Department of Computer Engineering
Digital Hardware Sytems
CpE 2303L – Computer Engineering Drafting and Design

Practical Activity #1
Schematic Diagram Fundamentals

Objectives:
1. Know KiCAD's capabilities and features.
2. Familiarize KiCAD's graphical user interface.
3. Use the different tools in the PCB Editor.
4. Draw a schematic diagram using KiCAD PCB Editor

Tools Required:
The following will be provided for the students:
1. Computer with KiCAD 8 installed

Deliver Process:
The instructor will conduct a short briefing about the exercise. Knowledge from Unit II Schematic Diagram will be applied in this exercise, a short review of the recent chapter will be done.

Note:
Some of the contents in this manual are derived with permission from other sources. This document is a guide only. Your answers are to submitted in a separate document (lab exercise report).

Part I. Theory

A schematic diagram is a graphical representation of interconnections of various electronic, electrical, and electromechanical components of an equipment. Every PCB designer must learn how to read and interpret the schematic diagram. The schematic provides the most broadly used view of the design and includes all components. In addition:

- It gives visibility into the status of all parts of the design process;
- Schematics are the primary source for developing deliverables to product design and manufacturing groups;
- Design variants are built around slightly differing schematics;
- Test departments rely on schematics;
- Field service relies on schematics and
- Bills-of-materials are generated from schematics.

Figure 1 shows a proper schematic diagram drawn using EAGLE. The following are guidelines for drawing a schematic diagram:

- Signal flow moves from left to right across the page with inputs on the left and output on the right.
- Electronic potentials (voltages) should increase as you move from bottom to the top of the page.
- Use the 'unit number' convention for assigning a unique IC package identification.
- Components like resistors, capacitors and inductors should have a value with the correct unit.
- All components must have a symbol following the IEEE or IEC standard.

- Every component in the schematic should have a unique designator. Multiple components with the same designator must have must use a numbering system for example: R1, R2, R3 etc.
- Schematic symbols for electrical and electronic components should be standard.
- All power and I/O connections should be terminated with a connector.
- Documentations can be added in the schematic diagram if necessary.
- The title of the schematic diagram and the author, document number and version control should
- included.

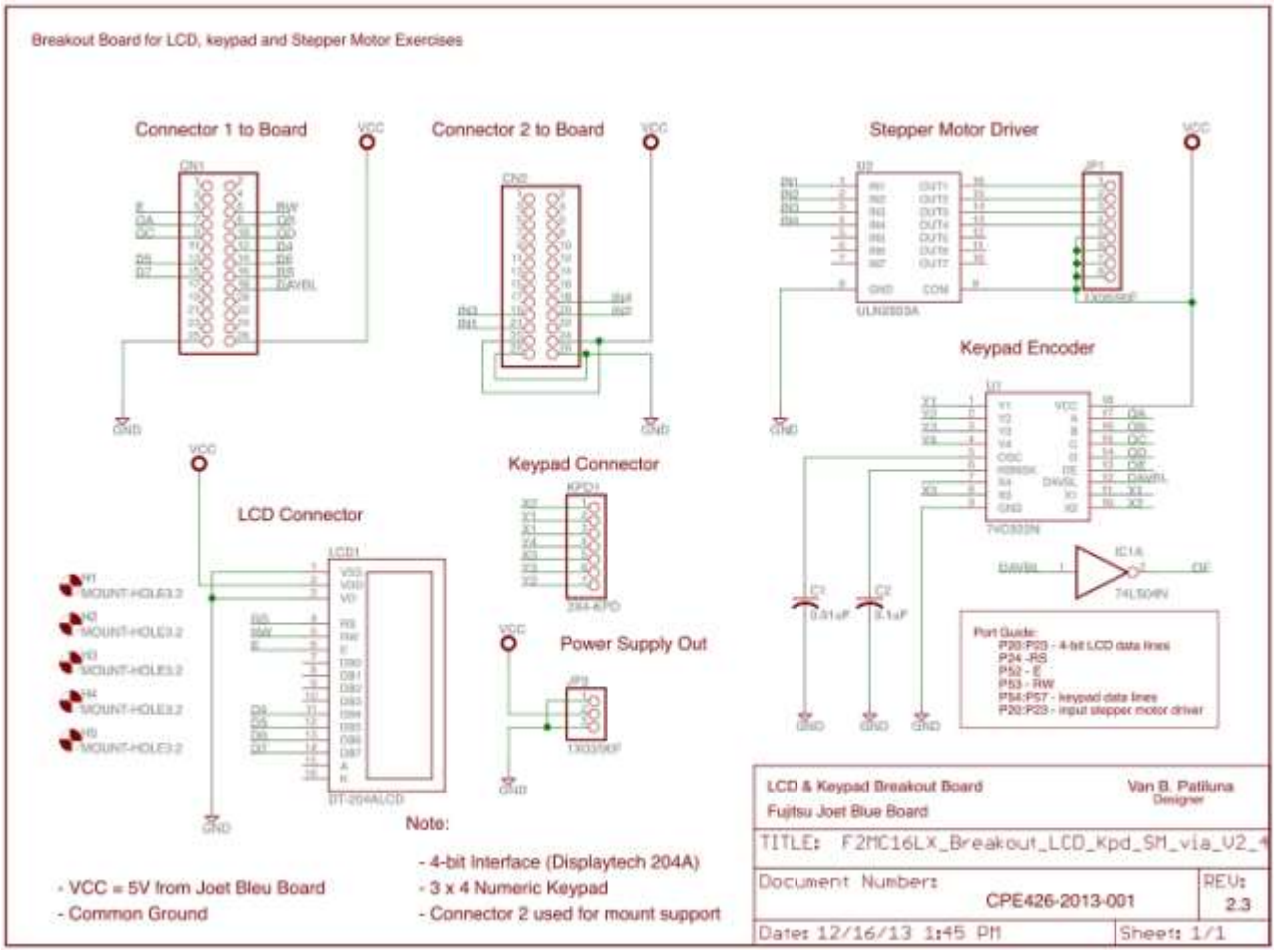

Figure 1 Schematic Diagram drawn using CAD Software.

Schematic Diagram Organization

Organize the schematic diagram by placing related circuits as close as possible. Identify circuit groups such power supply, connectors, microprocessor, I/O drivers etc. and if possible place them on different pages. Observe the signal flow convention: left (input) -> right (output). To make the schematic diagram neat and easy to read, do not criss-cross the nets, observe the signal flow convention. Use net naming to prevent criss-crossing (back-tracing) of nets.

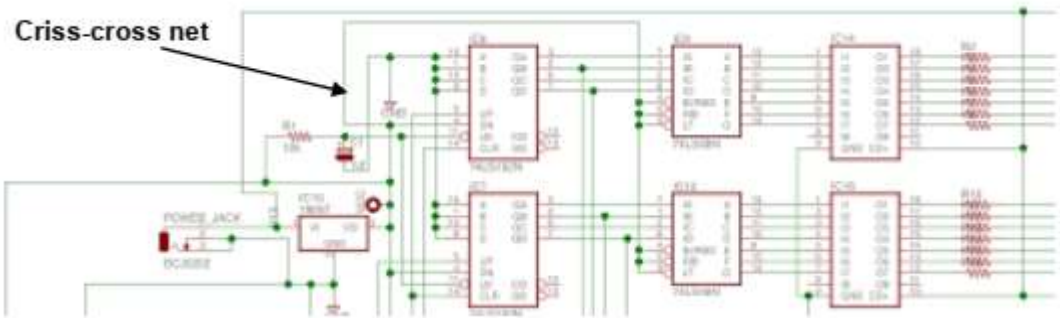

Figure 2 Part of a schematic with criss-crossing nets.

Running Electrical Rules Check

To check the electrical integrity of the schematic diagram, an ERC (Electrical Rules Check) can be run to automatically check the schematic diagram. It reports errors and warnings about illegal connections, unconnected pins, signals incompatibility, no supply to power pins etc. When errors or warnings are encountered, the schematic diagram must be edited to correct the errors reported. Figure 3 shows an example of an ERC report.

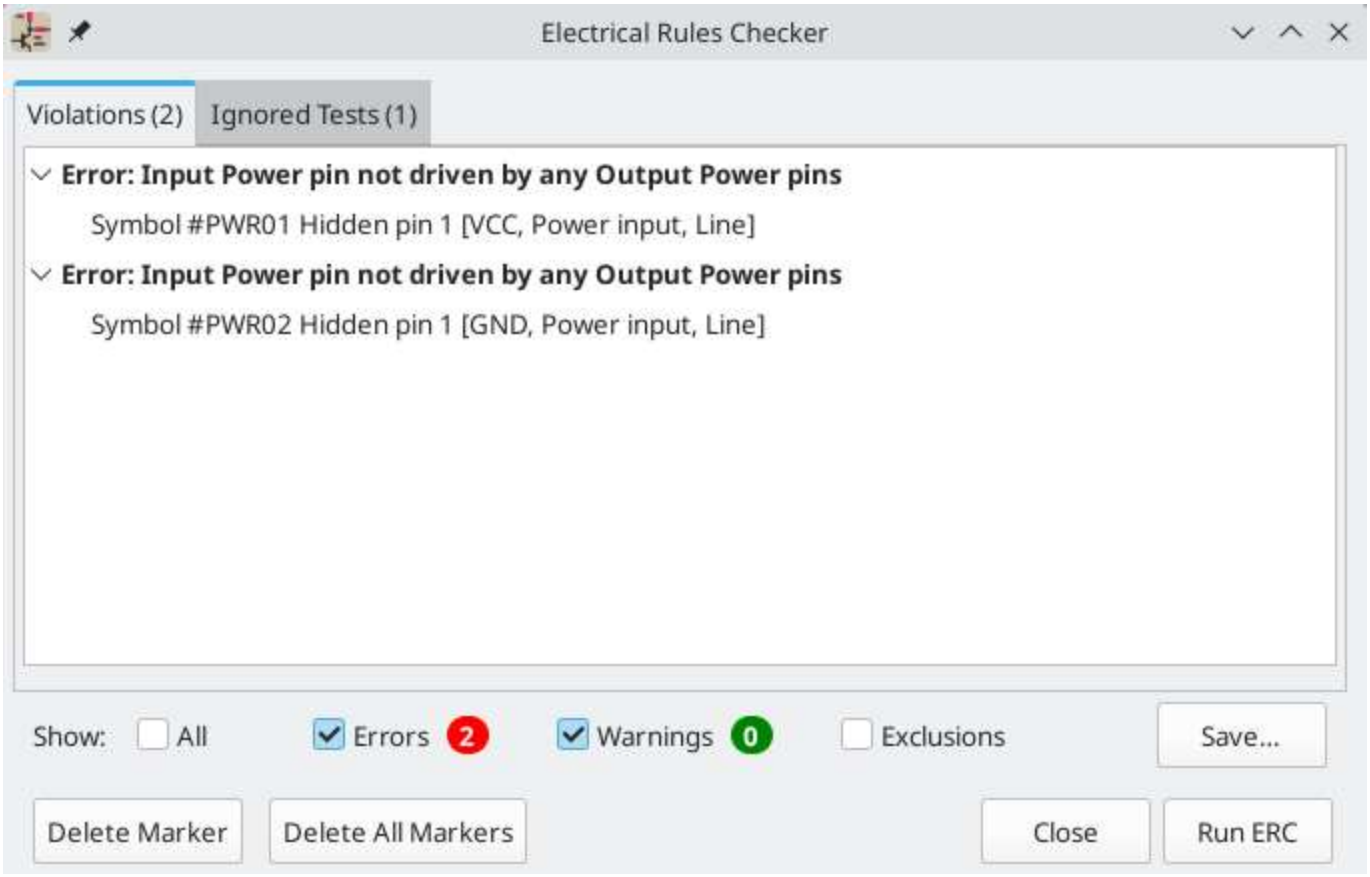

Figure 3 KiCAD ERC Report.

Once the corrections has been made, ERC can be run again to verify. Some warnings can be ignored but must be noted however errors must be fixed until no more reported during ERC.

Documentation

Some information about the circuit cannot just be shown using schematic symbols and therefore documentation can be added to the schematic diagram. Documentation should only be written on a blank part of the canvas.

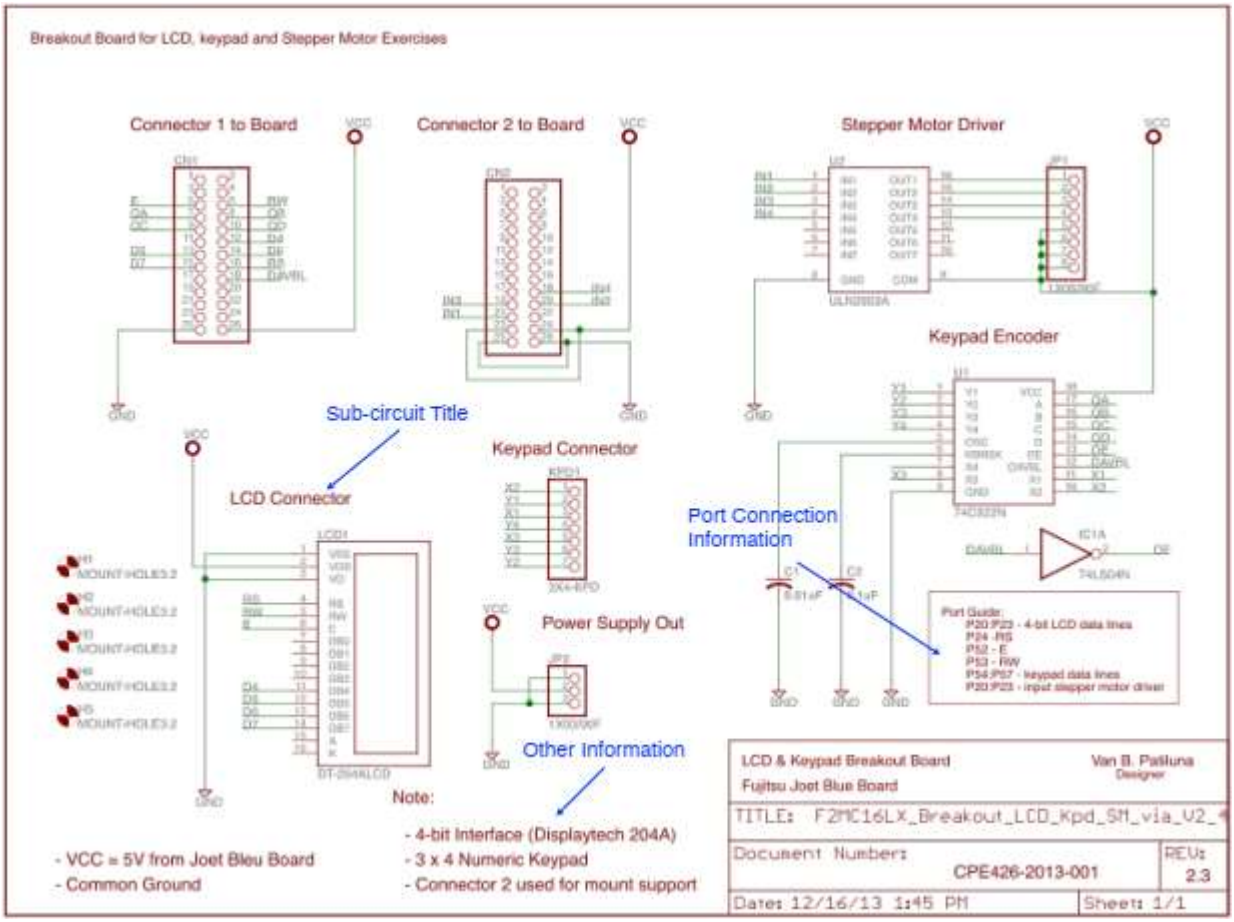

Figure 4 Documentation on a schematic diagram.

It should not interfere with the circuit diagrams and other information. Additional documentation may include circuit details, I/O ports guide, assembly details, package information, electrical specifications and change logs. Information about the functionality shall be limited or totally disclosed for confidentiality purposes if it is necessary. Documentations should be relevant to the schematic on the same page. Figure 4 shows documentation on a schematic diagram.

Information about the schematic diagram such as title of the schematic, name of the author, document number, date and time, sheet number and version control. Other information such as company name and department can be included.

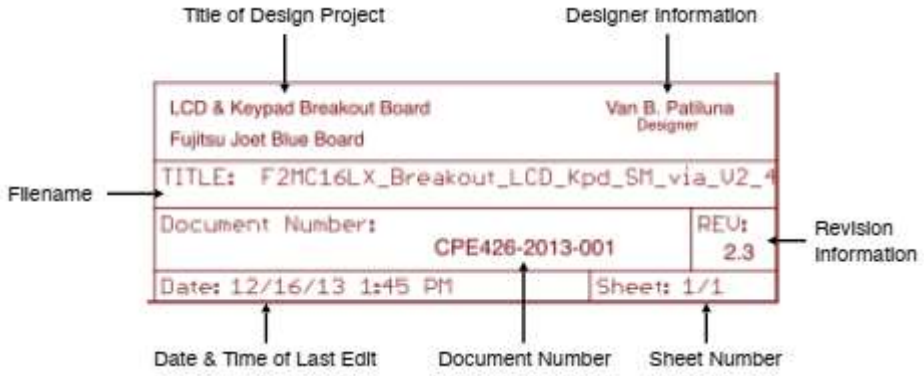

Figure 5 Schematic Information details.

Using NET Naming

Using nets allows interconnection of schematics especially with multiple pages. Nets have unique identifier in the form of N$X and can be renamed manually to rename a net, use the “Name” tool. Net names should not contain spaces and should start with an alphabet. It is important to understand that nets of the same name are connected. The schematic diagram in Figure 5 shows some nets are not visually connected but are logically connected using net names.

Using net names is also useful to minimize criss-crossing of nets across the schematic diagram. It can also be used to substitute for long nets.

Power and I/O Termination

Every electrical or electronic device requires power to function. Unless the power supply is part of the PCB, it necessary to terminate the power supply with a connector. Normally, a power supply connection has two pins for VCC and GND. If the circuit requires multiple power supply, then multiple connectors shall be used.

If an I/O device is external to the PCB then the I/O signals shall be terminated with a connector. There are hundreds of types of connectors but you should choose the connector compatible with the I/O device. For devices mounted on the PCB itself like an LED or a switch, no need for a connector.

Part II. Drawing a Schematic Diagram using KiCAD Schematic Editor

Before doing the exercise, make sure that you have listened to the demonstration of your instructor on drawing a schematic diagram using KiCAD.

Activity #1:

Figure 6 is a logic circuit with inputs A, B, C, D and E via the dual-inline package (DIP) switch. The output is a single LED.

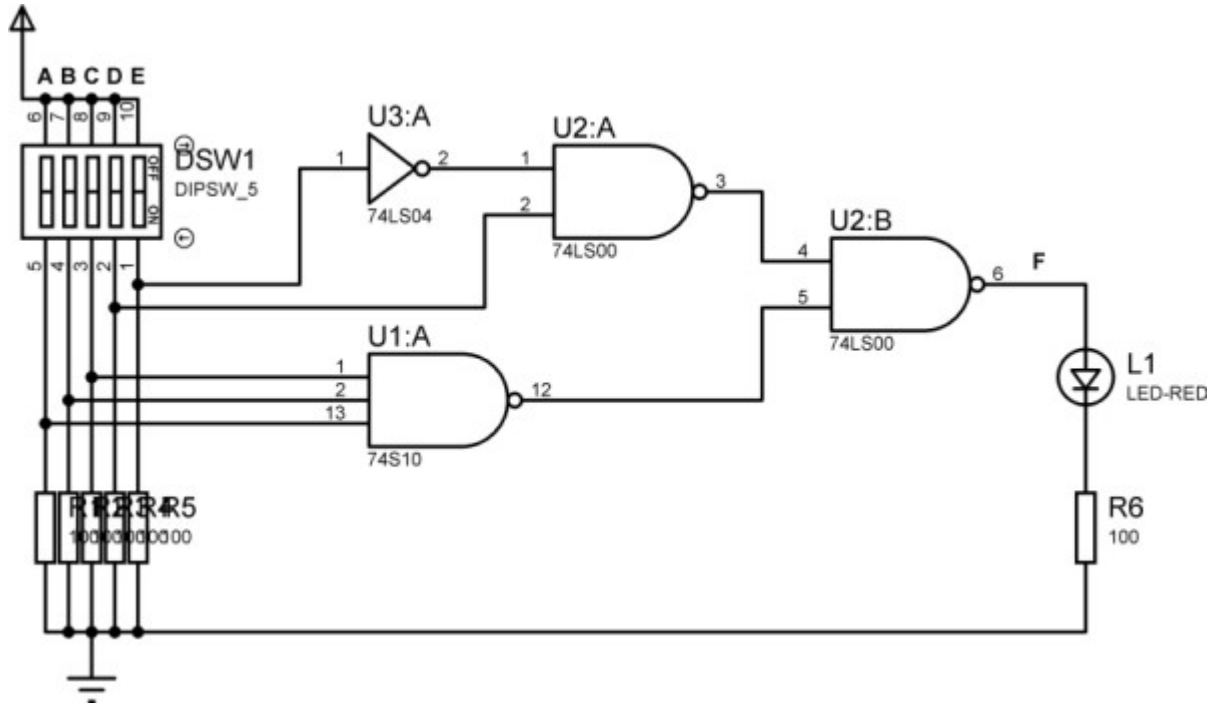

Figure 6 Logic circuit example #1.

Instructions:

1. Draw the circuit in **Figure 6** using EAGLE schematic editor. Use standard and agreed schematic symbols and all components shall have through-hole packages. Replace non-standard designators. Properly terminate the power supply connection. Refer below for the library references of the components. As mentioned, all packages are through-hole.

   DIP switch : library: “special”, name: SW_DIP-5
   NAND gate : library: “74xx-us”, name: 74LS00N
   NOT gate : library “74xx-us”, name: 74LS4N
   LED : library: “led”, name: LED5MM
   Resistor : library: “rcl”, name: R-US_0204/7

   Note: When using a gate such as 74LS08 (quad-input AND gate), KiCAD will invoke a single gate from an IC. The gates will be designated IC1A, IC1B, IC1C and IC1D but all of them belong to an IC called IC1. If you add another AND gate, it will another 74LS08 IC. Use the “Invoke” tool to invoke a specific gate in an IC.

2. The required frame size is LETTER_L (which can be found in the “frame” library). Add the required information below. All documentations should be at the “Symbols” layer.

   Title of the circuit: Logic Circuit Example #1
   Designer: <Your Name>
   Document No: CpE 2303L-S24-002
   Revision: 1.0

3. Run ERC and make sure errors and warning are addressed.

4. Save the schematic diagram with the filename format: "<YOUR LAST NAME>_PA1-1.sch". For example "PATILUNA_PA1-1.sch".

Activity #2:

Figure 7 is a logic circuit with inputs x and y via the dual-inline package (DIP) switch. The output is a single LED.

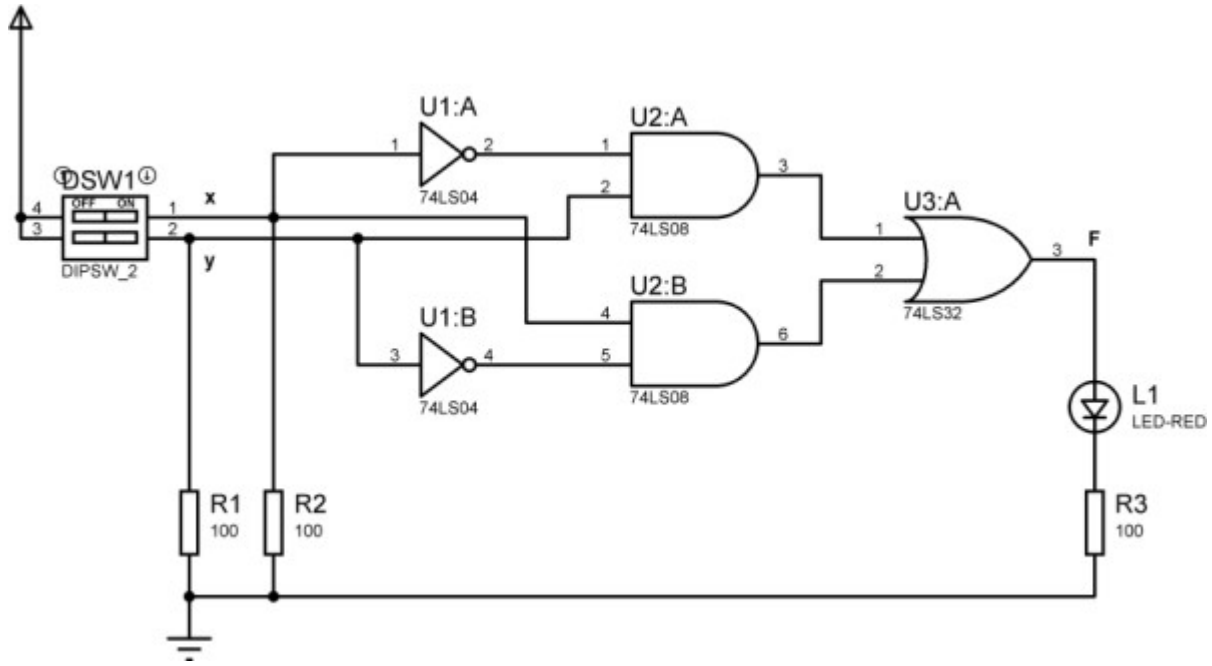

Figure 7 Logic circuit example #2.

Instructions:

1. Draw the circuit in **Figure 7** using EAGLE schematic editor. Use standard and agreed schematic symbols and the required packages. Replace non-standard designators. Properly terminate the power supply connection. Refer below for the component packages (search the component and proper package in the library).

   Gates : surface mount (SOIC)
   Resistor : surface mount (chip)
   LED : surface mount (chip)
   Switch and connectors : through-hole

   Note: Search for the components and correct packages.

2. Add a frame and the required information as shown below.

   Title of the circuit: Logic Circuit Example #2
   Designer: <Your Name>
   Document No: CpE 2303L-S24-003
   Revision: 1.0

3. Run ERC and make sure errors and warnings and addressed.

4. Save the schematic diagram with the filename format: “<YOUR LAST NAME>_PA1-2.sch”. For example “PATILUNA_PA1-2.sch”.

Activity #3:

Figure 8 is an MCU-based two-digit counter. The outputs are two common-anode seven segment display with a push button switch (SW2) to advance the counter. The other switch (SW2) is an MCU master reset.

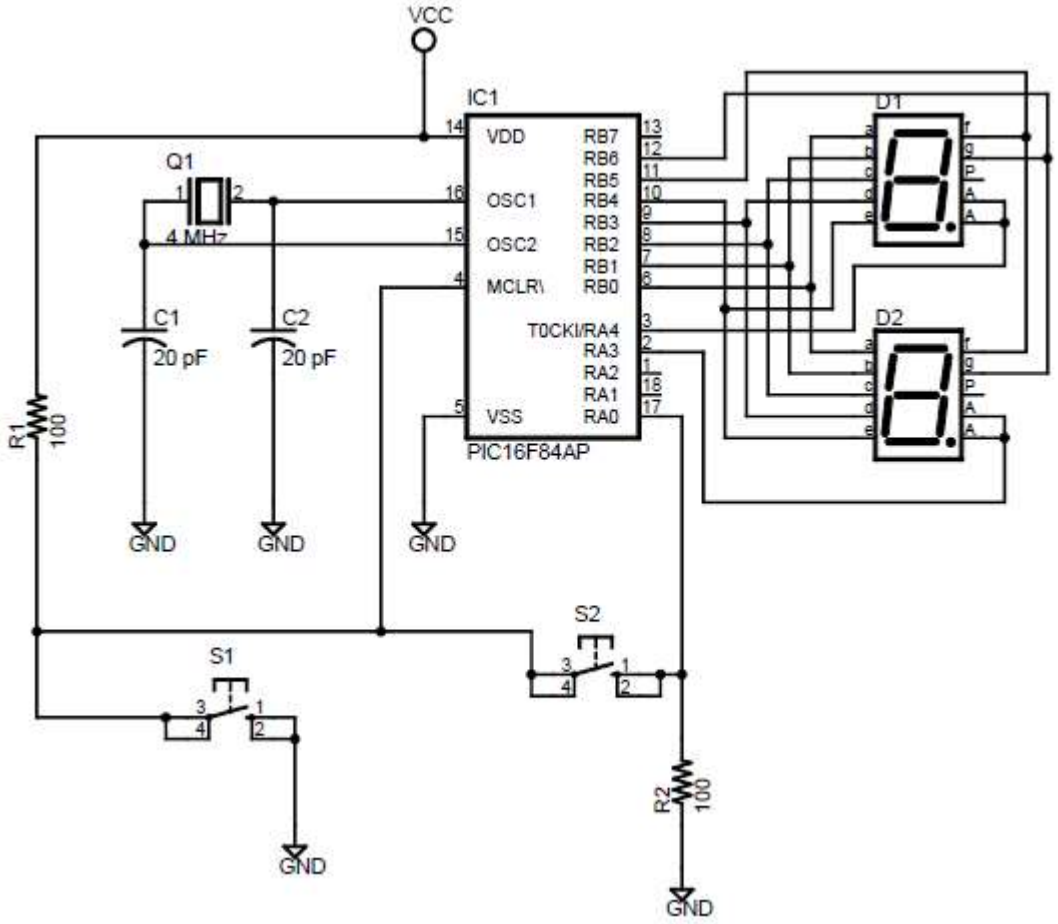

Figure 8 MCU-based two-digit counter.

Instructions:

1. Draw the circuit in Figure 8 using EAGLE schematic editor. Use standard and agreed schematic symbols and all components shall have through-hole packages. Replace non-standard designators. Properly terminate the power supply connection. Refer below for the library references of the components.

Push-button switch	: library: “switch-omron”, name: B3F-10XX
Crystal	: library: “special”, name: XTAL/S
MCU (PIC16F84A)	: library: “microchip”, name: PIC16F84AP
7-segment (CA)	: library “special”, name: 7SEG-CA

2. Add a frame and the required information as shown below.

   Title of the circuit: Logic Circuit Example #2
   Designer: <Your Name>
   Document No: CpE 2303L-S24-003
   Revision: 1.0

3. Run ERC and make sure errors and warnings and addressed.

4. Save the schematic diagram with the filename format: “<YOUR LAST NAME>_PA1-3.sch”. For example “PATILUNA_PA1-3.sch”.

Submissions Instructions:

Print all the schematic diagrams by clicking Files->Print. Click “Printer” and choose “Microsoft Print to PDF” as a printer. Click “Preferences” and make sure that the orientation is landscape and the paper size if US LETTER. Click “Ok” then click “Print”. To print click “OK” and save the file to the same folder you save your schematic diagrams.

Combine the three (3) schematic diagrams into a single PDF file1. The filename format should be “<YOUR LAST NAME>_PA1.pdf” for example “PATILUNA _PA1.pdf”. Submit the PDF file to the activity page in Canvas.