Lab 07B: PCB CAD Design

Lab Presentation: 03/11/2025 (both sections)
Manufacturing files submission: 03/24/2025 (both sections)

1. **Designing your PCB at home**
   1.1. Download and installation
   1.2 Overview of KiCad's Key Tools

2. **Steps to Design your PCB Using KiCad**
   2.1 Schematic Design
          2.1.1 Add components to the Kicad library
          2.1.2 Adding Components
          2.1.3 Wiring Components
          2.1.4 Adding Power and Ground
          2.1.5 Running the Electrical Rules Check (ERC)
          2.1.6 Annotating and Assigning Footprints
          2.1.7 Saving and Exporting the Netlist
          2.1.8 Moving to the PCB Layout
   2.2 PCB Layout and Traces
          2.2.1 Opening the PCB Editor
          2.2.2 Important PCB Layers
          2.2.3 Placing Components
          2.2.4 Creating Traces
          2.2.5 Finalizing the Board Outline
          2.2.6 3D Viewer
          2.2.7 Generating Gerber Files

The lab has no points or any demo requirements. The only required deliverables are the Gerber files which are necessary for the lab10 - PCB Soldering.

In this lab, we will design a custom PCB that acts as a shield for the Raspberry Pi, allowing it to interface with multiple analog sensors. Building on the prototype you created in Lab 7a, we will take the next step of designing a permanent, reliable printed circuit board (PCB) instead of relying on breadboards or temporary wiring.

1. Designing your PCB at home

Now that you tested both your hardware and software in Lab6-a , we can start to design a PCB based on your protoboard. We will use the KiCad software. KiCad is a powerful, open-source electronic design automation (EDA) tool that is widely used for designing schematics and PCB layouts. In this section, we will guide you through the essential steps to use KiCad, including downloading and installation, and working with its various editors.
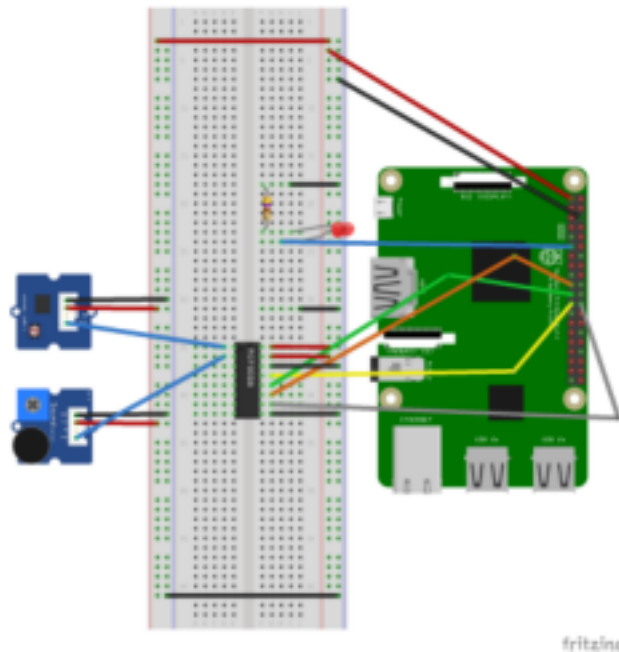

Figure 1: Migrating to the 26-pin RPi shield header

1.1 Download and installation

To begin using KiCad, you need to download and install the software:

1. Go to the official KiCad website: https://kicad.org/download/ .
2. Select the appropriate version for your operating system (Windows, macOS, or Linux).
3. Follow the installation instructions for your platform.
4. Once installed, launch KiCad from your applications or the start menu.

1.2 Overview of KiCad's Key Tools

In the KiCad Project Manager window (Figure 1.2), you will find several tools that are essential for designing a circuit. Below is a brief overview of each tool:

- Schematic Editor: Used to create the circuit design by adding components and logically connecting them.
- Symbol Editor: If a component symbol is missing or needs modification, the Symbol Editor allows you to create, edit, or import schematic symbols.
- PCB Editor: Once the schematic is complete, the PCB Editor is used to design the physical layout of the board, where you place components and route connections.
- Footprint Editor: In the Footprint Editor, you can define or modify the physical footprint of a component, including pad locations and component dimensions. You can also import footprint files that are otherwise not included in the standard KiCad libraries.
- Gerber Viewer: This tool is used to preview Gerber files, which are the manufacturing files needed to produce the PCB.
- Image Converter: You can use this tool to easily add images to the silkscreen layers of your board by uploading and converting image files into footprints for use in the PCB Editor.

Figure 2: KiCad's Key Tools

2. Steps to Design Your PCB Using KiCad

We have prepared a detailed video tutorial that will guide you through the steps of designing your PCB. These videos will be crucial for your success in this lab.

[Video Link](#)

In addition to the video tutorial, we have provided detailed instructions for the general steps you need to follow in this lab. This will help you understand the overall process and the order in which to complete the tasks.

To start, create a new project file. This file will store all the files generated during the design process.

1. Click the icon with the star to create a new project.
2. Select a folder to store your project files.
3. Name your project (e.g., Lab07), and the file will be saved with a .kicad_pro extension, indicating it is a project file.

Once the project is created, you will see the project files in the viewer window, including a schematic file (with the extension .kicad_sch) and a PCB layout file (with the extension .kicad_pcb).

2.1 Schematic Design

Now, we will focus on the schematic file. You can open it by double-clicking the file or by selecting the Schematic Editor.

2.1.1 Add components to the Kicad library

Some parts we need, like the [MCP3008 ADC](#), [Grove Connector](#), and a [26-pin Raspberry Pi header](#), are not in KiCad's default libraries. To add these components, follow these steps:

Step 1: Download Libraries

• Option 1: Download Libraries from SnapEDA

- Visit [SnapEDA](#) and search for your components (e.g., **MCP3008**).
- Download the **symbol, footprint**, and optionally the **3D model**.
- Repeat this process for the **Grove Connector** and the **26-pin Raspberry Pi header**.

• Option 2: Use Provided Materials

- We have also provided these component libraries for you.
- You can download the necessary files (symbols, footprints, and 3D models) from the Google Drive under the Lab 7 folder.

Below is a general description of the different types of files used in KiCad for components:

- .kicad_sym: This is the symbol file, used in schematic designs to represent the component symbolically.
- .step: This is the 3D model file, used to visualize the component in 3D when designing the PCB.
- .kicad_mod: This is the footprint file, which defines the physical layout of the component on the PCB, including pad placements and dimensions.

Step 2: Add Components to KiCad

- Open KiCad and navigate to **Symbol Editor** under **Preferences > Manage Symbol Libraries.**
- Click the folder icon to add the downloaded symbol files to your library.
- Do the same in the **Footprint Editor** for the downloaded footprint files.

Once added, you'll find your components in the library and can use them in your PCB design.

2.1.2 Adding Components:

To populate the schematic, you'll need to add component symbols. Click the **Add Symbol** button (or press A as a shortcut). The symbol and footprint libraries will load, which may take a few seconds. Here's an example of adding basic components to your circuit:

- Search for a resistor by typing R in the filter and selecting a resistor symbol.
- Place the resistor on the canvas. You can modify its value (e.g., change it to 100 kOhms) by double-clicking the symbol and entering the new value in the **Symbol Properties** window.
- Similarly, you can add an LED by searching for it, rotating it if needed (using the R key), and placing it next to the resistor.

2.1.3 Wiring Components:

To connect components, use the **Add Wire** tool (shortcut: W). Click on one component and drag the wire to another component to complete the circuit.

2.1.4 Adding Power and Ground:

You will also need to add power and ground symbols. Search for GND for ground and PWR for power (e.g., 3v3 for 3.3 volts). After placing these symbols, you can wire them to the appropriate points in your circuit.

Power Flags: To pass the Electrical Rules Check (ERC), you'll need to add power flags to the power and ground connections. Search for PWRFLAG, place it on your schematic, and connect it to the power sources.

2.1.5 Running the Electrical Rules Check (ERC)

Once your schematic is complete, it is important to run the ERC to check for any errors or unconnected components.

- Click the **ERC** button to run the check.
- If there are no errors, you're ready to proceed. If there are issues, the tool will point to them on your schematic for easy troubleshooting.

2.1.6 Annotating and Assigning Footprints

Before proceeding to the PCB layout, you need to annotate the schematic and assign footprints:

- Click the **Annotate Schematic** button to label each component with a unique identifier. •

After annotation, click the **Assign Footprints** button to associate each component symbol with its corresponding physical footprint.

2.1.7 Saving and Exporting the Netlist

Once you have assigned footprints, save your work by clicking **File > Save**. Then, generate the **Netlist** by clicking **File > Export > Netlist**. This file contains the electrical connections for your circuit and will be used in the PCB layout.

2.1.8 Moving to the PCB Layout

After completing the schematic and exporting the Netlist, the next step is to move on to the PCB layout. Click the **Switch to PCB Editor** button to start the layout process.

2.2 PCB Layout and Traces

After successfully completing the schematic design and assigning footprints to each component, the next step is to lay out the components on the PCB and create the traces that will connect them.

In KiCad, the PCB layout tool allows you to:

- Place the footprints of your components onto the board.
- Define the physical layout and routing of your circuit using conductive traces.
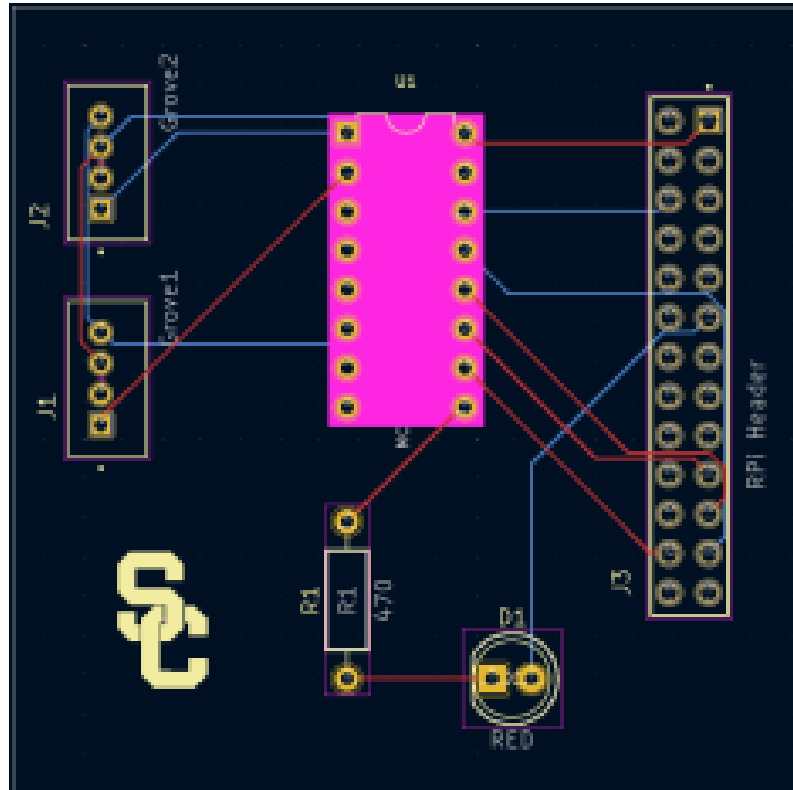- Set up design rules to meet manufacturing constraints.

Figure 3: PCB Layout and Traces

2.2.1 Opening the PCB Editor

To begin, open the PCB Editor in KiCad. You can do this by either:

- Double-clicking the .kicad_pcb file in the project manager.
- Clicking on the **PCB Editor** button in the project window.

The PCB Editor will open with a blue background and a grid, which helps align and measure the distances between components and traces. On the right side, you will see the layers tab, where different PCB layers (like copper and silk screen) can be selected.

2.2.2 Important PCB Layers

For this lab, you will primarily work with four layers:

- **F.Cu:** The front copper layer, where traces and components on the top side of the board are placed.
- **B.Cu:** The back copper layer, where traces and components on the bottom side are placed.
- **F.SilkS**: The front silkscreen layer, used for component labels and designators.
- **Edge.Cuts:** The board outline layer, used to define the physical dimensions of the PCB.

Keep in mind that the reason why we have F.Cu and B.Cu layers is so we can design a two-layer PCB which allows a smaller and optimal PCB design. Additionally, when designing, make sure the copper layers do not overlap, as this can cause issues in the design.

2.2.3 Placing Components

To place the components, follow these steps:

- Select the footprints from the imported netlist.
- Use the M key to move a component around the grid and the R key to rotate it.
- Align components logically and minimize the distance between connected components for efficient routing.

2.2.4 Creating Traces

Once components are placed, you will need to route the electrical connections between them:

- Select the **Route Tracks** tool from the toolbar.
- Click on a pin to start a trace, and move the cursor to another pin to complete the connection.
- You can route traces on either the **F.Cu** or **B.Cu** layers, depending on the circuit's complexity.

Ensure your traces are properly sized according to your design rules. Use the **Design Rules Checker (DRC)** to identify any errors in your layout.

2.2.5 Finalizing the Board Outline

Before completing the design, you need to define the board's dimensions:

- Select the **Edge.Cuts** layer.
- Use the **Draw Rectangle** tool to outline the board, ensuring it fits the required dimensions (e.g., 2in x 2in for this lab).

2.2.6 3D Viewer

Once your layout is complete, you can visualize your design using KiCad's 3D Viewer:

- Click the **3D Viewer** button to preview the board and its components in 3D.
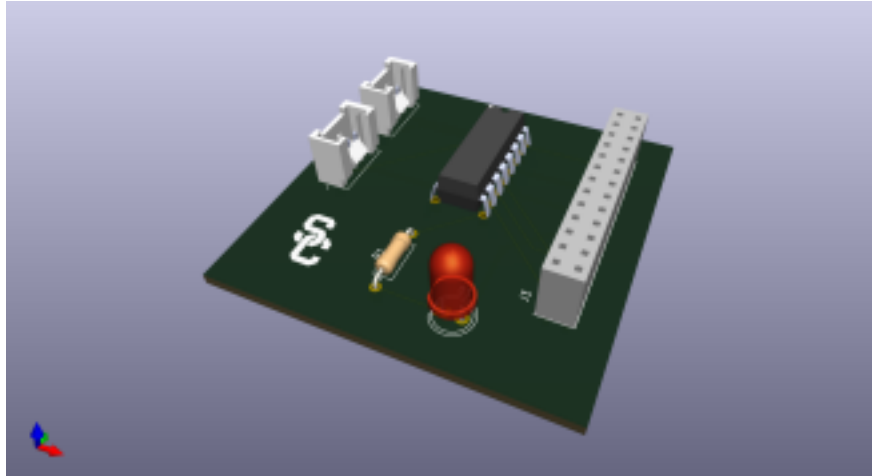- If you have downloaded 3D models for your components, you can view them in the 3D layout.

Figure 4: 3D Rendering View of the PCB Design

2.2.7 Generating Gerber Files

The final step in your design process is generating the manufacturing files:
- Go to **File > Plot** and select the Gerber format.
- Make sure to generate files for all necessary layers (F.Cu, B.Cu, F.SilkS, Edge.Cuts, etc.).
- Click **Generate Drill Files** to create the necessary drilling information.

These Gerber files can then be submitted to the PCB manufacturing service of your choice. To submit the files for manufacturing, you'll need to compress the Gerber and drill files into a zip folder:

1. Go to your project folder and select all the Gerber (*.gbr) files and drill (*.drl) files.
2. Right-click and select Send to > Compressed (zipped) folder.
3. Name the folder something identifiable, like lab07.zip.

Before submitting your zip file, it's a good idea to double-check your files using the Gerber viewer:

1. Open KiCad's **Gerber Viewer**.
2. Select File > Open Zip Archive, and choose your lab07.zip file.
3. Ensure all layers are correct by toggling the visibility of each one.

Finally, this portion of the lab has no points or any demo requirements. The only deliverables are the Gerber files which are necessary for the lab10 - PCB Soldering.