Lab 7 - PSoC 4 BLE Blinking LED

School of Engineering

Electrical and Computer Engineering Department

ECEG 721-61 Embedded Systems

Jamie Quinn & Lucia Rosado-Fournier

December 2nd, 2020

1.1 Objective

The main objective of this lab is to become familiar with PSoC Creator and implement a simple blinking LED.

1.2 Introduction

The specific LED that will be the blue LED which is connected to pin P3[7]. To make it bilink, the pin will be driven low to turn on and driven high to turn off. To drive the pin, the component configuration tool is used.

1.3 Component Requirements

1.3.1 Hardware Components

- CY8CKIT-042-BLE-A Bluetooth Low Energy Pioneer Kit
- Provided USB Cable

1.3.2 Software Components

- PSoC Creator 3.1 or newer
- CySmart 1.0 or newer
- BLE Pioneer Kit Revision C or newer

1.4 Software

Code

The code used in this lab is a for loop that runs forever to turn on the LED and then delay for a full second to make it blink.

```
#include "project.h"

int main(void)
{
      for(;;)
      {
         Pin_LED_Write(~Pin_LED_Read());
         CyDelay(1000);
      }
}
```

1.5 Procedure

As this was the first time using the board, there is some initial setup needed.

1. Install the softwares mentioned in the Software Components section.
2. Plug in the BLE Pioneer Kit into the PC.
3. If prompted, wait for the windows driver installation to finish, then click “Skip obtaining driver software from Windows Update”. This is because the previous softwares install the necessary drivers.
4. Open the PSoc Programmer located in the directory the previous softwares were installed in.
5. Click “KitProg/xxx…” to select the board. You may get a warning that the programmer is out of date. Click “Ok”.
6. Switch to the “Utilities” tab and click “Upgrade Firmware”.

After the initial setup is completed, a new project can be created and configured.

1. Open PSoC Creator located in the directory where the Cypress software was installed.
2. Click “File” -> “New” -> “Project”.

3. Select the correct Target Device, choose to create a blank project, name the project, and select a location. The exact process is different depending on the version of PSoC Creator being used. To get the board number, look at the chip on the board.
4. After the project is created, “TopDesign.cysch” will open in a new tab. Go to the “Component Catalog” on the right, click on “Ports and Pins”, and drag “Digital Output Pin” to the schematic.
5. Double click the pin to open the “Configure” window. Rename the pin to “Pin LED” and deselect “HW connection” under “Digital output”. Click ok to apply the changes.
6. On the left hand side under “Workspace Explorer” double click the .cydwr file to open the Design Wide Resources Window.
7. On the right hand side, find Pin_LED and click the drop down under “Port” to select the pin for the blue LED, P3[7].
8. Build the project by clicking “Build” -> “Build [Project Name]”.
9. On the right side, double click the main.c file and place the code mentioned above in the file, deleting any other code that may be in the file.
10. Build the project again and then program the kit by clicking “Debug” -> “Program”.
11. If prompted, confirm the device being used in the “Select Debug Target” window that pops up.
12. Press the reset button on the board and observe.
13. If you want to see the code run step by step, click “Debug” -> “Debug” where you can click the “Step over” button to go step by step.

1.6 Observation

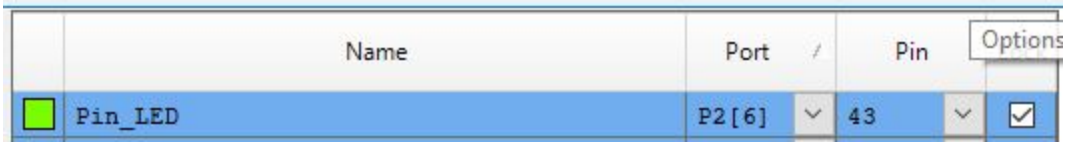
LED 3 on the board blinked blue as expected.

1.7 Summary

Overall, the lab was easy to implement and observe, though the initial setup process was a bit long. Overall, it took about three hours to set up the board, implement the lab, and write the report.

1.8 Exercise

1. **Instead of the Blue LED, blink the Red or Green LED.**
   The blue LED is pin P3[7], the red LED is P2[6], and the green LED is P3[6]. To blink the other colors, the Pin_LED needs to be changed to either the red or green pinout as seen below.



2. **Instead of blinking the LED, generate a constant white color from the RGB LED.**
   

   To make the LED white, the red, blue, and green LED all need to be turned on. To do this, add more pins to the schematic following the same process earlier. The schematic can be seen to the left. After adding the pins to the schematic, choose the pinout for each pin so that each one is one of the LED pins, as shown in the image below.

	Name	Port	Pin	Lock
	Pin_LED_Red	P2[6]	43	☑
	Pin_LED_Green	P3[6]	53	☑
	Pin_LED_Blue	P3[7]	54	☑

Lastly, the main.c code needs to be changed to remove the delay as to not turn off the LED. Also, all three LEDs need to be written to to make this work as well. The code for this is below.

```
#include "project.h"

int main(void)
{
      for(;;)
      {
         Pin_LED_Red_Write(~Pin_LED_Red_Read());
         Pin_LED_Green_Write(~Pin_LED_Green_Read());
         Pin_LED_Blue_Write(~Pin_LED_Blue_Read());
      }
}
```