

# ECE474/574 - Lab 1

## Introduction

The goal of this lab is to install Quartus Prime 15.1 on your personal machine and verify its functionality by programming a simple project to your FPGA. Program the FPGA so you can control 4 LEDs with the 4 switches on the board. Refer to figure 1 for a simplified schematic with pin-outs for use in Assignments ⇒ Pin Planner. For more information on the DE0-Nano, refer to Terasics documentation here.

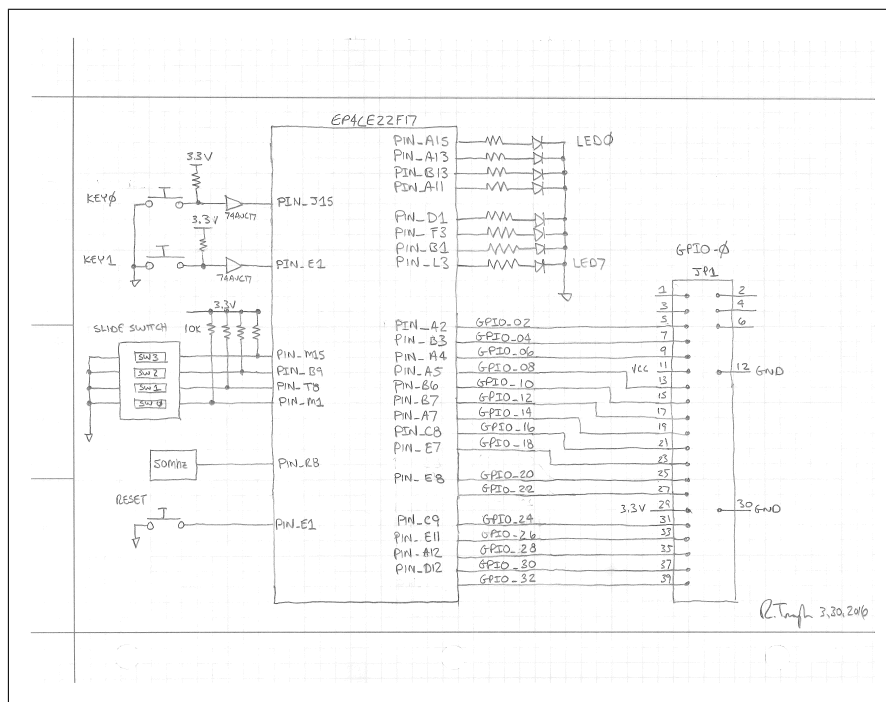


Figure 1: FPGA Schematic

## Building Your Project

Once Quartus 15.1 is installed you will need to make your project. Go to File ⇒ New Project Wizard... and follow the instructions to make a new project. Select an empty project and import the Lab1.sv file provided on the website. Make sure to select the correct FPGA device: Cyclone IV E: EP4CE22F17C6. Once you have finished making your project we will need to run Analysis and Synthesis. This will check your Verilog for errors. Once it is done we can assign pins to the inputs and outputs. Download the pin file provided with this lab

and select Assignment ⇒ Import Assignment and select the downloaded pin file. Now you can compile the project by double clicking "Compile Design" within the Tasks window. Alternatively, you can click Processing ⇒ Start Compilation or press CTRL+L.

## Simulating

Simulating your design will help you verify everything is working properly before you program your FPGA. To simulate your design, select Tools ⇒ Run Simulation Tool ⇒ RTL Simulation. Once ModelSim Altera edition opens it will automatically compile your files and load the created library, called `rtl_work`, in the Library window. To simulate the design type `vsim rtl_work.<Quartus_Project_name>`. For example, for me to simulate my project, called Lab1, I typed `vsim rtl_work.Lab1`. You should not see all of your inputs and outputs in the Objects window. For this lab that should be a switches input and a LED output. To add these signals to the wave you can type `add wave <signal_name>`. For example, add wave LED will add my output signal named LED. Now we need to force the switch signal to a value and see if the LEDs update accordingly. To do this, type `force <signal_name> <value>`. For example, `force Switches 1100` sets my 4-bit switch input to 1100. To see the resulting waveform type `run <time>`. The unit for time is defaulted to ps but you can type `run 10ns`, for example, to run for 10ns.

## Programming your FPGA

After compiling your project Quartus will output a SOF file, used to program your FPGA. The DE0-Nano contains Altera's USB-Blaster circuitry for programming with the SOF file. In order to use this you will need to install the USB-Blaster drivers onto your computer (which did not install during my basic Quartus Prime install). The drivers are located by default at `C:\altera_lite\15.1\quartus\drivers\usb-blaster`. To install on windows, open up Device Manager with your DE0-Nano plugged in, select the USB device, in the Drive tab select update drivers, and browse to the USB-Blaster directory. More information from Altera on the USB-Blaster can be found [here](#).

## Checkoff

Show the TA your FPGA with the working code.