IMPLEMENTATION

KENNY NISANTH

April 2024

1 Introduction

Here you will be able get knowledge about how to simulate verilog code on hardware using FPGA(Field programmable gate arrays). By following the instructions which mentioned bellow, you can get complete knowledge from, creating to verifying on a FPGA. For doing a hardware implementation make sure that you read the data sheet of that particular FPGA development kit, so that you provide the right voltage as input and you will gain complete knowledge on the uses of each part.

2 Instructions

Follow the video or below point for instructions

- Open xilinx vivado(make sure that you have a license. So, that you can implement the code on FPGA).
- Create project by clinking on create project button.
- Name the project with proper syntax and select RTL Project by clicking next
- By proper reading of the FPGA development kit data sheet, enter the part and board names.Here we have used zybo zynq 7000 and xc7z010clg400-1 as part names.
- By clicking on the finish button the project is created successfully.
- By clicking on Add sources option from the project navigator you can create the design and simulation sources
- Write the error free code for design and test bench code.(follow proper syntax for naming the files and write the code without any errors)
- simulate the project and verify the project.

- After verifying the simulation run RTL analysis. Here you can get the elaborated design and schematic of the project.

- Now, at the top right corner you can find i/o planning option, click and add details of the input and output configuration on which switches you wanna add inputs and output's. Ex:w14, m15. make sure that you add how much voltage the pin should get(these can be known by reading the data sheet of that particular FPGA development kit).

- Now run synthesis by using the synthesis option. you will get synthesized design, also run implementation.

- In implementation you will get implemented design, you will get to know which parts are getting used inside a FPGA.

- connect the FPGA development kit using a suitable wire and connect it to the PC or desktop on which you are working on.After connecting the FPGA on the switch you will find a LED glowing indicating PGOOD (indicating that all the FPGA is working good).

- After connecting the FPGA, open hardware manager and make sure the Target device is connected successfully

- Before connecting an FPGA make sure you get the bit-stream by clicking on the generate bit stream option.

- Now insert the code into the FPGA and verify the project(if the LED glows its 1, if not its 0)

For better understanding of the steps open the following link and watch the detailed video

https://drive.google.com/file/d/1izDa4S5C4g5nBxLyVCUIiz2lMlYY0kVn/view?usp=drive$_l ink$

For the hardware implementation video check the GitHub

https://github.com/KENNYNISATH/FPGAspeaks/blob/main/Implementation/hardware