

Getting Started with Quartus II and the DE0-Nano board

Creating a new project

1. Select **File > New Project Wizard**
2. Specify a new directory for your project. Each project should have its own separate directory as lots of files will be created as you develop a design.
3. Enter a project name. e.g. MyProject. (Note spaces are not allowed in project names). The top level design name is the same. Enter the same name if it hasn't already appeared.
4. Click **Next** and then **Next** again to take you to **step 3 of 5**.
5. Select the device family **Cyclone IV E** and then select the **EP4CE22F17C6** device from the list below. This is the FPGA device that is on the DE0-Nano board.
6. Click **Finish**.

Add a schematic design file

We now need to add a top level module to specify the design. Here we will use a schematic to specify the design.

1. Select **File > New > Block Diagram/Schematic**.
2. Click **OK**. Block1.bdf, which you will save as the top-level design will appear in the main window.
3. Select **File > Save As. MyProject.bdf**
4. Click **Save**.

Adding an Input or output pin to the schematic design

The following steps describe how to add an input or output pin to the schematic.

1. Right click in the blank area of the BDF and select **Insert > Symbol**.
2. Under Libraries, select **quartus/libraries > primitives > pin >input**.
3. Click **OK**
4. Double click on the pin to rename the pin to match the input switch on the board e.g. SW0
5. Add more input pins as required.
6. Output pins are added from **quartus/libraries > primitives > pin >output**.

Physical pin assignment

Now we need to make pin assignments to link them to the actual pins on the FPGA. Before making pin assignments, perform the following steps:

1. Select **Processing > Start > Start Analysis & Elaboration** in preparation for assigning pin locations.
2. Click **OK** in the message window that appears after analysis and elaboration completes.

To make pin assignments to the KEY [0] and CLOCK_50 input pins and to the LED[3..0] output pins, perform the following steps:

1. Select **Assignments > Pin Planner**, which opens the Pin Planner, a spreadsheet-like table of specific pin assignments.

Pin assignments for the DE0-Nano Board.

Signal Name	FPGA Pin	Type
KEY0	J15	Input
KEY1	E1	Input
SW0	M1	Input
SW1	T8	Input
SW2	B9	Input
SW3	M15	Input
LED0	A15	Output
LED1	A13	Output
LED2	B13	Output
LED3	A11	Output
LED4	D1	Output
LED5	F3	Output
LED6	B1	Output
LED7	L3	Output

2. Enter the FPGA pin name into the Location column. The full pin name will appear. Press return to select.
3. When finished **File>Close**

Compiling the design

In the **Processing** menu, select **Start Compilation** or click the **Play button** on the toolbar. The console should eventually say *Full Compilation was successful*, if you have no errors.

Uploading your design to the FPGA on the DE0-Nano

First we need to install the Altera USB-Blaster if you have not already done so:

1. Plug in the DE0-Nano USB cable. It will not install properly ignore and cancel where necessary
2. Open Device Manager. Under Other devices you should see USB-Blaster.
3. Right click and select Update Driver Software, then Browse my computer.
4. Select folder C:\altera\14.0\quartus\drivers
5. The driver should then install

Now to upload the program

1. **Select Tools > Programmer.**
2. Click **Hardware Setup.**
3. If it is not already turned on, turn on the USB-Blaster [USB-0] option under currently selected hardware
4. Click Close.
5. If the file name in the Programmer does not show MyProject.sof, click Add File.
6. Select the MyProject.sof file from the output_files folder of the project directory
7. Click the **Start button.**

Your program should now be running on the DE0-Nano