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

Practical Activity #3
Mechanical Design Features

Objectives:
1. Implement the different mechanical design features.
2. Use KiCAD's PCB Editor tools to implement the features.
3. Create a PCB layout with the required mechanical design features.

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 IV Mechanical Design Considerations 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

Copper Zone

This method fills the idle areas (areas without tracks, holes or pads) of the PCB with copper and isolating the tracks and pads from the fill. The copper fill surface also could serve as a ground plane for single and double-sided design, thus providing excellent ground for the circuit. It also provides shielding of clock signals within the board. This also affects the production process by speed it up since it requires less copper to etch.

However, it can cause manufacturing difficulty especially when not using professional equipment. Tracks are prone to bridging or accidental short circuit especially when there is minimal isolation. A hatch fill consists of horizontal and vertical wires spaced evenly while a solid fill covers the idle area entirely with copper. Figures 1 shows the different types of fill.

Figure 1 Hatch Fill (left) and Solid Fill (right)
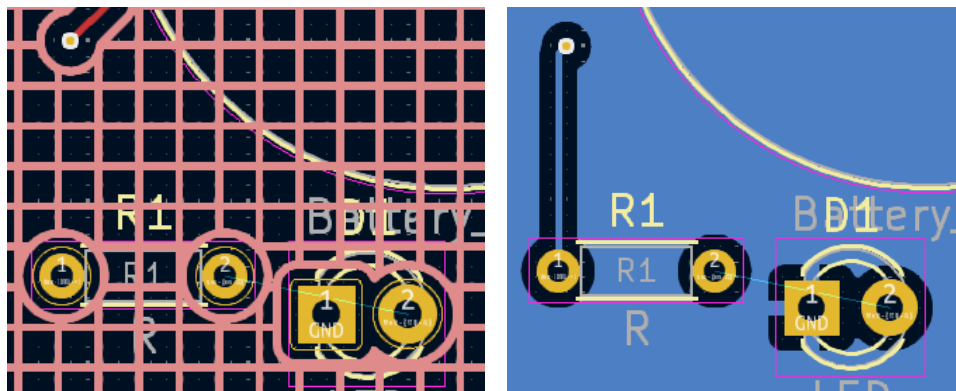
The clearance area is provided during "Fill all Zones" to prevent tracks, pads and vias from shorting to the fill. Recommended minimum clearance is the same to the widest track in the design.

Mounting Holes

Mounting holes are necessary if the PCB will be assembled on a chassis or casing. These holes can be also be connected the ground plane which extends the ground to the mounting chassis. It can be added holes can be added in the schematic (if grounded) or directly to the board. Holes size depends on the bolt or screw diameter (M2, M3 etc).

Ungrounded mounting holes an ordinary hole without copper plating or pad. The hole will be isolated if the board is copper filled. On the other hand, grounded mounting holes have pads which is connected to ground and the hole can also be plated. Figure 2 shows the different type of holes.

Figure 2 Ungrounded Hole (left), Grounded hole (right)

To provide good mechanical stability, the board should be supported within 25 mm of the board edge on at least three sides. As general reference, boards between 0.031 inch and 0.062 inch (0.785 mm and 1.57 mm) thick should be supported at intervals of at least 10 cm. If the mounting hole is grounded, the distance from mounting hole pad to edge should be observed (40 mils or 1.016 mm standard). However, if the mounting hole is ungrounded, the distance between the center of the hole will be: (hole radius + minimum copper to edge distance).

Power and I/O Termination

Most of the PCB does not have a power source as a component. It relies on external power source like batteries or voltage regulators. Therefore, it is important that power supply must have a terminator such as connector or a socket.

Figure 3 Power Input Options

PCBs like the motherboard requires connection to the hard drives, front panel switches and LEDs, front panel USB, fans, displays, input devices etc. These connections must be terminated with proper connectors. The choice of the connector depends on the requirements and the connector of the external I/O device.
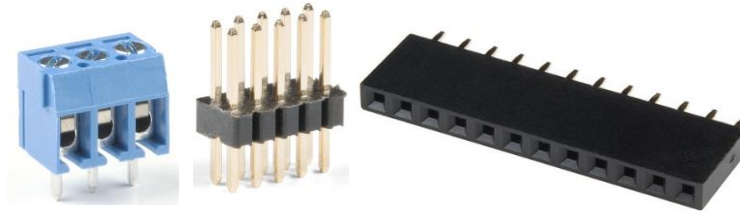
Figure 4 Screw Terminal (left), Header Pin Male 2x5 (center),
Header Pin Female (right)

Part II. Creating a PCB Layout using KiCAD PCB Editor

Before doing the exercise, make sure that you have listened to the demonstration on implementing mechanical design features using KiCAD PCB Editor.

Activity #1:

For this activity, use the schematic diagram as used in the demonstration. A copy of the schematic is available in your network drive.

Instructions:

1. Create a PCB layout from the schematic. Use only the bottom layer for a single-sided PCB design. Estimate the size of the PCB to fit the components. Make sure that the power and I/O are properly terminated.

2. Add a solid copper zone on the bottom layer with a 10 mil isolation.

3. Run DRC and make sure errors and warnings are addressed.

4. Edit the silkscreen and arrange the component designators (name). You may add documentation on the silkscreen to indicate the function of the connectors. On a blank portion within the bottom layer, add the following (without the quotation marks). Replace 'y' with the document number in the schematic and the 'x' with the iteration number of your design.

   "CpE 2303L-2024-yyyy Rev 1.x"

   Adjust the font size if necessary.

5. Save the the PCB layout with the same filename format as of the schematic diagram (except for the file extension): "<YOUR LAST NAME>_PA3-1.kicad_brd". For example "PATILUNA_PA1-1.kicad_brd"

Activity #2:

For this activity, use the PCB layout in Activity #3 (Practical Activity #2 – "MCU-based two-digit counter") with filename "PATILUNA_PA103.kicad_brd".

Instructions:

1. Add four grounded mounting holes with a diameter of 3 mm. You may need to edit the layout to accommodate the holes.

2. Add a solid copper zone on both layers with a 10 mil isolation.

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

4. Edit the silkscreen and arrange the component designators (name). You may add documentation on the silkscreen to indicate the function of the connectors. On a blank portion within the bottom layer, add the following (without the quotation marks). Replace 'y' with the document number in the schematic and the 'x' with the iteration number of your design.

   "CpE 2303L-2024-yyyy Rev 1.x"

   Adjust the font size if necessary.

5. Save the the PCB layout with the same filename format as of the schematic diagram (except for the file extension): "<YOUR LAST NAME>_PA3-2.kicad_brd". For example "PATILUNA_PA3-2.kicad_brd"

Submissions Instructions:

Layout Printout

Print all the PCB layout 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 and PCB layouts.

Combine the three (3) PCB layouts into a single PDF file1. The filename format should be "<YOUR LAST NAME>_PA2.pdf" for example "PATILUNA _PA2.pdf". Submit PDF to the Practical Activity #2 page in Canvas.

Schematic and Layout Files

Submit the **schematic** (.kicad_sch) and **layout** (.kicad_brd) files for the three activities. Do not archive or zip the files.