ARTICLE

Project X Session 3 - PCB Design Using KiCAD

Junzhe Chen, Oli Sharratt, and Yuxuan Han

Introduction
In the third Project X session, you will be introduced how to design your first printed circuit board (PCB) using KiCAD.

Contents

1.   General PCB Design Workflow Introduction

1.1   Overview

When designing a PCB, the process typically involves the following steps:

- **Create a schematic**: Draw the circuit diagram showing the components and their connections.
- **Assign footprints**: Map each component to its physical interface on the PCB.
- **Design the PCB layout**: Arrange footprints, draw traces, and prepare the board for fabrication.
- **Fabricate the PCB**: Manufacture the PCB based on the design file.
- **Populate and solder components**: Place and solder components onto the fabricated PCB.
- **Test the PCB**: Verify the board's functionality and ensure it works as intended.

1.2   Detailed Explanation

The first step is creating a schematic. This is a circuit diagram that represents the electronic components using standardised symbols. An example of a PCB schematic looks like Figure 1.
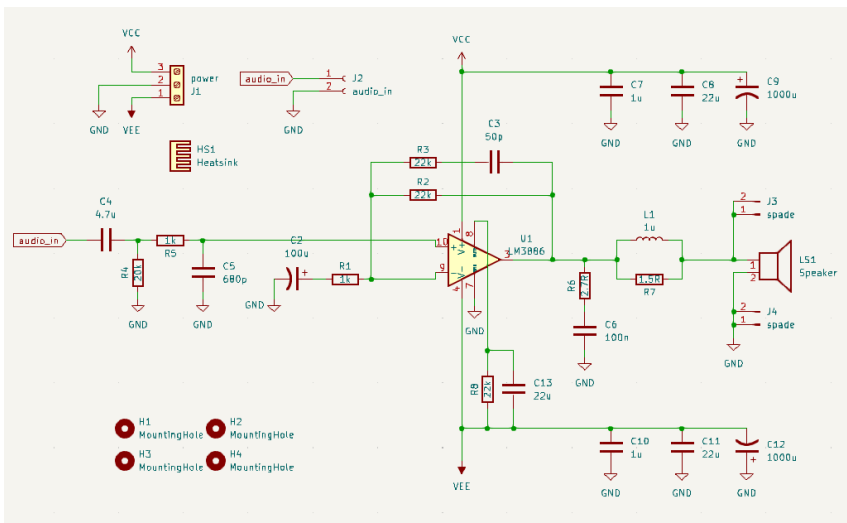

Figure 1. Example of a PCB Schematic

After completing the schematic, the next step is assigning footprints to the components. A footprint represents the physical dimensions of the component, along with the locations and dimensions of its connectors. This is used in the PCB layout so it can be position correctly without interfering with other components on the PCB.

Once the schematic is complete and the footprints are assigned, the next step is to design the PCB layout. This involves drawing the physical board, arranging the footprints, and adding traces to connect the components electrically. In the PCB design file shown in Figure 2a, the footprints correspond to the schematic symbols, and the traces connect them. Figure 2b illustrates a 3D rendered image of the PCB design.
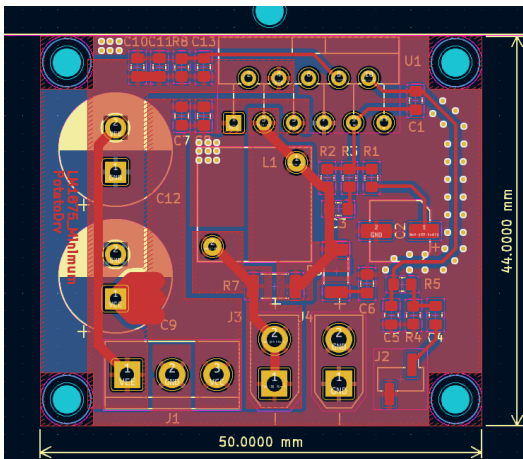
(a) PCB Layout Example
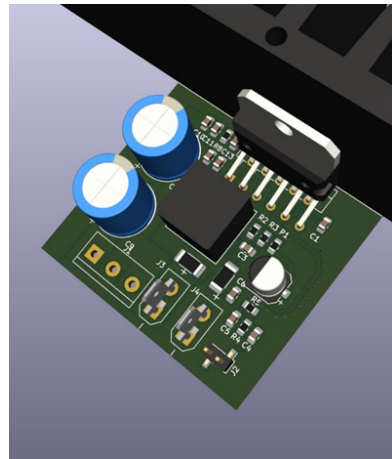
(b) 3D Rendered PCB

Figure 2. PCB Layout

For a typical two-layer PCB design in KiCAD, the top layer is shown in red, while the bottom layer is in blue. A significant portion of the design is covered with copper, which connects to the ground (GND) of the circuit. This approach minimizes return path impedance, a common practice in PCB design.

For Project X, after completing the PCB design, it will be sent to the UCL PCB lab for fabrication. Producing a double-layer PCB usually takes one day. Once the PCB is fabricated, the components are populated and soldered onto the board. Finally, the testing phase begins to ensure the PCB functions as intended.

2.   KiCAD Template & Tasks

To begin with, we have drawn part of the important parts of the mood card project: the microcontroller and the sensor. These are placed in the schematics and the layout. The edges of the board have also been added to constrain the dimensions as we will panellise (put in to a grid) all boards in to sheets for fabrication. Your tasks will be the following:

- Connect the traces between the microcontroller and the sensor.
- Design and layout your own LED circuits.
- *[Advanced]* Try to make the footprints of the microcontroller and sensor.

3.   Start With KiCAD

3.1   Open KiCAD Project

Firstly, open the KiCAD project file via **File > Open Project.** The KiCAD project file contains both the schematic, and the PCB layout files inside the file directory and it will be displayed in the **Project Files** section.

3.2   Schematics and PCB File

The screenshot in Figure 3 displays what will be seen when opening the project file. The most important files are the schematics and the PCB layout files, which ends with '_sch' and '_pcb' respectively.
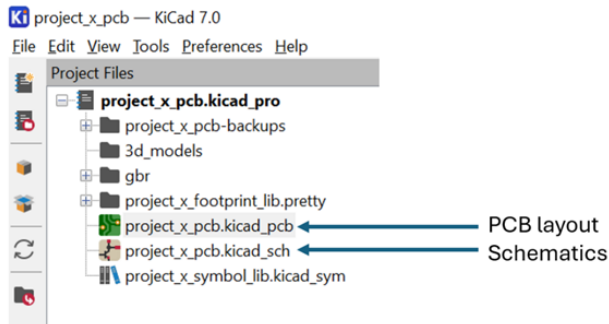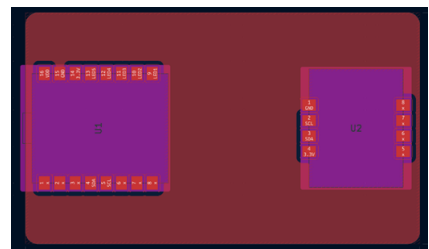

Figure 3. PCB Layout and PCB Schematic Files in Project View

Try to open the schematics and PCB files by double clicking each item – you should be able to see the following shown in Figure 4. Where Figure 4a shows the schematic template, and Figure 4b shows the PCB in the design template. The microcontroller (U1) and the sensor (U2) has been placed. However, the traces have not been placed yet.
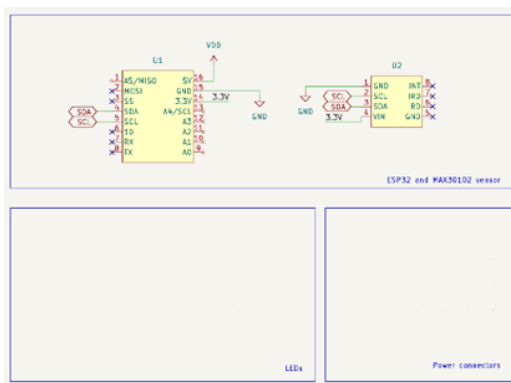
(a) PCB Schematic Template   **(b)** PCB Layout Template

Figure 4. The templates you will use

You will need to route the traces between the microcontroller and the sensor, design your LED circuits in schematics and lay it out onto the PCB, as well as adding a battery connector to the PCB to supply power to the microcontroller and the sensor.

4.   Schematic Design

4.1   Toolbar

On the right–hand side of the schematic editor, there is a tool bar. You will use the toolbar to edit, add the components and wiring the connections most of the time. The commonly used tools with its shortcut keys are shown in the screenshot in Figure 5.
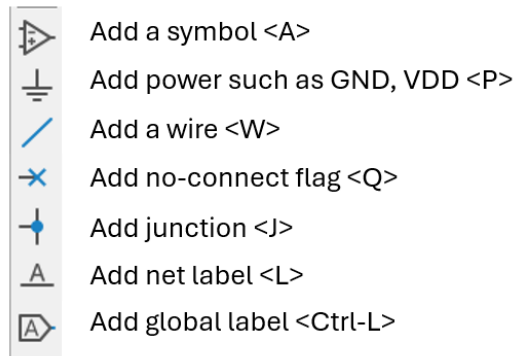

Add a symbol <A>

Add power such as GND, VDD <P>

Add a wire <W>

Add no-connect flag <Q>

Add junction <J>

Add net label <L>

Add global label <Ctrl-L>

Figure 5. Commonly used tools in the toolbar

4.2   Add Components

To add a component, click the **Add a symbol** icon or press <A> in the keyboard. The following window in Figure 6 will be opened. The components can be found by using the search bar on the top. In this example, we are searching a LED in the symbol library and the symbol is shown on the top right. After selecting the symbol you want, press OK and then the symbol can be placed on the schematic.
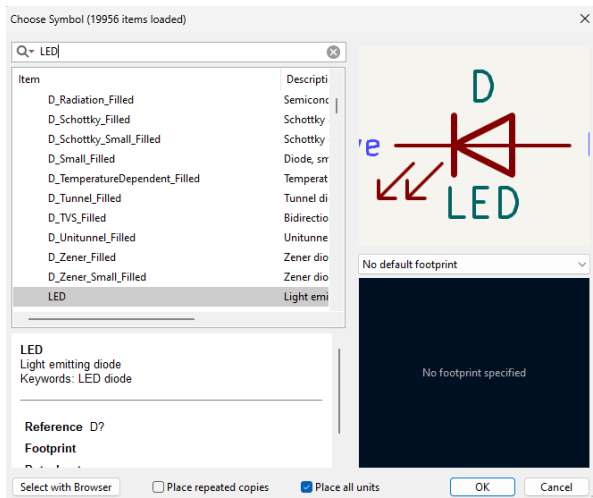

Figure 6. Add an LED to the schematic template.

<u>**Task**: Place one LED and one resistor in the schematic's LED region.</u>

4.3   Connecting Components with Wires

After placing down the components, we can start connecting them. To connect components together, we can move the cursor close to the unconnected pin (marked with small circle), and a wire will be generated for us to connect. A better method is to use the **Add a wire** button in the toolbar (or use the shortcut <W>) to connect the symbol together.
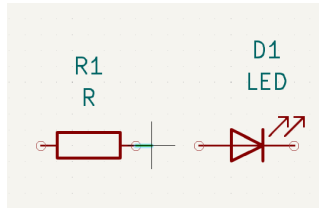

Figure 7. Connect components with wire.

In Figure 7, may notice that the value of resistor is R but not the exact value we want (*220*). To change this, double click on the R, then the value of the resistor can be modified in the following window shown in Figure 8:
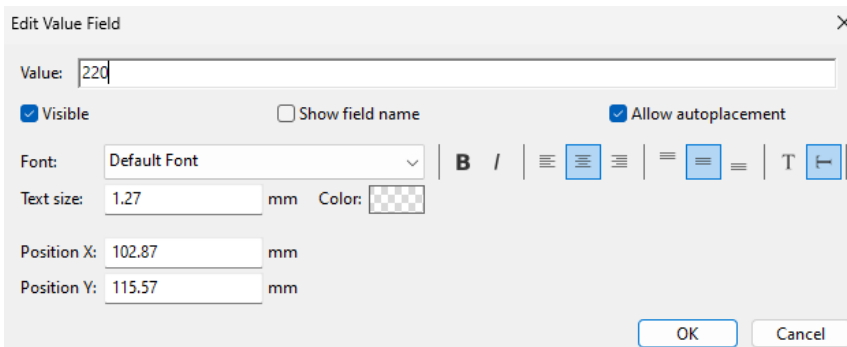

Figure 8. Edit resistor value.

4.4   Good Practice: Use "Nets" to Connect Components

4.4.1   A Negative Example

As the number of wires in a schematic increases, the design can become difficult to read and manage. A negative example is shown in Figure 9, where only wire connections are used instead of implementing net connections.

Using net connections can greatly simplify the organization of a schematic. Nets not only reduce the mess of wires but also improve readability by assigning names to connections. These net names are carried into the PCB layout stage, which can save time during the Design Rule Check (DRC).

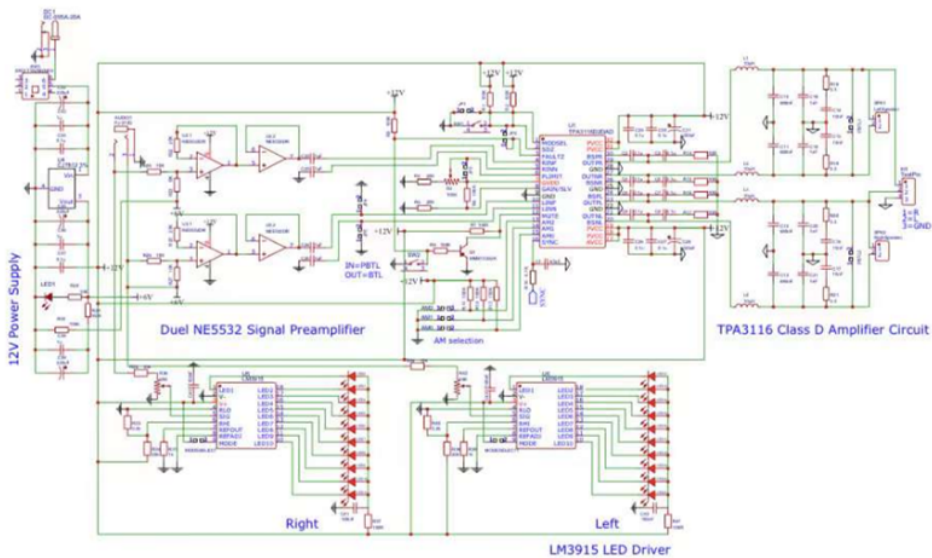For example, knowing the name of a problematic net makes it easier to identify and fix connection errors.

Figure 9. Negative example of not using net connections.

Note from editor: Don't be afraid, the schematic you are going to design will be much simpler than this.

4.4.2   Two Net Types in KiCAD

There are two main types of nets in KiCAD:

1. **Net Label (shortcut: <L>):**
   A net label assigns a name to a specific net, effectively "labelling" it. If two nets share the same label, they are automatically connected. This can be verified using the highlight tool (Figure 10).
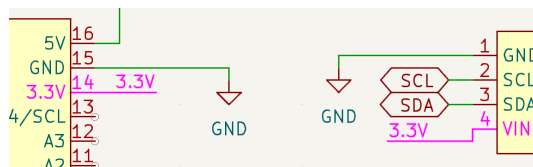
Figure 10. Net Labels

Disadvantages: While net labels are useful for naming connections, they do not indicate the characteristics of the net, such as whether it is an input or output. Additionally, net labels work only within a single schematic sheet and are not carried over to other sheets in multi–sheet designs.

2. **Global Label (shortcut: <Ctrl-L>):**
   A global label not only connects nets with the same label but also specifies their characteristics,

such as input, output, or power. (See Figure. 11)

Advantages Compared to Net Label: Unlike net labels, global labels are accessible across different schematic sheets, making them suitable for multi-sheet designs.
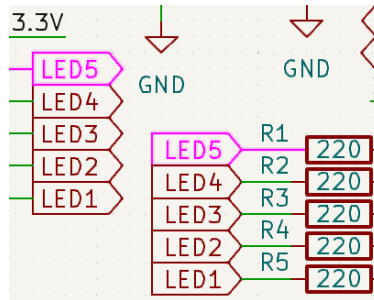

Figure 11. Global Labels

Task: Please try to connect the LEDs to the microcontroller using global labels.

4.5   Assign Footprints

At this stage, the schematic design should be complete. This means that all component symbols have been correctly connected using net labels and verified by the volunteers. The next step is to assign the corresponding footprints to the symbols.

To assign footprints, use the top menu bar and navigate to **Tools > Assign Footprints...** (See Figure 12a) or click the following icon on the top toolbar (See Figure 12b).

Once the footprint assignment tool is opened, the wizard will appear as shown in Figure 13. The footprint assignment tool is divided into three columns:

- **Left column**: Displays the footprint library, which contains all available footprints.
- **Middle column**: Lists the symbols in the schematic. Symbols without assigned footprints are highlighted in yellow for easy identification.
- **Right column**: Shows the filtered footprints, based on the criteria entered in the footprint filter.

To find a specific footprint:

1. Use the **Footprint Filters** in the top bar of the wizard.
2. Type the name of the desired footprint into the search bar.
3. The filtered footprints will appear in the right column.

Figure 14 shows the correct footprints for the LED after applying the footprint filter with correct keywords. Once the correct footprint is identified, double-click it to assign it to the corresponding symbol. Repeat this process for all symbols in the schematic. (Remember that different components have different footprints!)
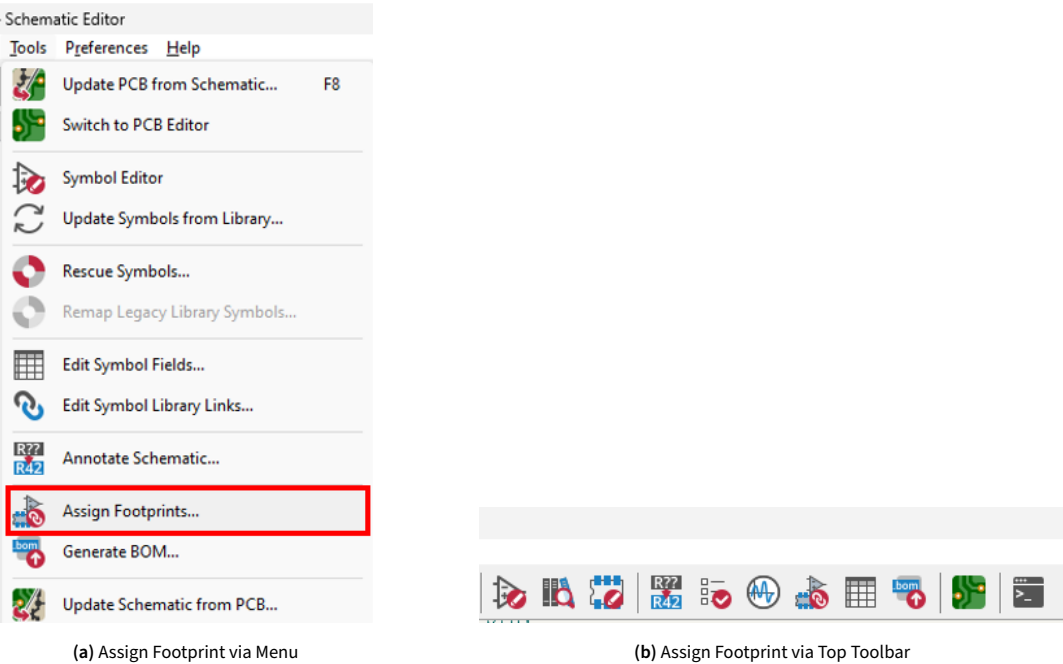
(a) Assign Footprint via Menu

(b) Assign Footprint via Top Toolbar

Figure 12. Two ways to enter the footprint assignment wizard.
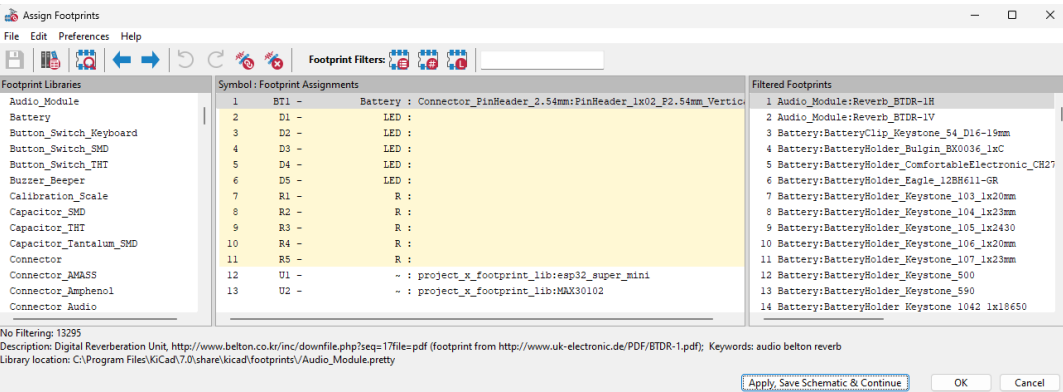
Figure 13. Footprint Assignment Wizard

Table 1. Footprints for your first PCB.

Component Types	Search keyword	Footprints
Resistors	R 0805	Resistor_SMD:R_0805_2012Metric_Pad1.20x1.40mm_HandSolder
LEDs	LED 0805	LED_SMD:LED_0805_2012Metric_Pad1.15x1.40mm_HandSolder
Batteries	header 1x02 2.54 vertical	Connector_PinHeader_2.54mm:PinHeader_1x02_P2.54mm_Vertical

The footprints that you will use are listed in Table 1.

Figure 14. Correct footprints after filtering

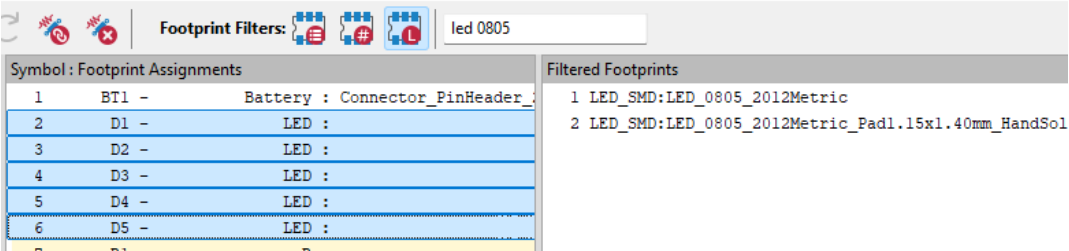**Task**: Assign the footprints to the symbol. After that, ask one volunteer to check.

4.6   Final Check: Schematic Design

Please ask one of the volunteers to check if your schematics is correct. If so, you could move onto the PCB design in the next section.

5.   PCB Design

Congratulations on completing your schematic design and having it reviewed by the volunteers! Now, it's time to move on to the PCB design phase.

5.1   Importing and Placing Components

When you want to start doing PCB layout:

1. First, open the '_pcb' file.
2. Second, navigate to **Tools > Update PCB** from schematics on the menu bar, or press <F8> on your keyboard.

A window like the one shown in Figure 15 will pop up. If there are no errors or warnings, click Update PCB. Once updated, the component footprints from your schematic will appear at your mouse cursor, as shown in the Figure 16 below:

In the PCB layout, you will notice thin blue lines connecting different pins or components—these are called *ratlines*. Ratlines represent the physical connections between the pins based on the schematic.

When choosing where to place the components, it is best to look at the ratlines so that the ratline has the least crossing. As ratline indicates the physical connections between the pins, making them to have least crossing will make the trace routing much easier.
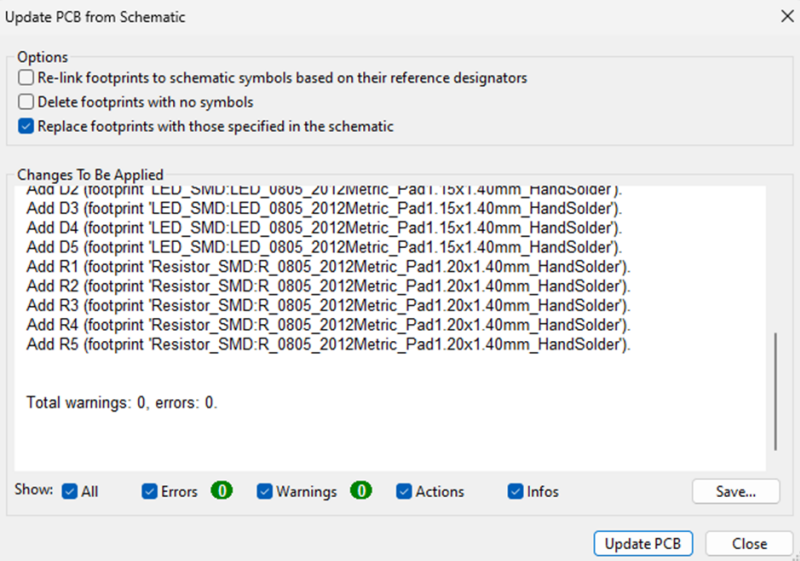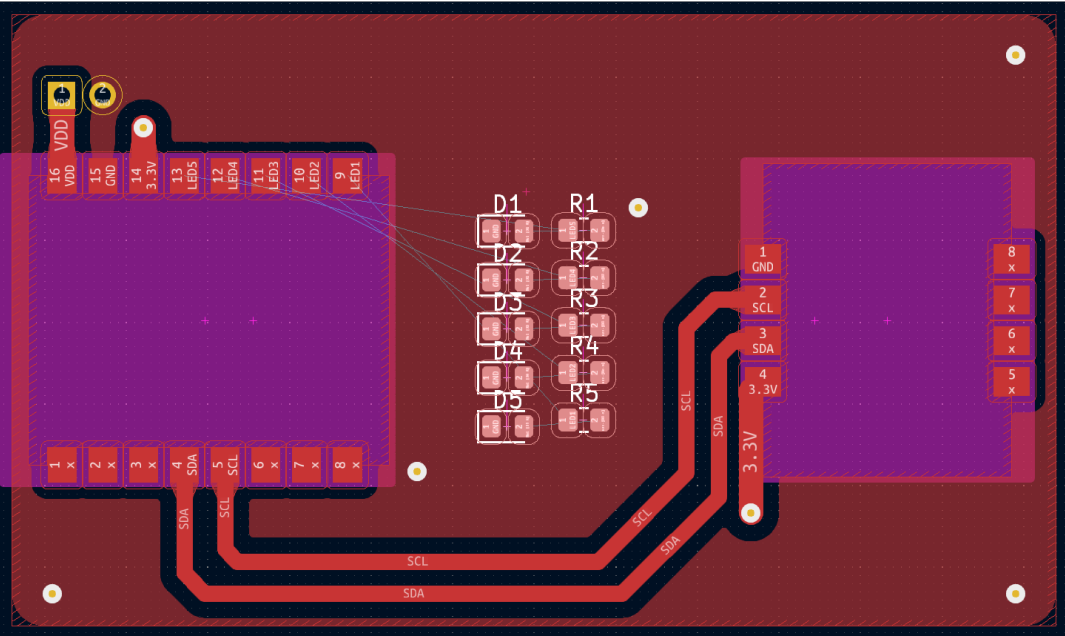
Figure 15. Update PCB from schematic

Figure 16. An example of the PCB board after updated from schematic.

5.2   Setting Up Design Rules (IMPORTANT)

5.2.1   Open Board Setup

After placing the components to the appropriate position, the routing can begin. But before start routing, it is important to set appropriate trace width, clearance and other parameters so that the board can be manufactured from the UCL lab. To do that, click *board setup*, which is on the top left corner of the PCB editor (See Figure 17 below).
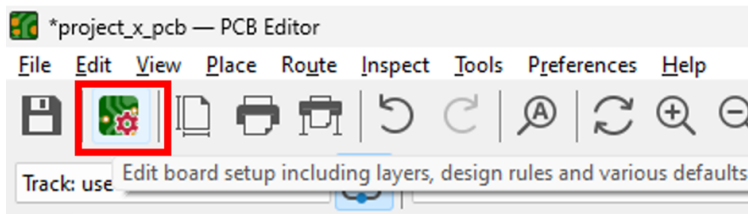
Figure 17. Find and click the "board setup" icon

5.2.2   Set Up Constrains

After opening the board setup, go to the *constraints* page and fill in the following parameters. The Copper section is the most important section for reaching the UCL lab manufacture. Please fill in the constrains exactly the same as those shown in Figure 18 below, so that the requirements can be met for PCB manufacture.
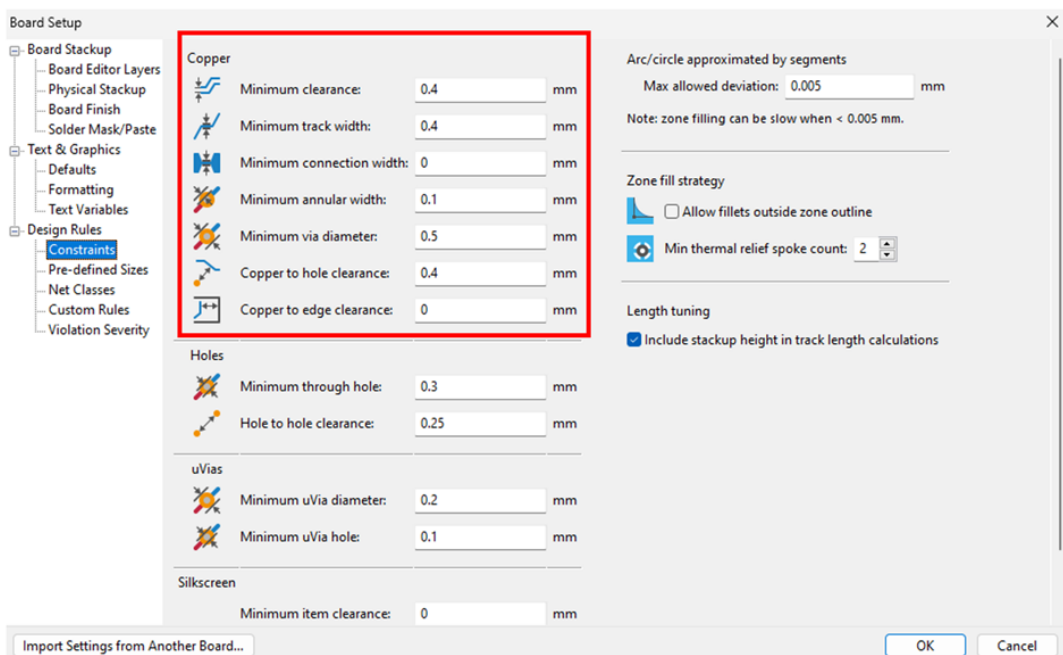

Figure 18. Design Constraints.

5.2.3   Set Up Pre-defined Sizes

The *Pre-defined Sizes* settings define the properties of tracks, vias, etc. that you will use for routing. Firstly, we need to setup the track width. Go to Pre-defined Sizes (which is just under Constraints, and the Track property should be shown on the left column (see Figure 19). As indicated by the constraints, the track cannot be thinner than 0.4 mm. So, a track width larger than 0.4 mm should be added. You can edit multiple pre-defined sizes for the tracks, depending on the type of track you are routing. For example, when routing power tracks, like VDD and 3V3, since they draw more current than signal tracks, they are usually thicker.
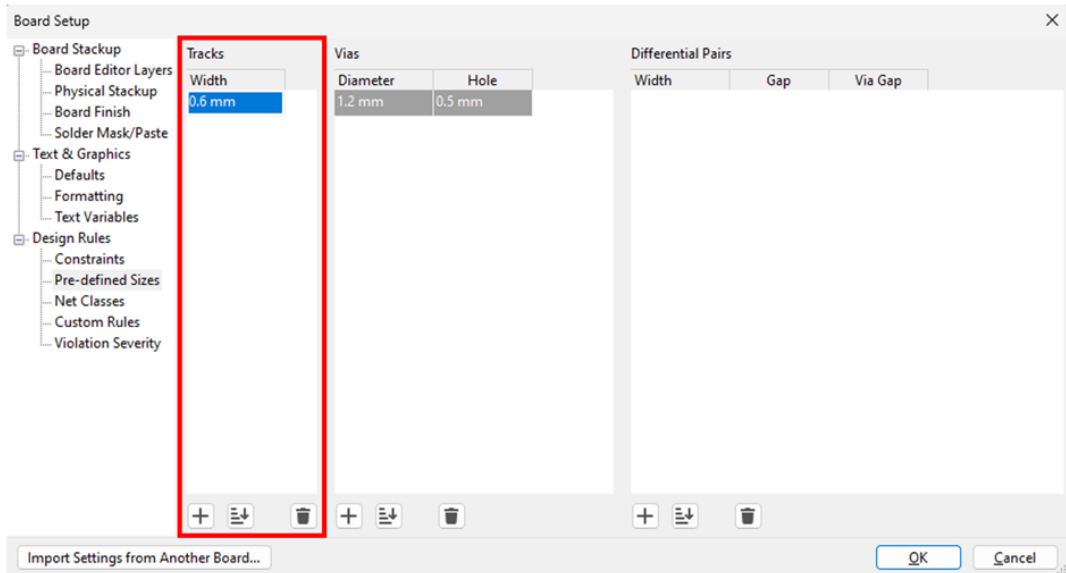

Figure 19. Pre-defined Sizes

For vias (in the next column), UCL uses specific size vias. The via size should be set as same as the given example in Figure 19 (Diameter = 1.2 mm, Hole = 0.5 mm).

Task: Set the constraints and pre-defined sizes and ask one of the volunteers to check.

5.3   Track Routing

After setting the board setup, it is the time to route the tracks.

To route the track in KiCAD, firstly, place the cursor on the pad we want to route, and then press the routing key <X> to enter the routing mode. The pads that need to be connected will be highlighted. (See Figure 20)

A copper pour is already added for this board, which connects all the ground connection, so you do not need to connect the GNDs. Remember to update the copper pour when you finished routing the tracks. This can be achieved by pressing <B>.
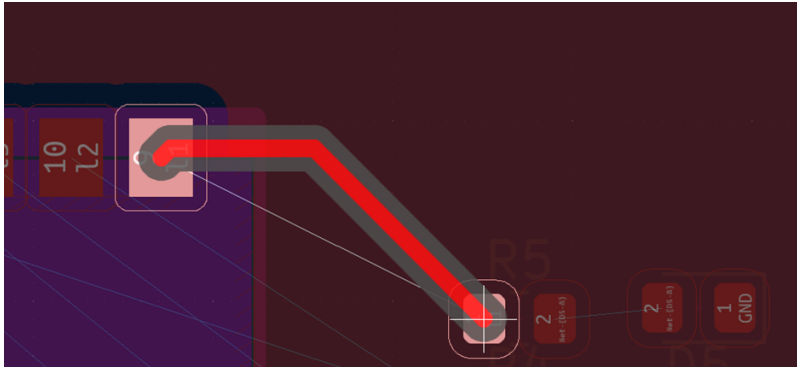
Figure 20. Route a track

5.4 Using Vias to Switch Layers

Sometimes, it is difficult to route certain tracks that goes across other tracks. In this case, it is good to switch the layer to "bridge" the track over (or under) it.

To connect the top and bottom together, a *via* is required. Via is a hole that is made right between the bridging tracks, and it connects the top and bottom together by plating the copper across the hole. When placing the track to the desired position, press <V>, then a via will be generated on the cursor. Move the via to the desired place, then click the left mouse button, the via will then be placed. (See Figure 21a)
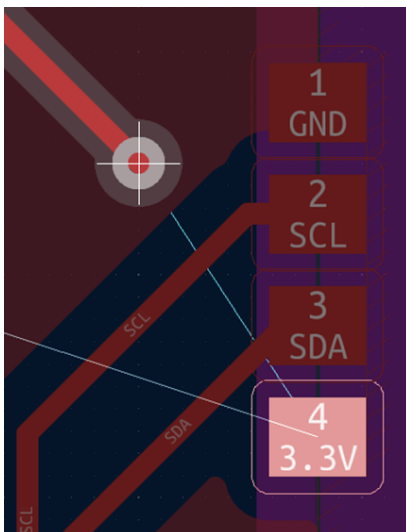
(a) Placing a via.
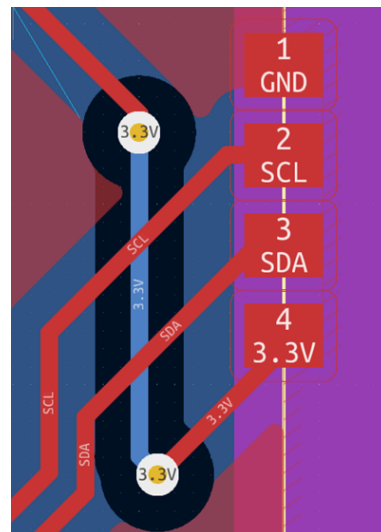
(b) A trace with vias.

Figure 21. Using vias during routing.

After placing the via, the KiCAD will automatically switch the track to the bottom layer, where the background is showing in blue. Then route the bottom track across the "bridge", press <V> again,

then the track will be completed as shown in Figure 21b.

6. Design Rule Check

After the routing has been completed, it is necessary to perform design rule check (DRC) before sending the design file to be manufactured. The design rule check can be found in Inspect/Design Rule Checker in the menu bar shown in Figure. 22a below:

(a) Design Rules Checker in the menu.   (b) An error in the DRC report.
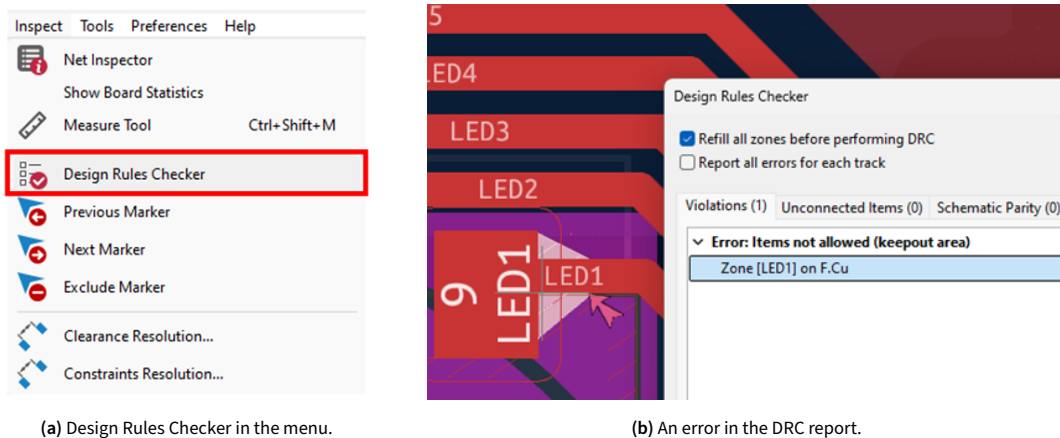
Figure 22. Performing design rule check.

If the DRC reports an error (See Figure. 22b for example), a quick way to locate the error is by double-click the error. Usually, the error will be highlighted as shown above in the right screenshot. It is important to keep the design error-free, so that it maximizes the chance of getting a working board after the board have been fabricated out.

7. Congratulations

Congratulations to make it till this far! The PCB should be ready and please ask one of the volunteers to check if your board is correct and ready for fabrication!

8. Appendix I: ESP32C3 Super Mini

(a) Pinout of the ESP32C3 Super Mini Board

(b) Dimensions of the ESP32C3 Super Mini Board

Figure 23. ESP32C3 Super Mini Board pinout and dimensions.

9. Appendix II: MAX30102

(a) Pinout of MAX30102

(b) Dimensions of MAX30102

Figure 24. MAX30102 pinout and dimensions.

10.   Appendix III: KiCAD Shortcuts

Schematic Editor

Shortcut	Action
M	Move a component
R	Rotate a component
X	Flip a component
E	Edit a component
G	Drag a component
Del	Delete a component
W	Place a wire
P	Place a power symbol
L	Place a label
CTRL + L	Place a global label
F1/F2	Zoom in/out

PCB Editor

Shortcut	Action
M	Move an item
D	Drag an item
R	Rotate an item
E	Edit an item
Del	Delete an item
X	Place a track
V	Add a via
B	Rebuild copper zones
H	Cycle layer display options
Ctrl + F	Find a component
Alt + 3	3D viewer

General Shortcuts

Shortcut	Action
Left click/drag	Select item(s)
Right click drag	Navigate canvas
Middle click drag	Navigate canvas
Scroll in/out	Zoom in/out
Ctrl + S	Save the document
Ctrl + Z	Undo
Ctrl + Y	Redo
Ctrl + C	Copy
Ctrl + V	Paste
Ctrl + X	Cut