

ECE383: Microcomputers – Lab 2

PIC24 System Schematic Creation in PCB Artist

Goals: The goals of this lab are to introduce students to the creation of a partial PIC24-based schematic and printed circuit board layout in PCB Artist.

1. Introduction

The purpose of this lab is to increase student familiarity with the PCB Artist software. Students will design the PIC24 power circuitry that will be used in later labs as well as create a printed circuit board layout of the design.

This lab requires you to capture portions of the screen. The lab computers use the Windows 10 operating system. This includes the “Snipping Tool” that may be used to capture portions of the screen. As always, read through the entire lab and scan any supplied files before starting work.

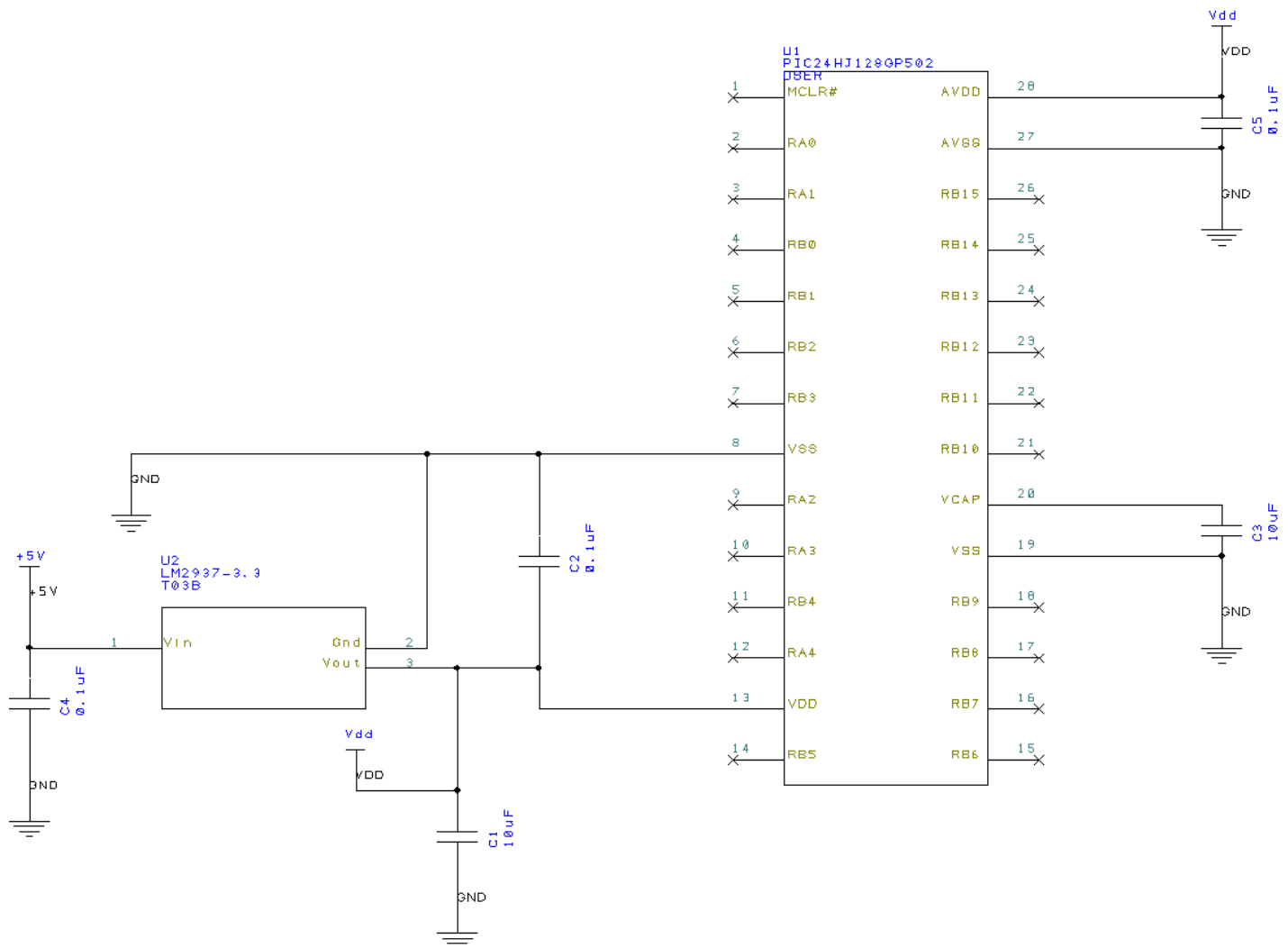


Figure 1. PIC24HJ128GP502 and Partial Power Circuitry

2. TASK 1: Voltage Regulator

The schematic in Figure 1 uses the LM2937-3.3 component as the voltage regulator for the PIC24 system. This is the device that will provide the power necessary for the PIC24 to operate correctly. Without a correct and stable power source, the PIC24 will either not operate at all or (even worse) operate with unexpected behavior.

Deliverable 1: Using the information available from DigiKey for this component (Hint: search for **LM2937ET-3.3/NOPB**) what is the voltage that will be output by this regulator and what is the maximum amount of current that it can supply?

3. TASK 2: Basic PIC24 Power Circuit

Using PCB Artist, create a basic power circuit for a PIC24 system. Your schematic should closely resemble the schematic shown in Figure 1. The ECE383 component libraries available on the class website include the PIC24HJ128GP502 microprocessor and the LM2937-3.3 voltage regulator components. The *Vdd* and *GND* components are located in the *schema* library. The capacitor (C) components, labeled C1-C5, are available in the *discrete* library. You will use multiple instances of the capacitor (C) component in this design. Note carefully that the capacitors have differing values and these values should be displayed in your circuit. The values for individual capacitors may be changed by right clicking on the component, selecting *Properties->Values* and setting the "Value=..." entry to the appropriate value. The capacitor's value is displayed in the circuit diagram if the radio checkbox for the "Value=..." entry is checked.

For the PIC24 we will also enter Manufacturer and Distributor information and part numbers. The values for individual components may be changed by right clicking on the component, selecting *Properties->Values* and setting the "Value=..." entry to the appropriate value. For the PIC24, the manufacturer will be "Microchip" and the Manufacturer part number will be PIC24HJ128GP502-I/SP. Use Digikey as the distributor and, using the Digikey website, find and enter the Digikey part number for the processor. Create a new value called "Website" and enter the Website URL for the Digikey website for the PIC24HJ128GP502-I/SP processor.

Deliverable 2: Use a screen capture tool to capture the schematic window for Task 2 and upload it to Blackboard. Make sure that your name and CWID are visible (in a text editor window) but do not cover your layout details.

4. TASK 3: Creation of a Printed Circuit Board Layout with PCB Artist

For this task we will create a printed circuit board layout from the PIC24 schematic created as a part of task 2. The board size should be 60mm x 60mm. The board part number should be “ECE383-LAB2” and the Revision Number should be “001”. These parameters can be set through the PCB wizard. Use your experience from previous labs to create a PCB similar to the diagram shown in Figure 2 below.

Further, add your initials and CWID on the top copper layer to the top left corner of your design and the text “ECE383-Fall2020-Lab2” on the top silkscreen layer to the bottom right corner of your design.

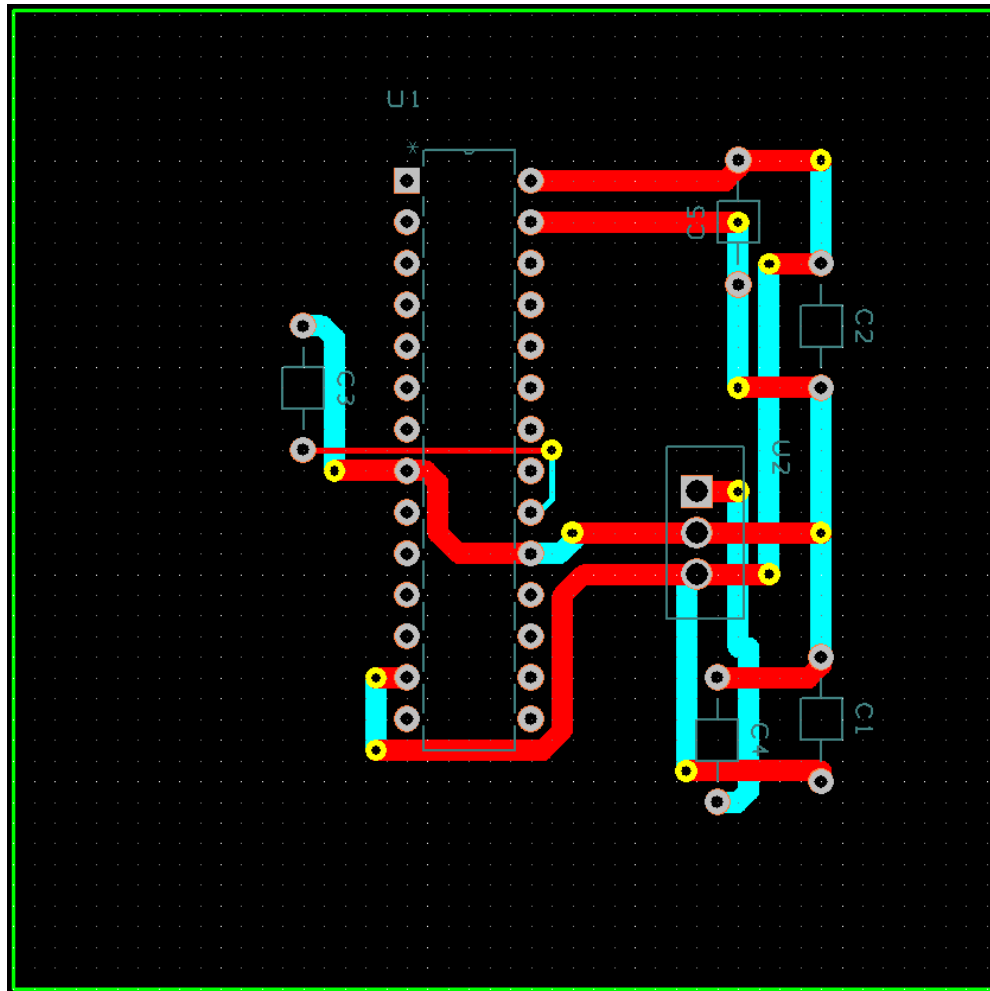


Figure 2. PIC24HJ128GP502 and Partial Power Circuitry Printed Circuit Board

Deliverable 3: Use a screen capture tool to capture the printed circuit board window for Task 3 and upload it to Blackboard.

5. TASK 4: Basic PIC24 System Schematic

Using PCB Artist, create a basic PIC24 system as shown in Figure 3 below. This expands on your earlier design to include a switch (SW1) and LED (LED1) which could be used for user inputs (switch) and notifications (LED). You may use your schematic from the previous task as a starting point for this design. The resistor (RA) components, labeled R1-R2, are available in the *discrete* library. You should use the SW_FSM2JH component for the pushbutton and the LED component for the single LED in the system. These components are also available in the *discrete* library.

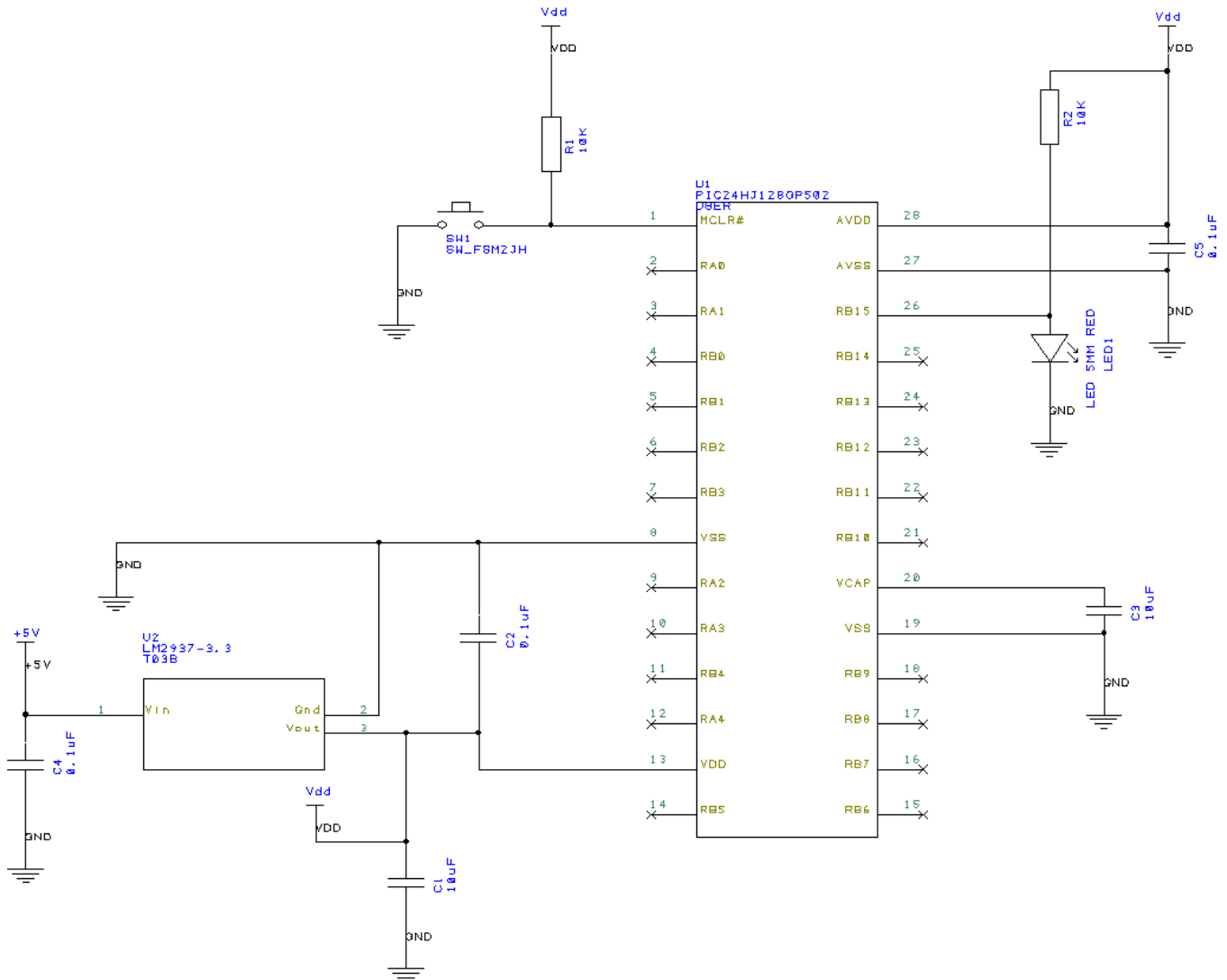


Figure 3. Basic PIC24 System Schematic

Deliverable 4: Use a screen capture tool to capture the schematic window for Task 4 and upload it to Blackboard. Make sure that your name and CWID are visible (in a text editor window) but do not cover your layout details.

6. TASK 5: PIC24 System Printed Circuit Board Layout

For this task we will create a printed circuit board layout from the PIC24 schematic created as a part of task 3. The board size should be 60mm x 60mm. The board part number should be "ECE383-LAB2" and the Revision Number should be "002". These parameters can be set through the PCB wizard. Use your experience from previous labs to create a PCB similar to the diagram shown in Figure 4 below.

Further, your initials and CWID on the top copper layer to the top left corner of your design and the text "ECE383-Fall2020-Lab2" on the top silkscreen layer to the bottom right corner of your design.

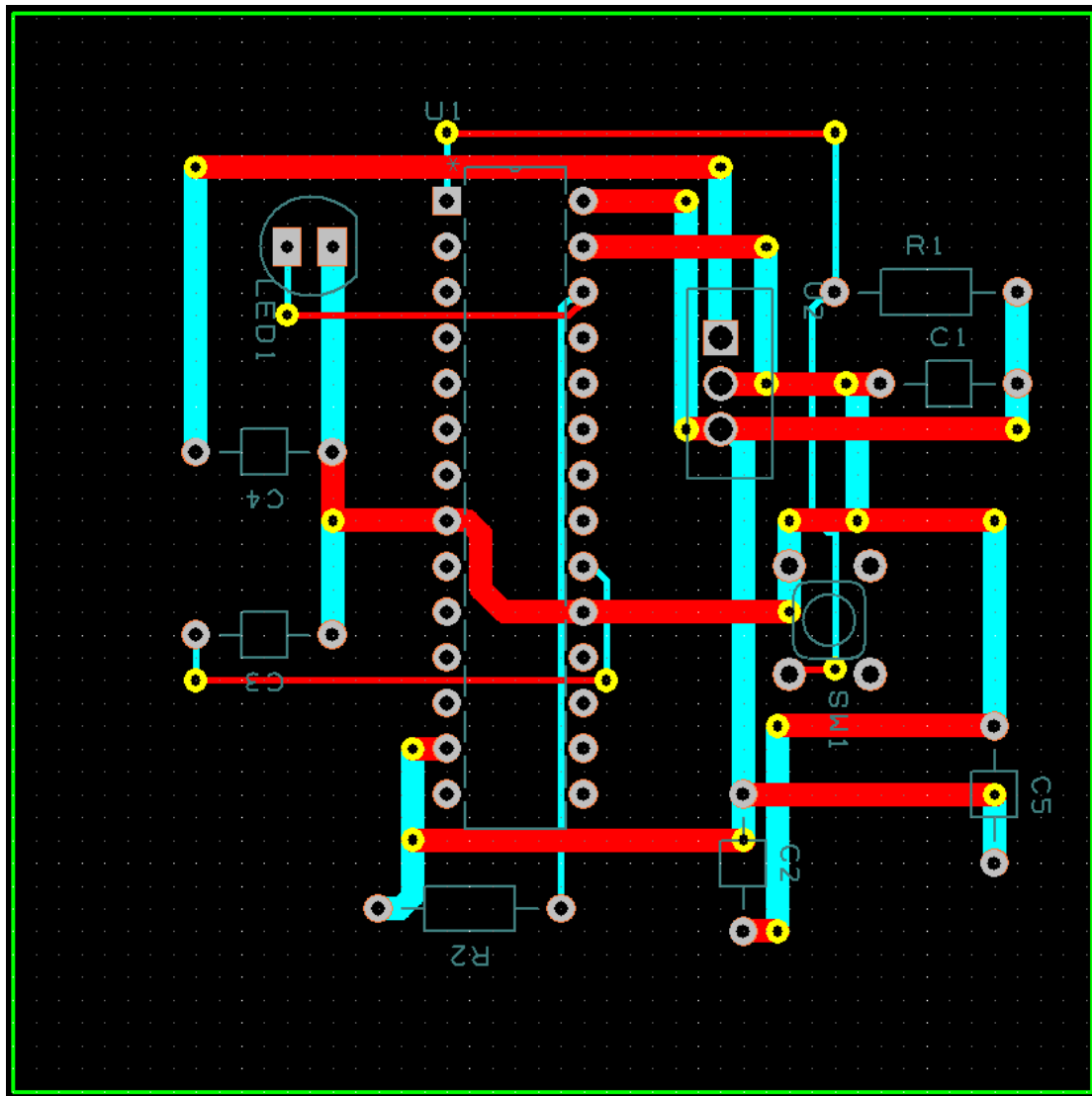


Figure 4. PIC24 System Printed Circuit Board

Deliverable 5: Use a screen capture tool to capture the printed circuit board window for Task 5 and upload it to Blackboard.