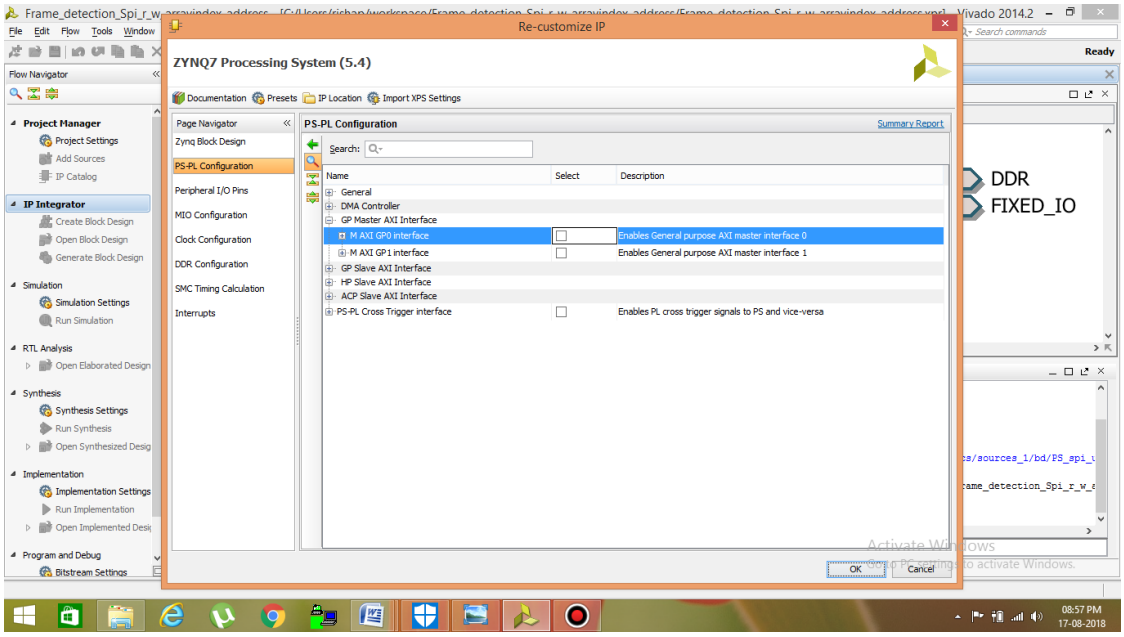
7: Click on →Run Block Automation and select → ok.



8: Double Click on → ZYNQ IP in block diagram to customize the Zynq IP as per our requirement.



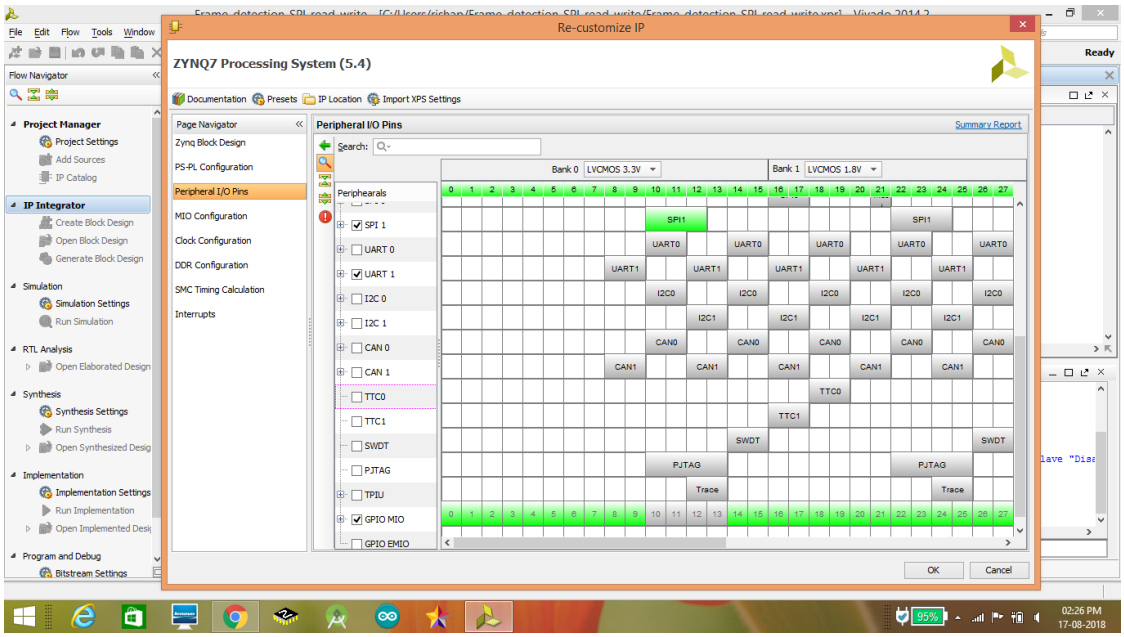
9: Uncheck the M AXI GP0 interface if not required.



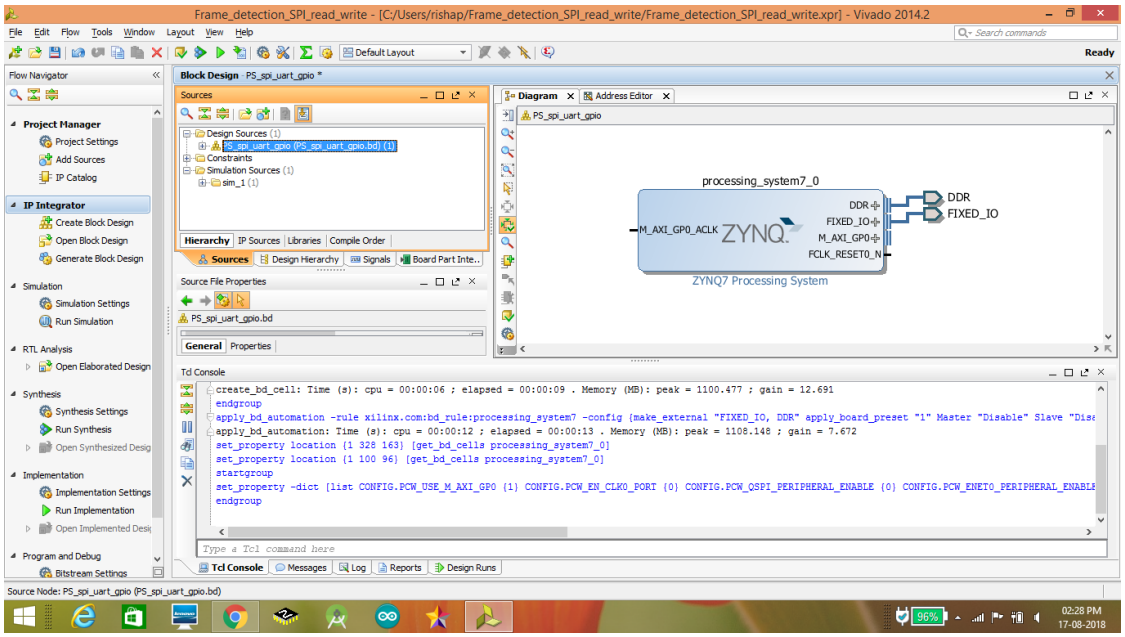
10: Check the peripherals required and Uncheck remaining peripherals

Customize the clock of peripheral, Interrupts as per requirement.

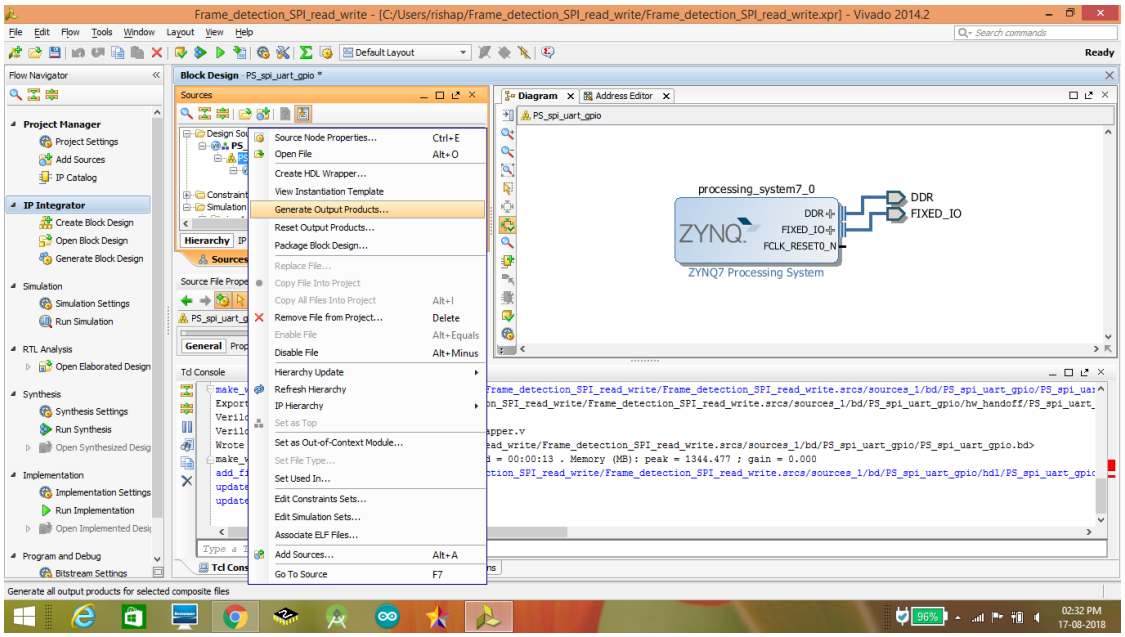
Click→ ok.



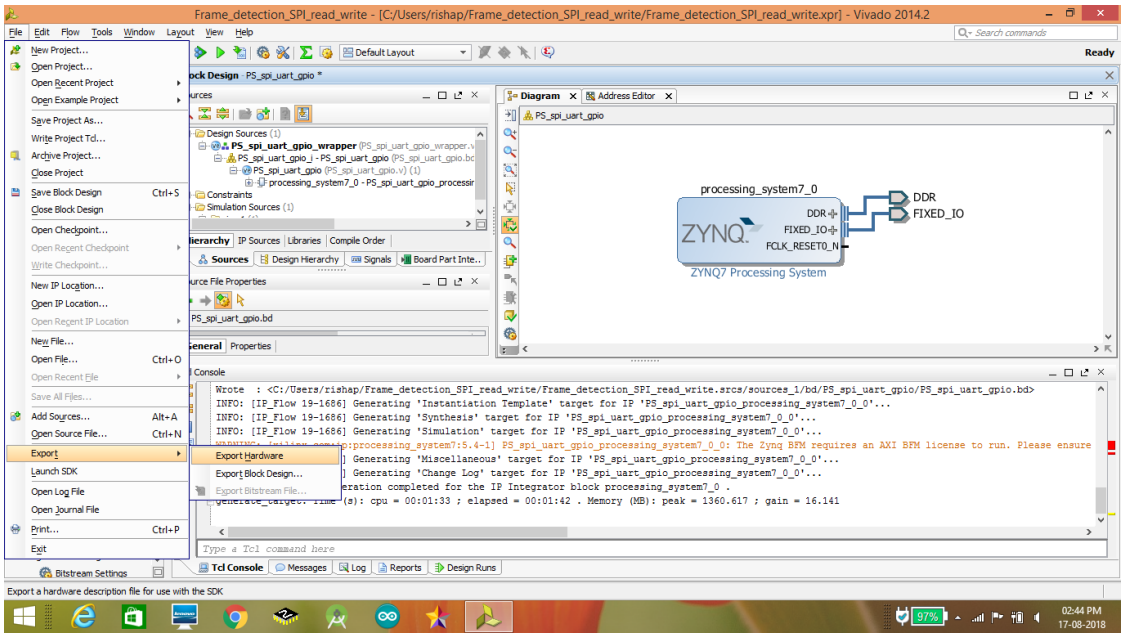
11: In Sources , Under →Design Sources , Right click on → block_diagram_name.bd



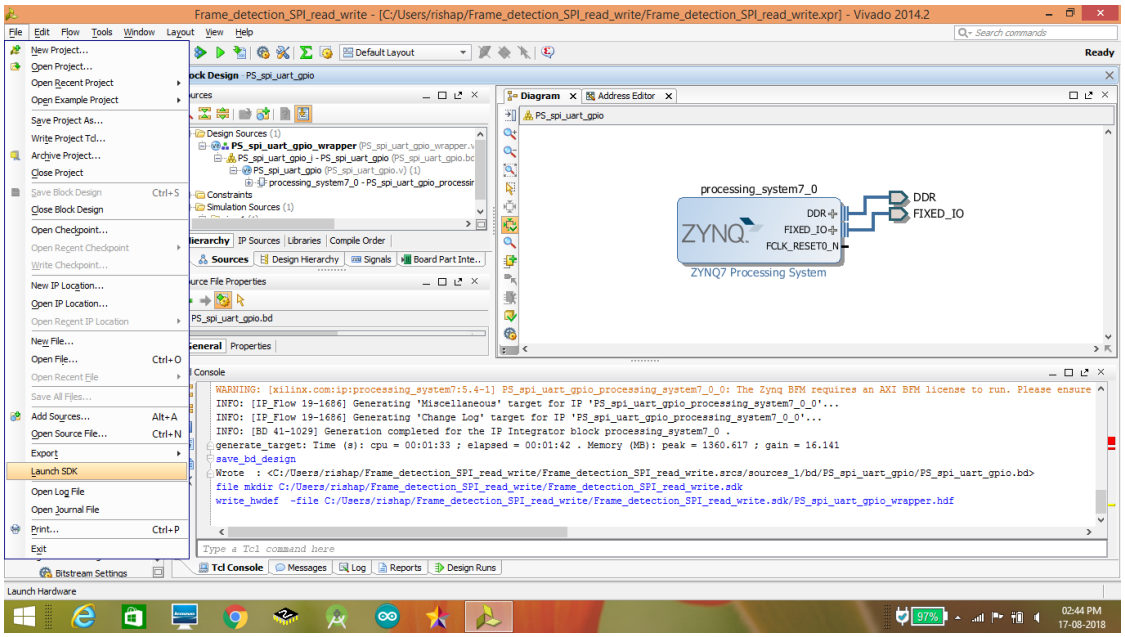
12: Select → Generate Output Products and wait to generate the hardware file



13: In File , Under → Export , click on → Export Hardware and click on→ ok

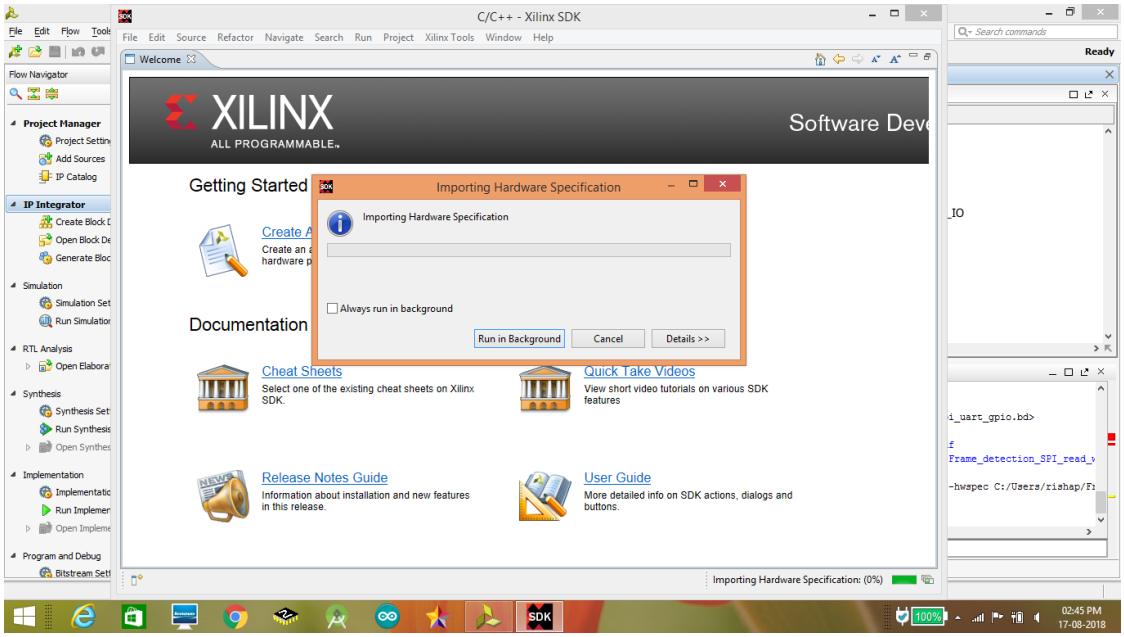


14: In File , Select → Launch SDK

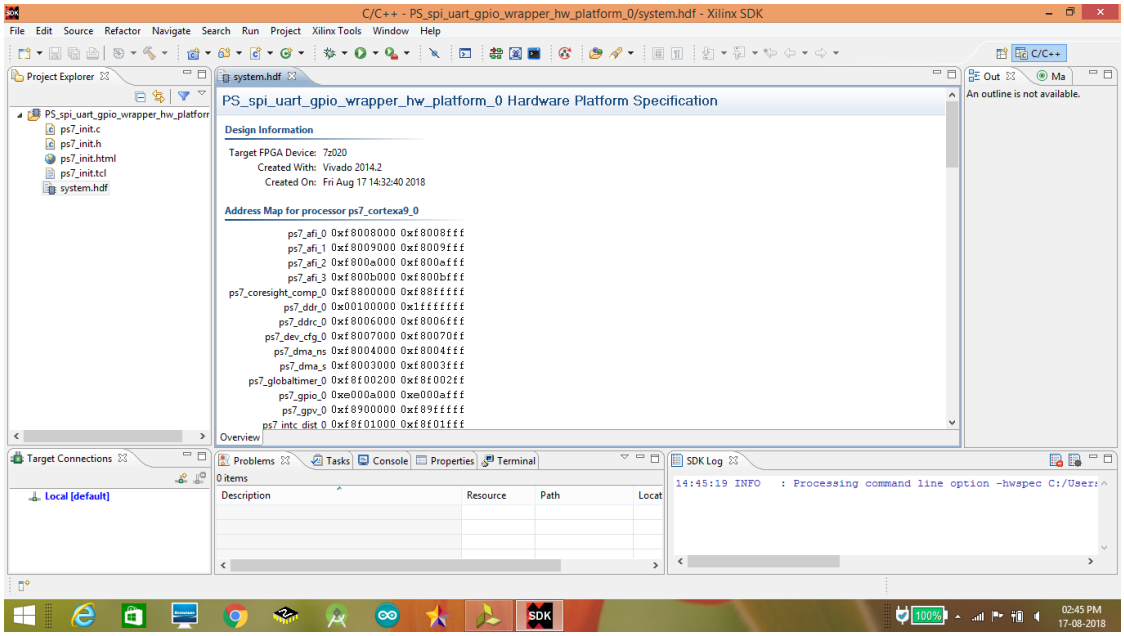


C : Steps to Create the C Application Project in Xilinx SDK and Program the Zedboard.

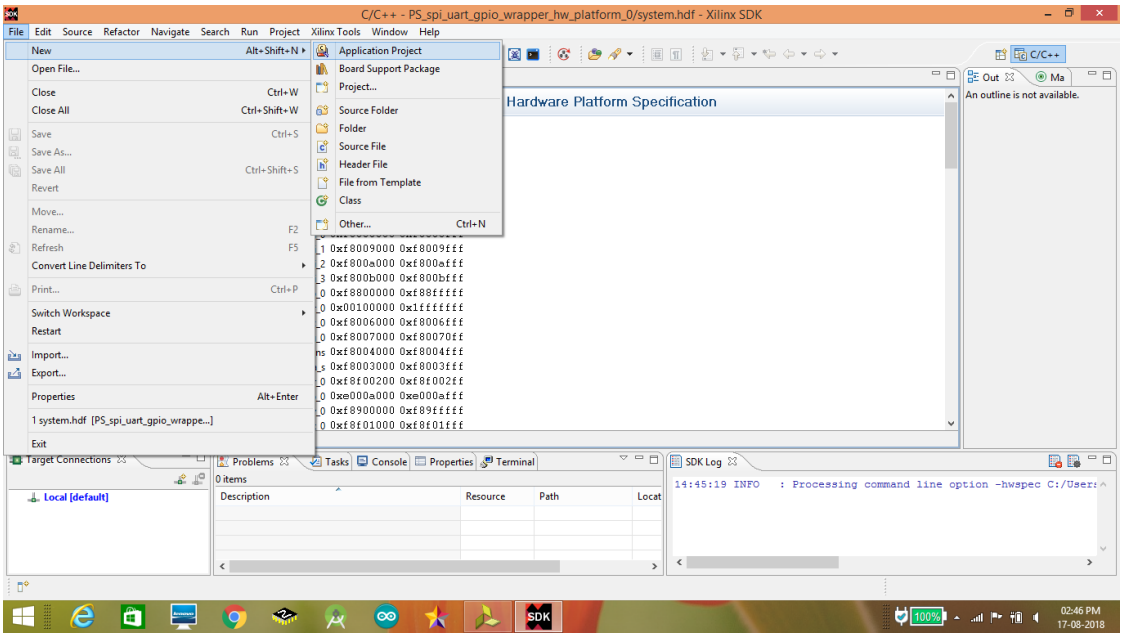
1: After launching the SDK from Vivado , all the hardware specification files get imported from Vivado to SDK.



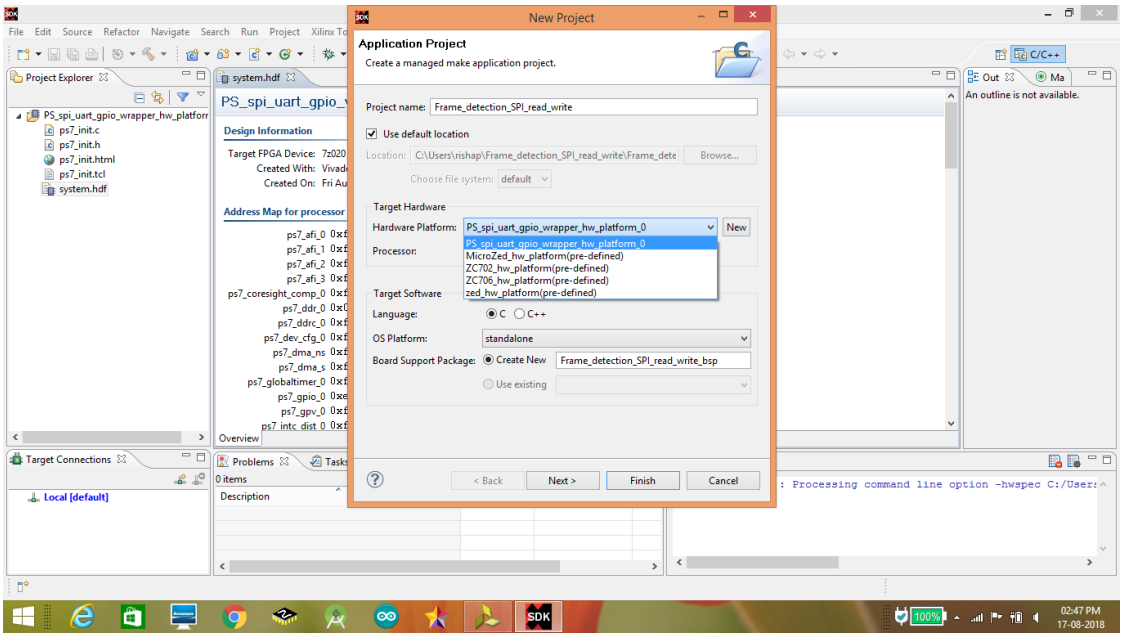
2 : Hard Definition file is added and can be viewed.



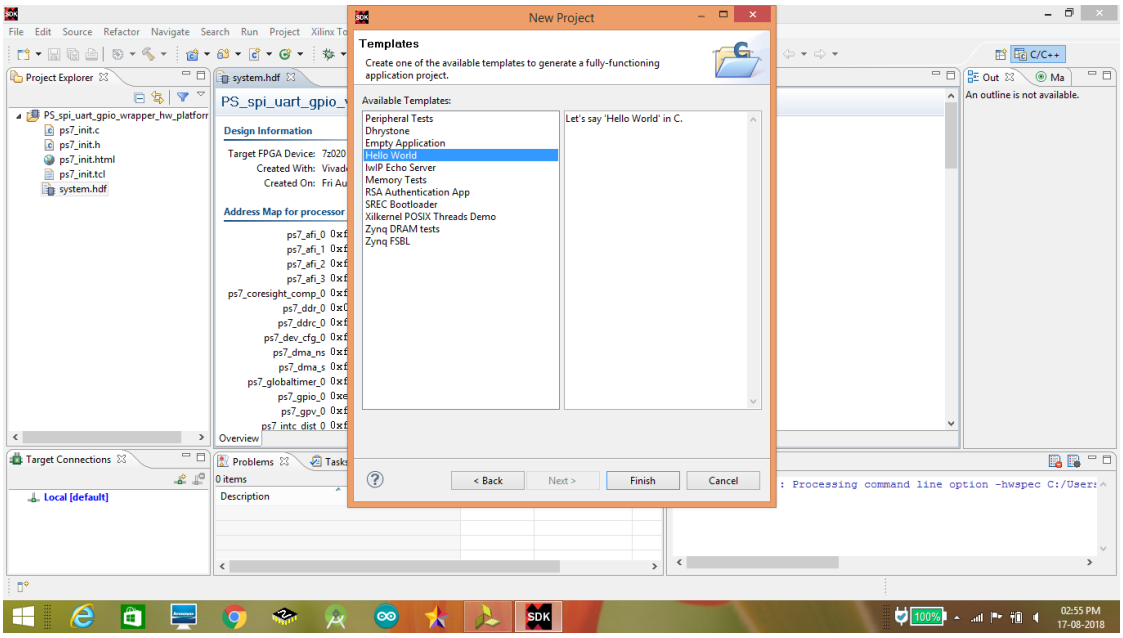
3 : Create new Application Project.



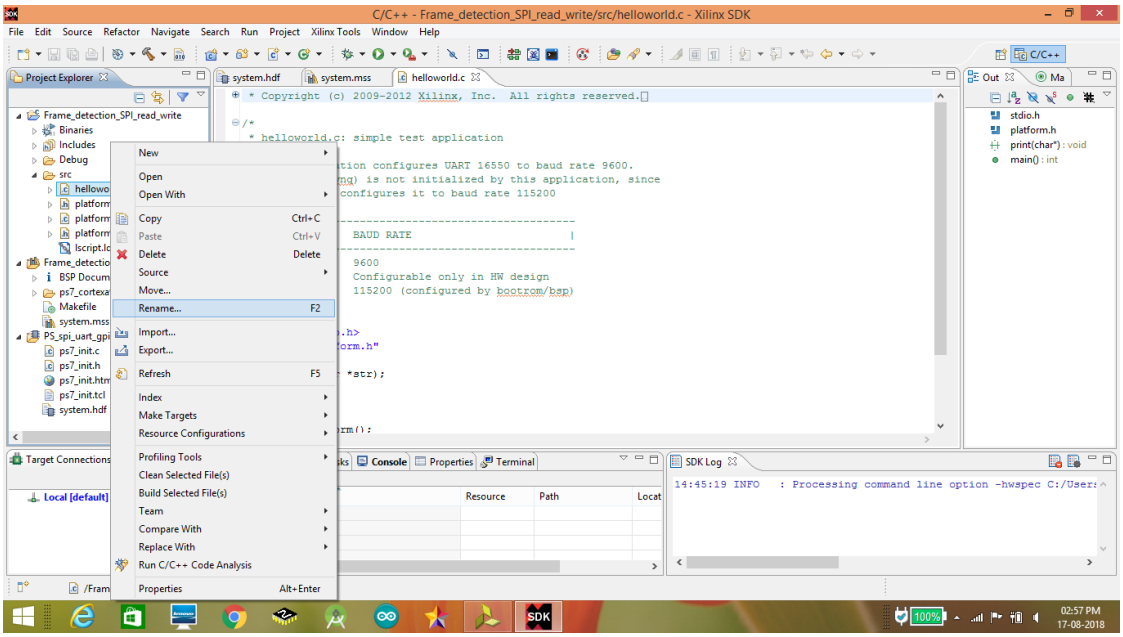
4: Fill project name, check →create new BSP and click →next



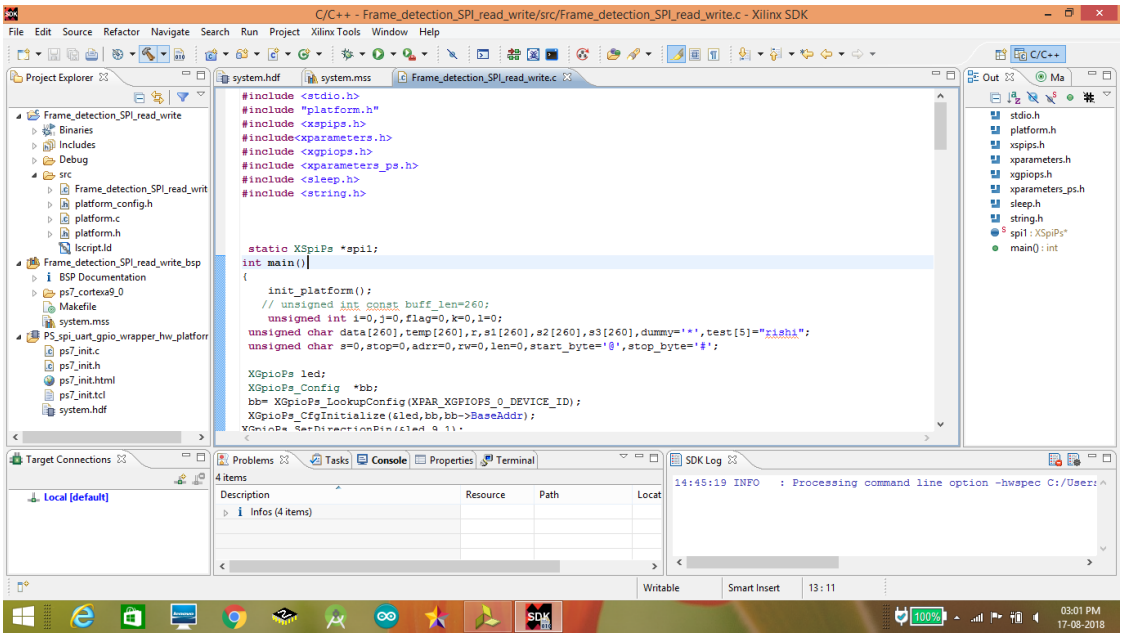
5: Select → Hello World template to have a default uart1 source files.



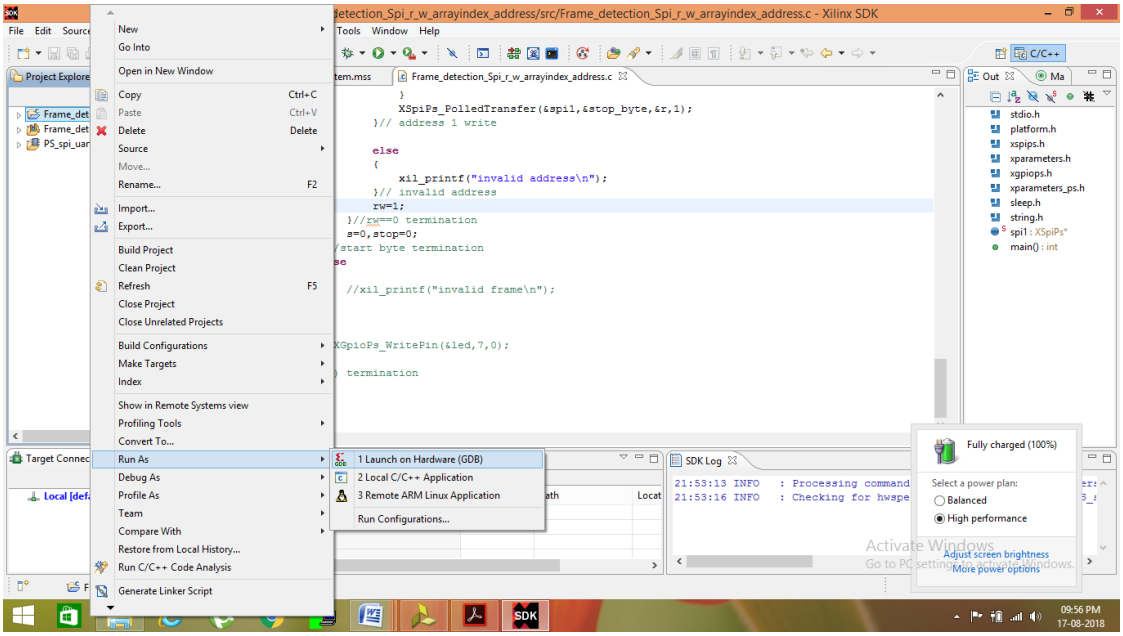
6: Rename the helloworld.c file (if required)



7: Type ,Save and Build the Code.



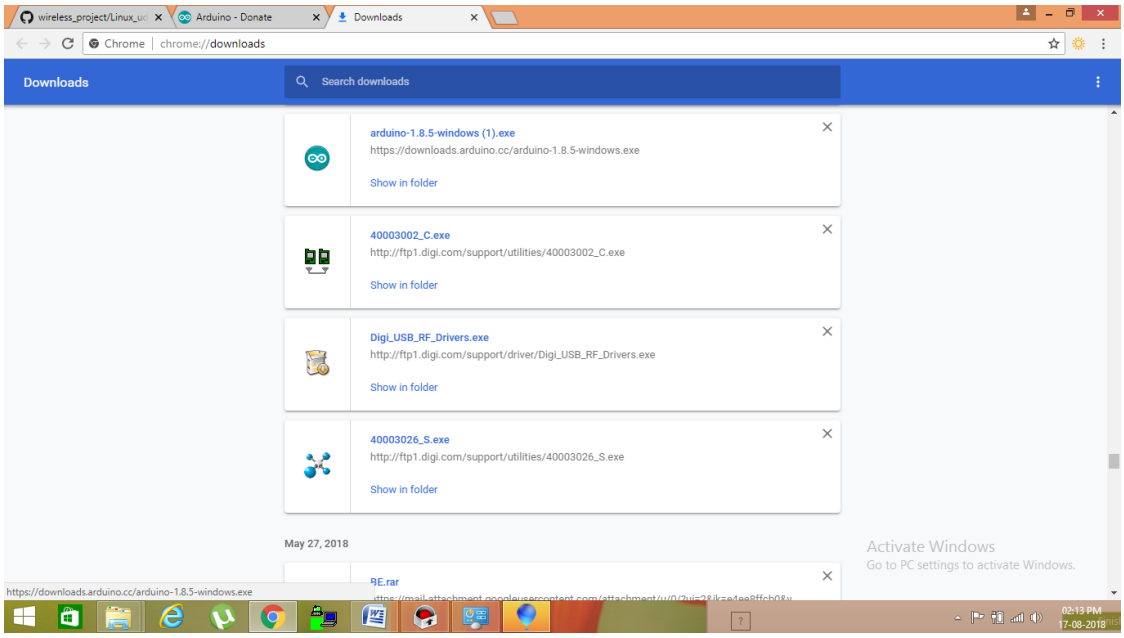
8: Connect the zedboard and launch the code .



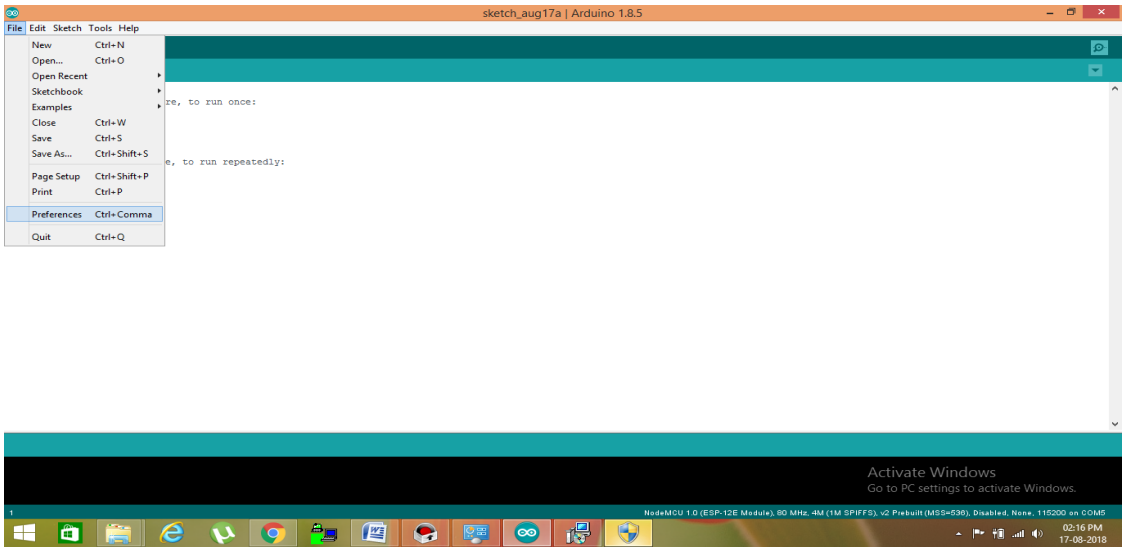
D : Arduino IDE Installation and Adding ESP8266 Library.

1: Download the Arduino IDE from the Arduino website.
(https://www.arduino.cc/en/Main/Software)

2 : Click on the downloaded .exe file to begin the installation and it will automatically get install.



3 : Open Arduino IDE and in File → Preferences



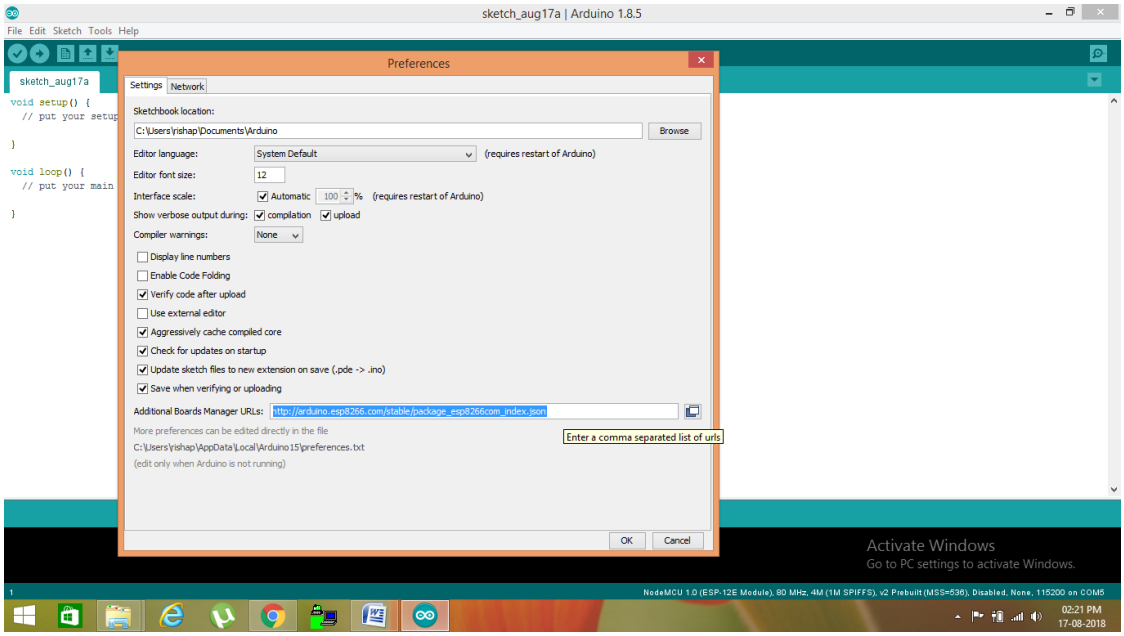
4 : Visit http://esp8266.github.io/Arduino/versions/2.0.0/doc/installing.html

to copy the URL of the ESP8266 library for adding it into the Preferences.

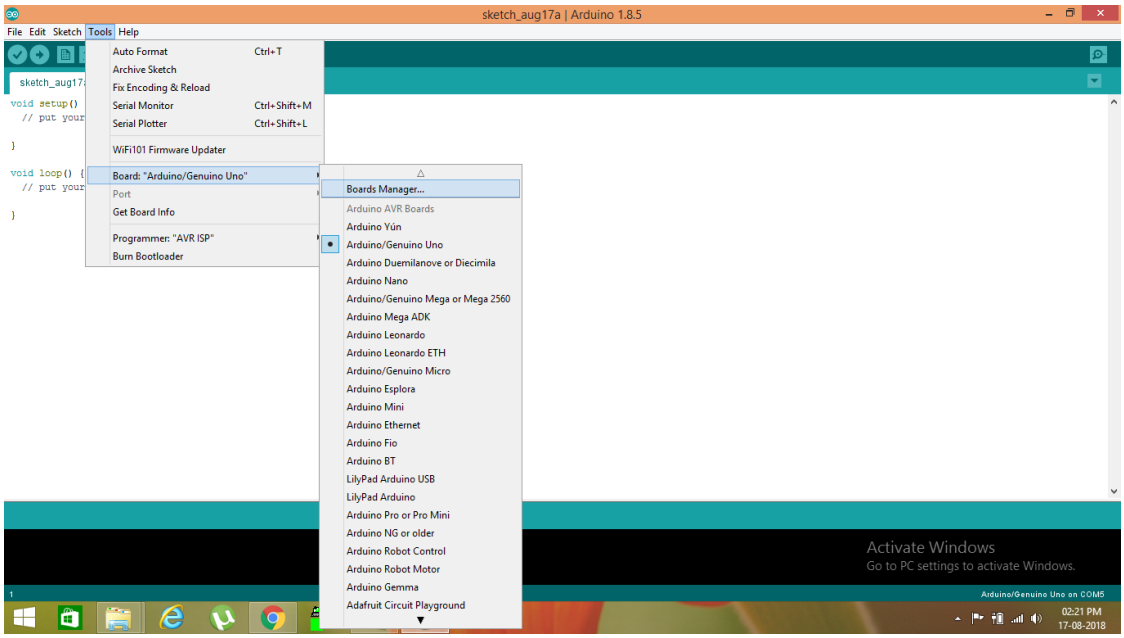
Or copy from below

http://arduino.esp8266.com/stable/package_esp8266com_index.json

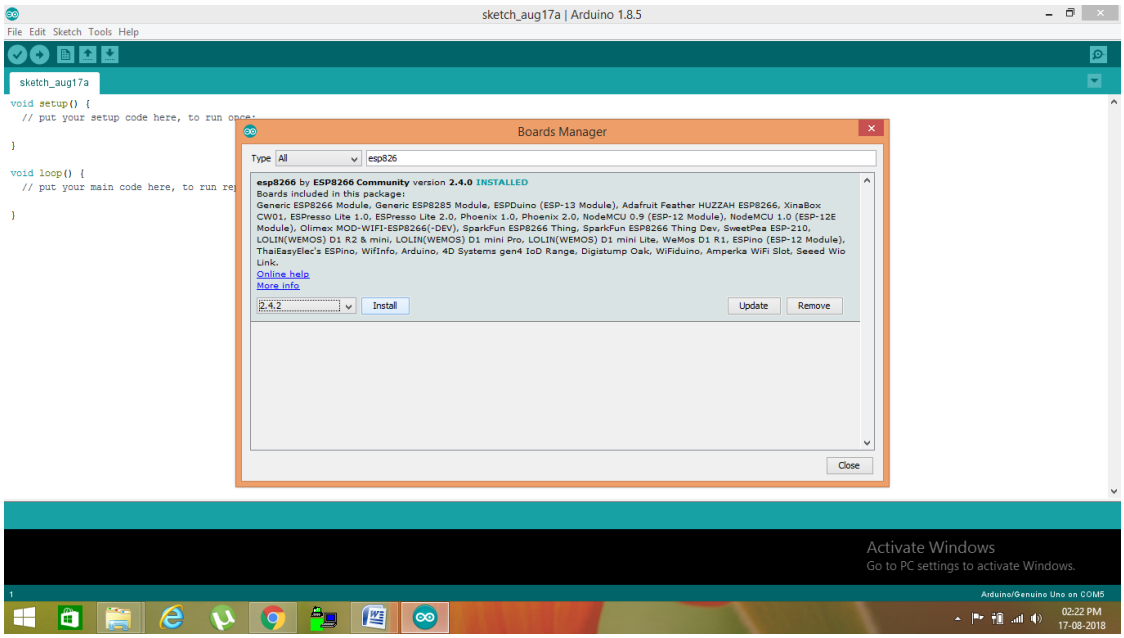
Copy and paste the above URL in : Additional Boards Manager URLs



5 : Goto → Tools → Boards and Select → Boards Manager

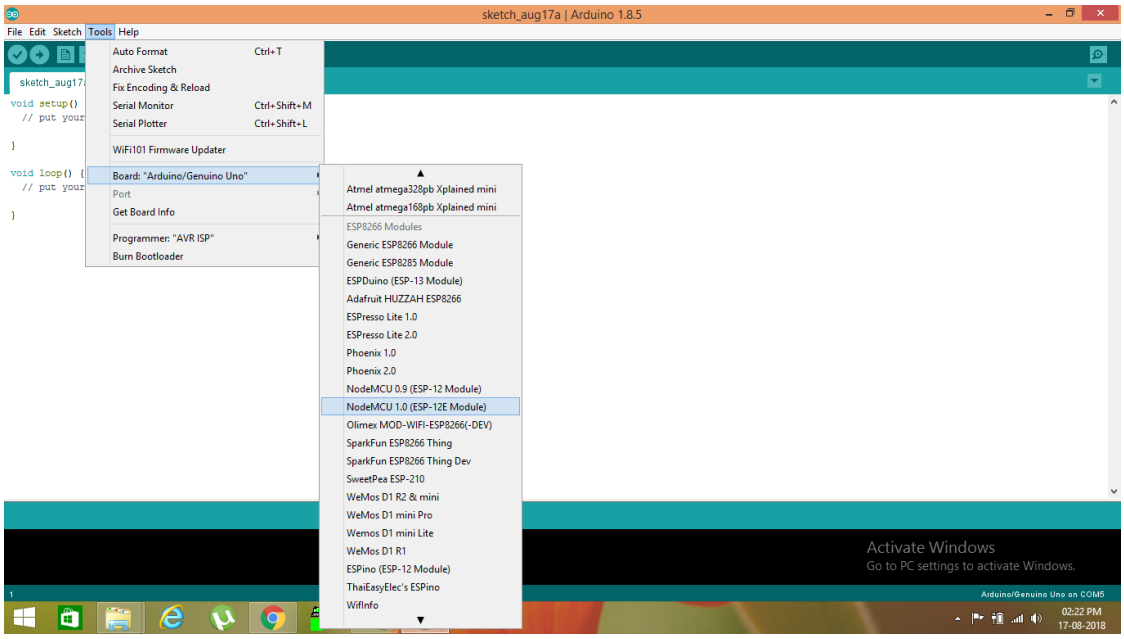


6 : Search for ESP8266 and Select → Install. (choose latest version).

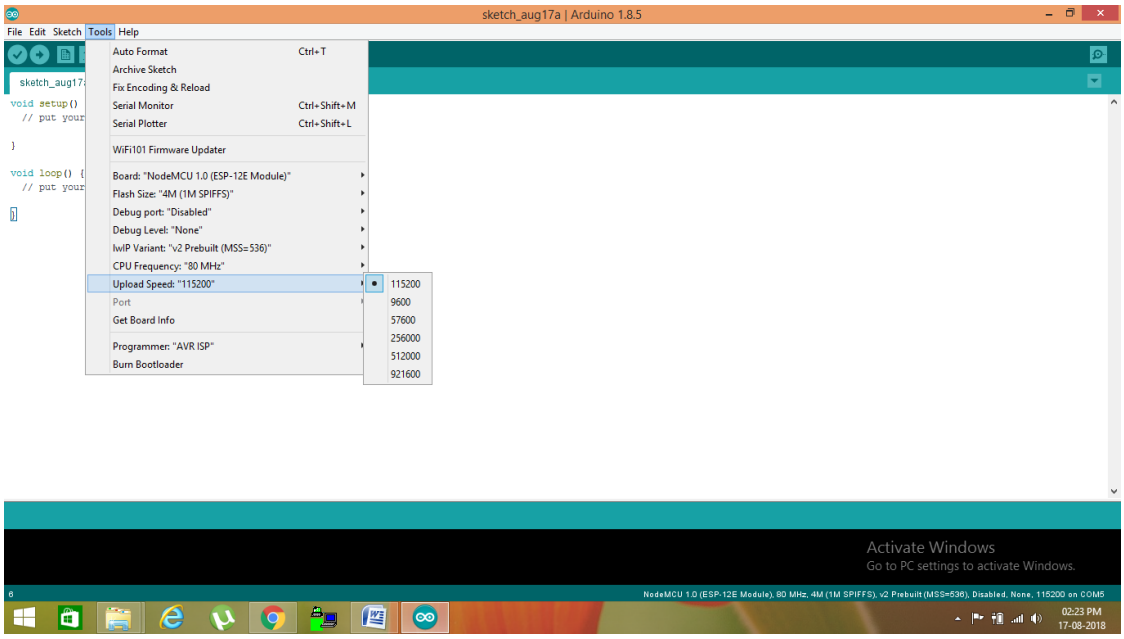


7 : After Board installation ,check for NodeMCU 1.0 board under ESP8266 Modules

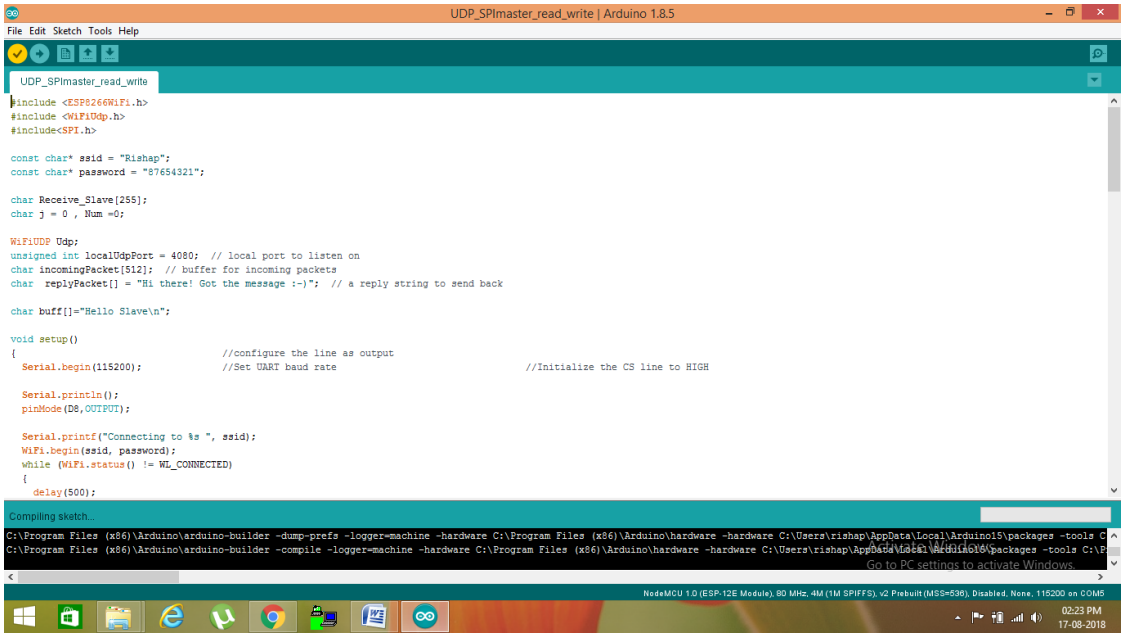
Tools→Boards→Scroll down the list→ select → NodeMCU 1.0.



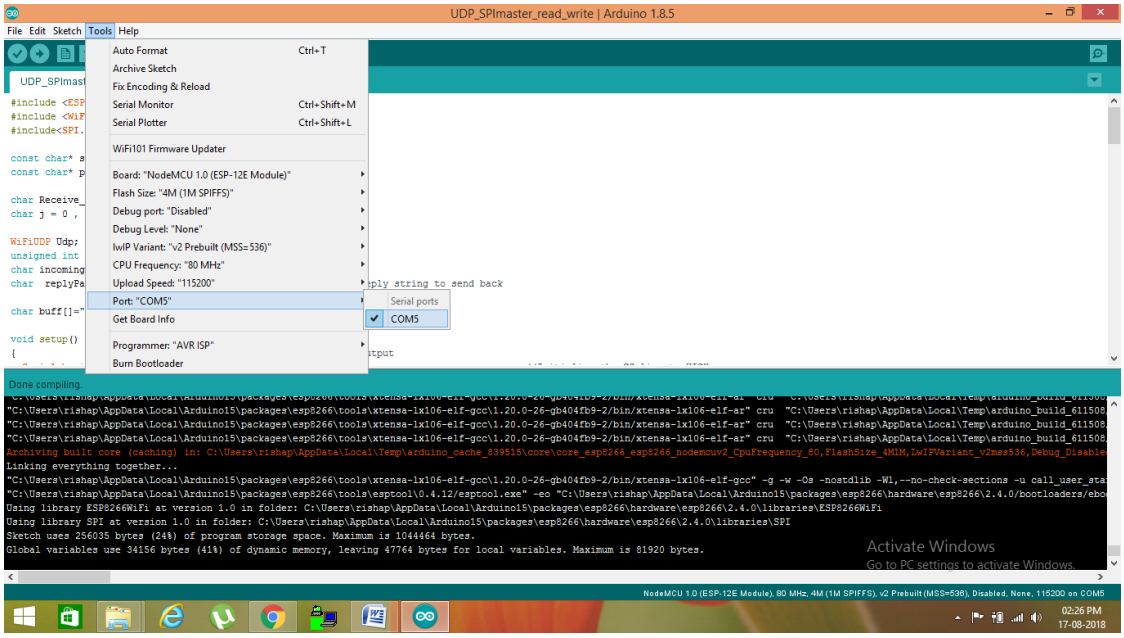
8 : After Selecting Board ,we can configure some basic Functionality. Under Tools.



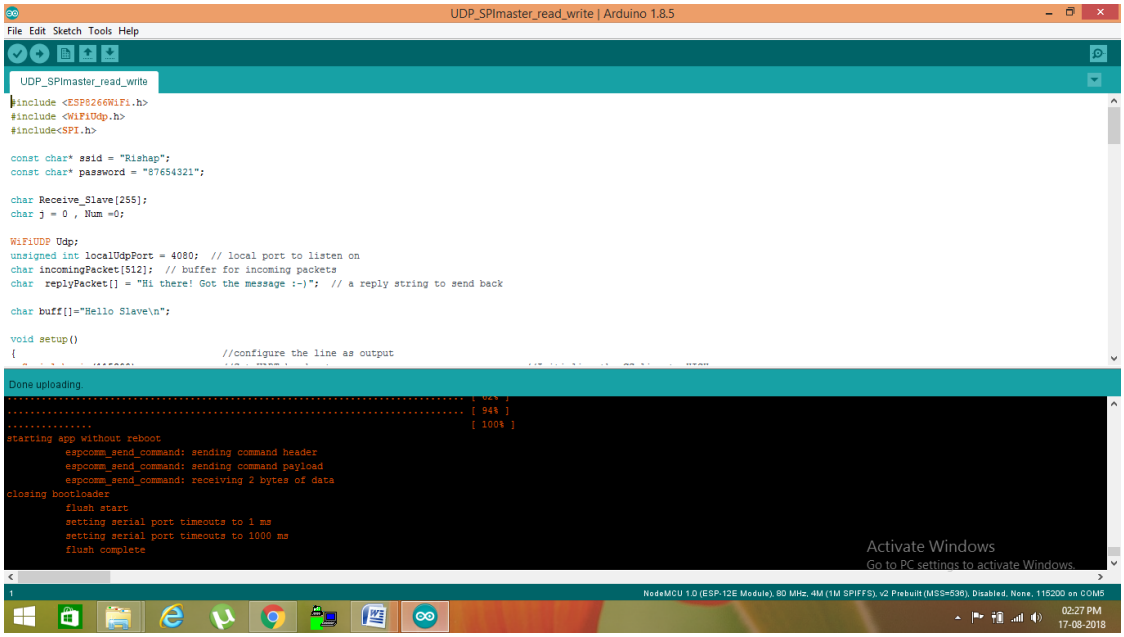
9 : Write and compile the code using compile option(yellow coloured symbol on top left corner).



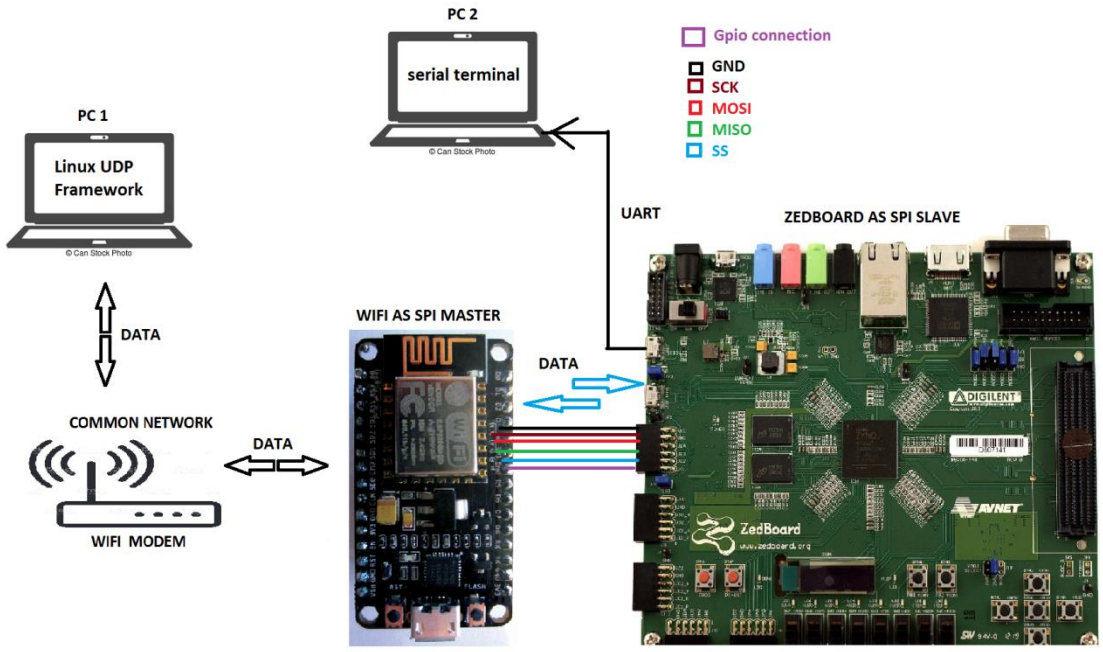
9 : Connect the NodeMCU to PC and select the ‘COM PORT’



10: Upload the code using upload option (Right arrow Symbol on top left corner)

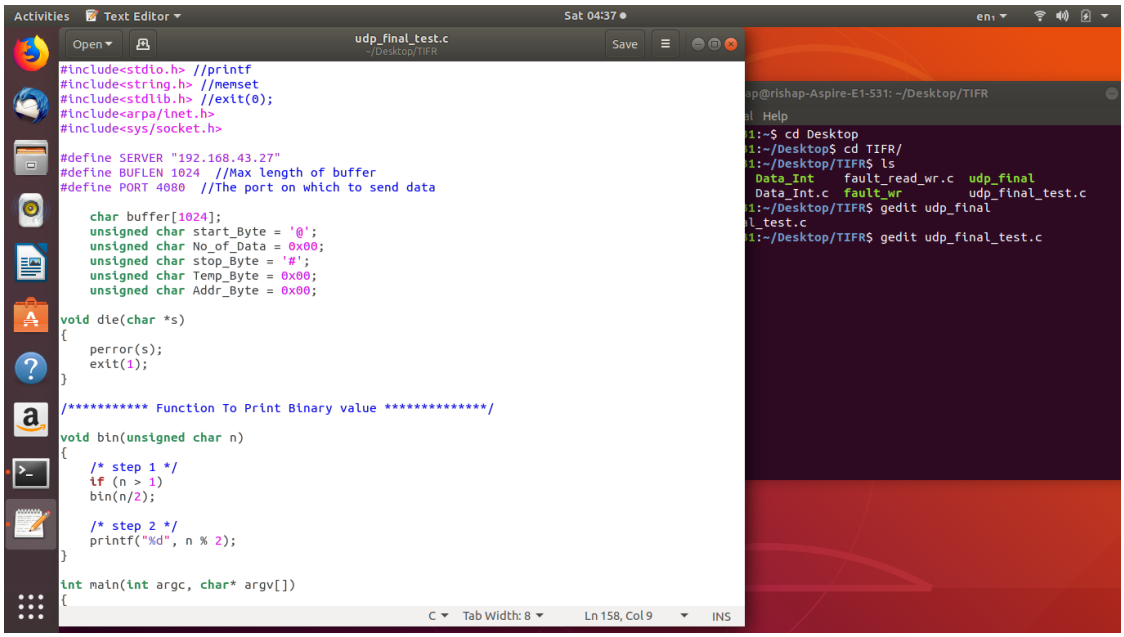
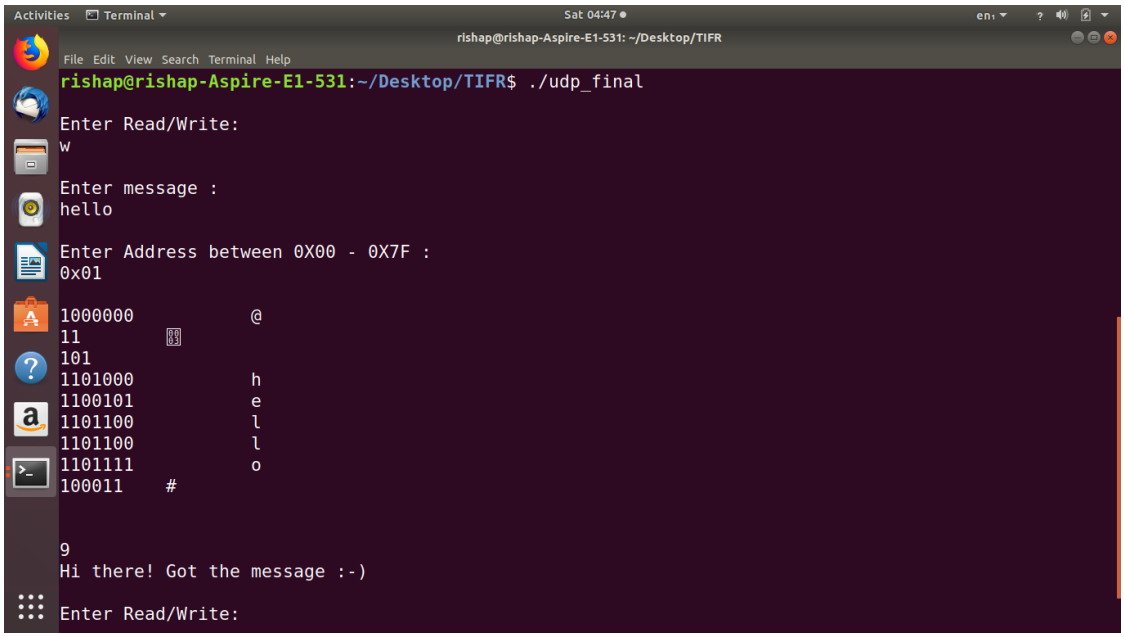


D : System Interconnection:



D : Running the Framework in Linux.

1: Open default text editor and write the framework code and save the file with an extension .C (C source file)



2: Compile and Run the framework .