



Practical Activity #2
PCB Layout 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 III PCB 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 be submitted in a separate document (lab exercise report).

Part I. Theory

The PCB Editor is part of the suite of applications for the Electronic Design Automation software KiCAD. It allows the design from the Schematic Editor to be converted into a PCB layout in a few steps. Here are some of its features:

- Allows Manual routing.
- Design Rules Check with expansive configurable parameters.
- Support copper zones for surface area maximization.
- Scripts for component placement, dimension, and routing procedures.
- Forward and backward annotations with schematic designs.
- Supports up to 32 layers in the board stackup.

Figure 1 shows an example of a PCB layout down in EDA application.

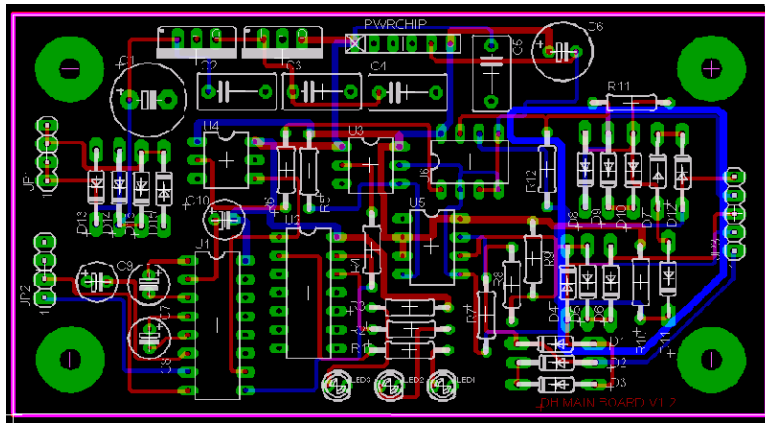


Figure 1 PCB Layout created using EAGLE.

Design Rules Check

A PCB design project has requirements for quality control, manufacturing constraints and operational stability of the board. To aid PCB designers in making sure that the requirements and other constraints are met, a Design Rules Check function is used. It can check for:

- Minimum track width
- Minimum hole size
- Clearances (track-track, track-pad, pad-pad, track-smd, etc.)
- Distances (copper to dimension)
- Restring
- Layers
- Shapes
- Other Requirements.

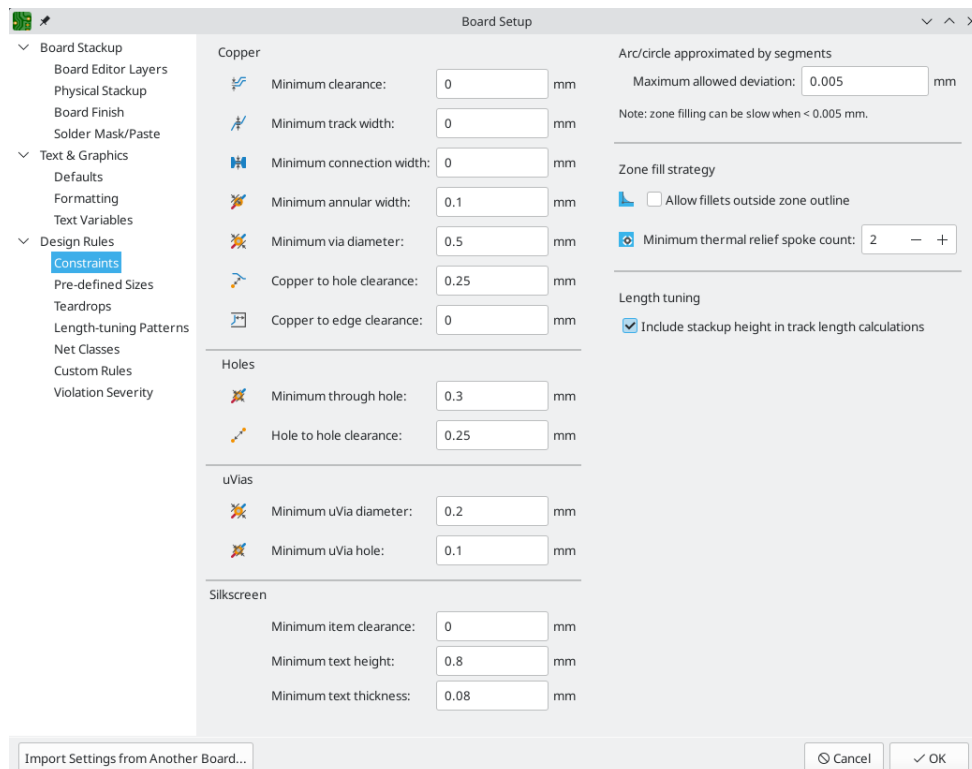


Figure 2 DRC constraints.

PCB Dimension

PCB must have a specific dimension. Some design will have unusual shapes but still exact measurements must be considered. All components, tracks, labels, and other documentations must be within the dimension. The size of the PCB depends on the mounting medium or enclosure where it will be mounted.

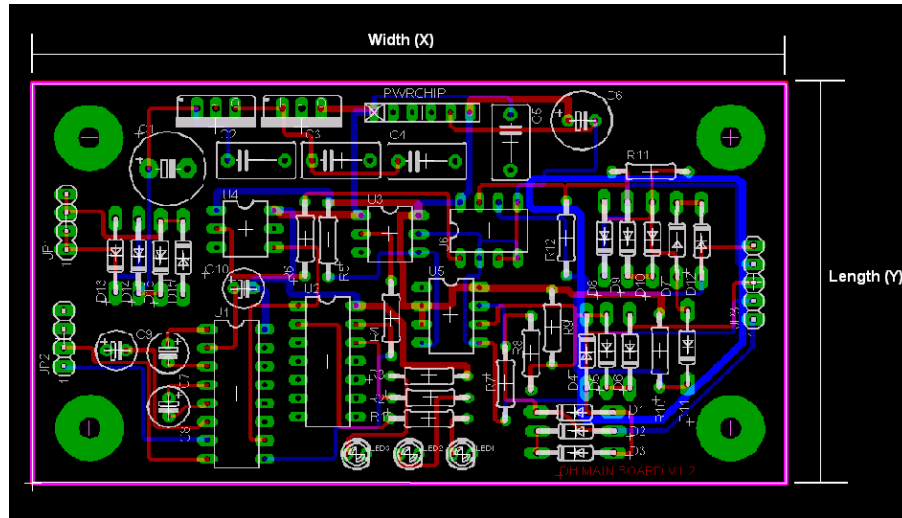


Figure 3 PCB dimension.

To create a board dimension in PCB Editor, choose the correct layer, "Edge.Cuts". Then go to the Rectangle Tool, to draw a rectangle. Get the rough size of the rectangle correct, as you move the cursor in the canvas, it will show the current size.

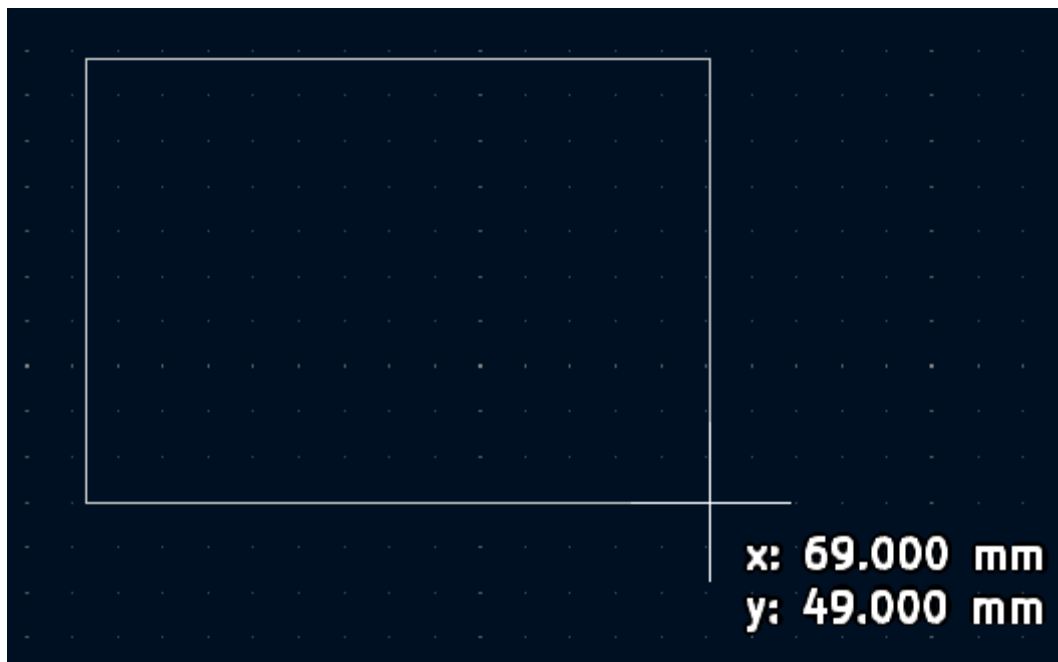


Figure 4 Drawing the dimensions in PCB Editor.

Component Placement

Component placement depends on the type of circuit (analog or digital), type of components, board performance, stress condition, casing type and size and thermal consideration. These factors may affect the positions of the component on the board.

The following are the general guide for placing components within the dimension:

- On-board connectors such as the D-type connectors, RJ connectors etc, must be placed at the edge of the board. This for easy access to the connectors.
- Power supply circuit must be isolated from the rest especially in digital circuits.
- If possible, all ICs must be placed in a common orientation (vertical or horizontal).
- Components must be parallel to the board dimension.
- To save board space, resistors and diodes can be placed side by side horizontally.
- For easier routing, always refer to the schematic diagram to determine the proximity of the components.

Track Routing

There are two ways of routing the traces in a PCB: manual and automatic. Manual routing provides total control of the design which minimizes the number of vias and the size of the PCB. It also allows design constraints to be set properly. However, when the circuit becomes complicated, manual routing can be very tedious and time consuming.

On the other hand, auto-routing automatically routes the PCB without any effort from the PCB designer but the latter can configure the routing algorithm and other parameters. The number of vias, layers, routing directions can be set. However, auto-routing might take some time for complex designs and does not guarantee 100% routing. It also require editing of the layout after routing.

Editing Silkscreen and Adding Documentation

Like the schematic diagram, you can also add documentation to the PCB layout either on the silkscreen or at the copper itself. Editing also of the silkscreen layer can also be done to make sure the markings does not overlap a pad or each other. The following must be followed when editing silkscreen and adding documentation:

- Component values are recommended to be hidden, only the component designator must remain.
- When editing silkscreen, do not overlap texts to other objects in the silkscreen or pads.
- Do not put component designators under a component. It must be visible after the PCB has been assembled.
- When adding documentation to the copper layer, do not overlap it to tracks and pads.

General Steps in Creating a PCB Layout

To create a PCB Layout in KiCAD or an CAD Software. Some steps must be inserted in between if necessary.

1. Set the PCB Board Stackup.
2. Set the Design Rules Checker constraints.
3. Set the PCB Dimension.
4. Place the components.
5. Rout the airwires.
6. Run DRC to check for violations.
7. Edit silkscreen (component names and values).
8. Adding documentation (in silkscreen or copper layer).

Part II. Creating a PCB Layout using KiCAD PCB Editor

Before doing the exercise, make sure that you have listened to the demonstration of how to create a PCB Layout using KiCAD PCB Editor. Copy the schematic diagrams into a new project titled PA2. Launch the schematic diagrams from the PA2 project.

Activity #1:

Create a PCB layout from the schematic diagram in Practical Activity #1 (Activity #1).

Instructions:

1. Draw 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 power and I/O are properly terminated.
2. Run DRC and make sure errors and warning are addressed.
3. Edit the silkscreen and arrange the component designators (name). You may add documentation on the silkscreen to indicate the inputs A, B, C, D, E the power and I/O terminators. On a blank portion within the bottom layer, add the following (without the quotation marks). Replace 'x' with the iteration number of your design.

"CpE 2303L-2024-001 Rev 1.x"

Adjust the font size if necessary.

4. Save the the PCB layout with the same filename format as of the schematic diagram (except for the file extension): "<YOUR LAST NAME>_PA1-1.kicad_brd". For example "PATILUNA_PA1-1.kicad_brd"

Activity #2:

Create a PCB layout from the schematic diagram in Practical Activity #1 (Activity #2).

Instructions:

1. Draw Create a PCB layout from the schematic. Use only the bottom layer for a single-sided PCB design. The size of the PCB must be exactly 70 mm x 40 mm. Make sure that power and I/O are properly terminated. Vias are allowed but not more than 20.
2. Run DRC and make sure errors and warning are addressed.
3. Edit the silkscreen and arrange the component designators (name). You may add documentation on the silkscreen to indicate the inputs A, B the power and I/O terminators. On a blank portion within the bottom layer, add the following (without the quotation marks). Replace 'x' with the iteration number of your design.

"CpE 2303L-2024-002 Rev 1.x"

Adjust the font size if necessary.

4. Save the the PCB layout with the same filename format as of the schematic diagram (except for the file extension): "<YOUR LAST NAME>_PA1-2.kicad_brd". For example "PATILUNA_PA1-2.kicad_brd"

Activity #3:

Create a PCB layout from the schematic diagram in Practical Activity #1 (Activity #3).

Instructions:

1. Draw Create a PCB layout from the schematic. Use only the bottom layer for a single-sided PCB design. The size of the PCB must be exactly 100 mm x 50 mm. Make sure that power and I/O are properly terminated. Only 3-5 vias allowed.
2. Run DRC and make sure errors and warning are addressed.
3. Edit the silkscreen and arrange the component designators (name). You may add documentation on the silkscreen to indicate the inputs A, B the power and I/O terminators. On a blank portion within the bottom layer, add the following (without the quotation marks). Replace 'x' with the iteration number of your design.

"CpE 2303L-2024-003 Rev 1.x"

Adjust the font size if necessary.

4. Save the the PCB layout with the same filename format as of the schematic diagram (except for the file extension): "<YOUR LAST NAME>_PA1-3.kicad_brd". For example "PATILUNA_PA1-3.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.