EE128: Data Acquisition, Instrumentation and Process Control
University of California, Riverside

Lab 1: Schematic Drawing Exercises

Spring 2019

Objective

To gain experience with using Cadence-Design CIS on the workstation to create electronic schematics. Subsequent labs will require schematic drawings for lab reports; this set of exercises provides the necessary skills.

Equipment

- PC running MS Windows
  - Use virtual machines, e.g., Virtual Box, if you have a Mac
- Cadence-Capture CIS
- Additional Cadence libraries (Cadence_libraries.zip)
  - Downloadable from the course website
  - Unpack it in the PSpice library folder of Cadence (e.g., C:\Cadence\SPB_17.2\tools\capture\library\pspice)

Lab Computing Environment

By the BCOE computing policy, students are expected to have their own laptops. Use your laptops in lab sections. See "Software_Downloads_for_Labs.pdf" and install relevant software tools (only Cadence is needed for Lab 1).

Lab computers: If you plan to use desktop computers in the lab, you should first log in using your BCOE account and password. Upon logging in, your personal BCOE account should be mapped as a "Z drive" as part of the computer. Cadence-Capture CIS is already installed on the lab computers.

You should create any project on your BCOE account drive (aka "Z drive"). By doing so, your files are backed up automatically. DO NOT PLACE YOUR PROJECT FOLDERS ON THE C: DRIVE. Files left on the C drive may be removed at any time. The ECE department and the engineering school cannot be held responsible for lost files.

For security reasons, do not leave a computer running under your account without your presence. Make sure to log out of the system before leaving the lab.

Lab Exercise

PART 1.

1. Open **Cadence Capture CIS** (Start > All Programs > Cadence Release > Capture CIS). Select Capture CIS and click OK at the *Cadence Produce Choices* window. Then click **File > New > Project**. Select **Schematic**. Enter a name for the file (e.g., Ex1), change the location to your Z drive (e.g., Z:\EE128\) if using Lab's PC or your local drive (e.g., C:\EE128\) if using your laptop, and click OK.

2. Open the **PAGE1** schematic, if it isn't open already. We will draw the schematic shown in Figure 1.
3. Click the **Place part (P)** icon or click on **Place > Part** from the top menu bar.
4. In the Place Part tab, click on **Add Library** button and add all libraries in the **PSpice** library folder. Then make sure all libraries are selected.
5. Begin typing **555alt**, notice the list below automatically narrows down with each new letter.
6. Double click the correct part from the list to place the part on your schematic page.
7. Repeat for resistors (**R**), capacitors (**C** & **C_elect**), IC (**IC1**) and Vdc (**VDC**). Place one non-polar capacitor and one polar capacitor. Select the part and right click, then click **Rotate** to rotate the part if necessary.
8. Click the **Place ground (P)** icon or click on **Place > Ground** from the top menu bar. Begin typing **GND** and double click the correct part from the list to place the part on your schematic page. Repeat for power (**VCC_BAR**).
9. Add properties (e.g., name, value, etc.) to all the parts.
10. Wire all the parts as is shown in Figure 1.
11. Save your schematic drawing. This schematic should be included in your lab report.

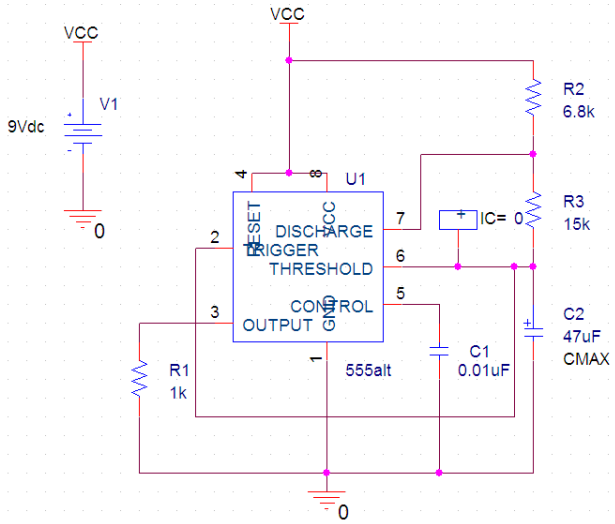

Figure 1. Timer Schematic

PART 2.

1. Create a new schematic project in **Cadence Capture CIS**. We will draw the schematic shown in Figure 2.
2. Click the **Place part (P)** icon or click on **Place > Part** from the top menu bar.
3. In the Place Part tab, click on **Add Library** button and add all libraries in the **PSpice** library folder. Then make sure all libraries are selected.

4. Search for **MC9S12DJ128BCPV.** If it exists, place it on your schematic page and continue the next step. Otherwise, perform the following:
5. Place a DIP switch (**SW_DIP_2)**. If it does not exist, perform Step 5(a) to find it (enter **DIP switch** in the description field, instead of the part number field).
6. Place resistors (**R**), two 4-bits LED bars (**LEDARR_IS_4**), and a function generator (**VCO_sqr**).
7. Click the **Place ground (P)** icon or click on **Place>Ground** from the top menu bar. Type **GND** and double click the correct part from the list to place the part on your schematic page.
8. Wire all the parts as is shown in Figure 2.
9. Save your schematic drawing. Include this schematic as **Part 2-A** in your lab report. This circuit is for an interrupt handling example of the 9S12DG256 microcontroller.
10. Alter your schematic by moving the LEDs to **Port B** (PB0-7). Include this schematic as **Part 2-B** in your lab report.

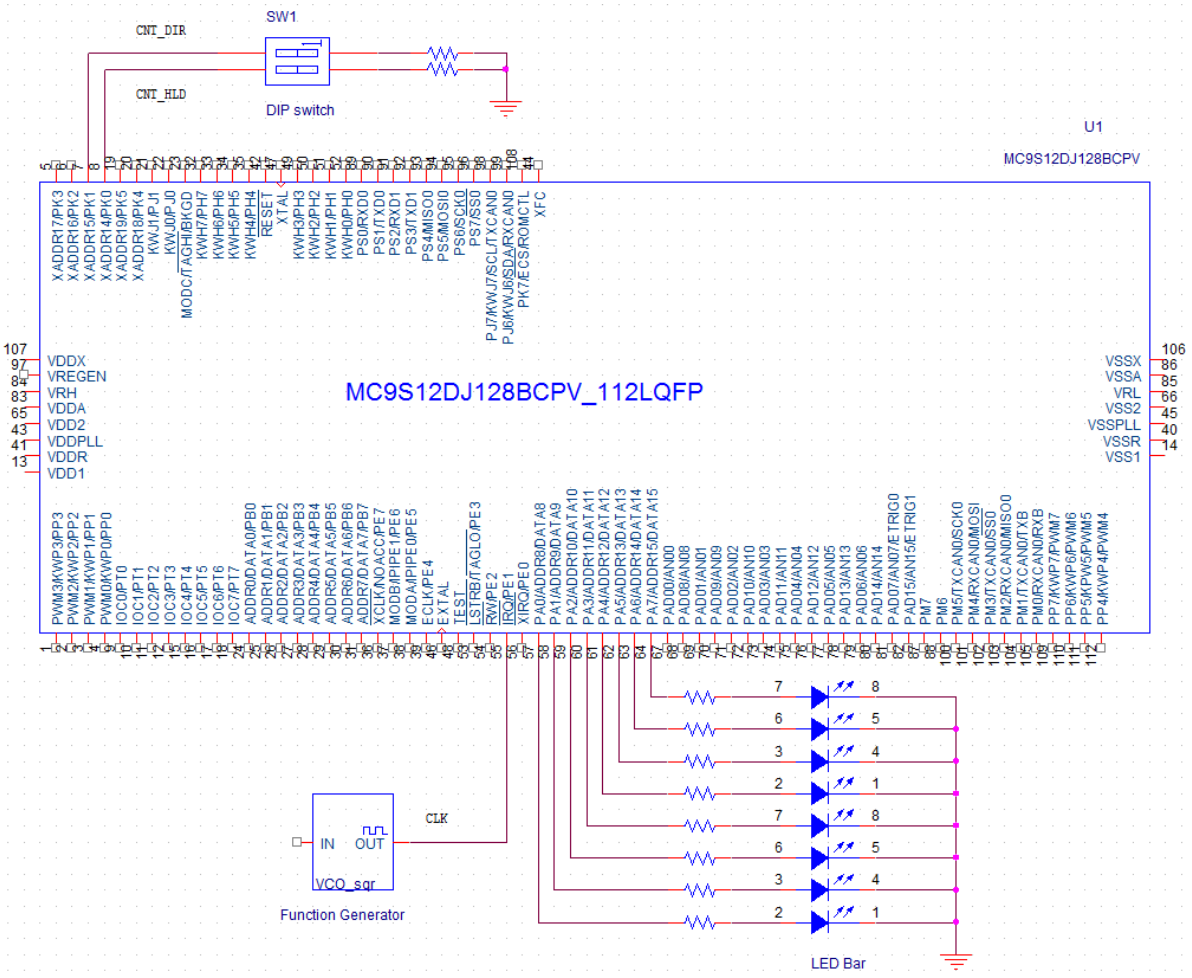

Figure 2. Interrupt Handing Circuit Schematic

Lab Demo (20 points)

- Show your schematics to TA and get a confirmation of completion

Lab Report (20 points)

Make sure to include the following in your lab report **(Schematics: 10 points; others: 10 points)**:

- A very short abstract (objectives and accomplishments)
- Experiment system specification (what is to be designed and implemented)
- Schematic diagrams (**Part 1, Part 2-A,** and **Part 2-B**)
- Technical problems encountered, and how they are solved
- A very brief conclusion

* Lab 1 total possible points: 40 points