Laboratory 6: Digital Piano - Introduction to Soldering and PCB Design

Introduction and Objective: In this laboratory, you will learn the principles, processes, and tips for designing a Printed circuit board (PCB). Two PCB of the circuits, the Inferred-sensitive light and the digital piano circuits, will be designed and soldering in the lab. You should acquire the basic skill of PCB design by practice.

1 Introduction

PCBs are essential in modern electronics for connecting components, from calculators to smartphones. They ensure circuit functionality, durability, and compactness. Designing and soldering PCBs is crucial for translating theoretical designs into practical electronics, a key skill for engineers. This laboratory covers PCB design, from concept to assembly, including schematic creation, component placement, routing and fabrication ordering. By the end, you will be able to create professional-quality PCBs, whether simple prototypes or complex multi-layer boards.

2 What is a PCB?

A PCB is a board made of insulating material, like fiberglass or composite epoxy, with conductive pathways etched or printed on it. These pathways connect electronic components like resistors, capacitors, ICs, and connectors, forming a complete circuit. There could be single, double and multi-layer of conductive pathways for simple to complex circuits. PCBs are essential in modern electronic devices, offering reliable mechanical support and electrical connections. Figure 1a and b shows the front and back sides of a two-layer PCB. The following are to Key Components of a PCB as shown in Figure 1:

1. **Substrate (Base Material)**: The base layer of the PCB, typically fiberglass (FR4), ceramic, or flexible polyimide, providing support and insulation between conductive layers.

2. **Copper Layers**: Copper foil layers on the substrate, etched to create traces for electrical signals between components.

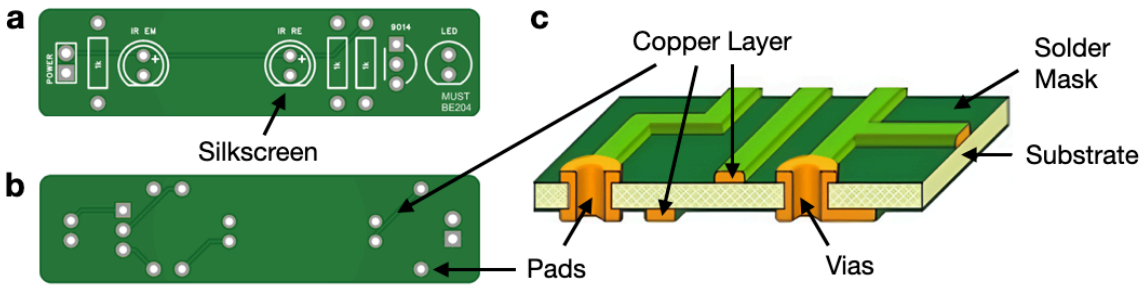


Figure 1: **a** and **b** the front and back sides of a two-layer PCB. **c** the layer structure of the PCB.

3. **Solder Mask**: A protective layer over copper traces to avoid oxidation and shorts, usually green, but available in other colors.
4. **Silkscreen**: Printed text and symbols for component labeling and test points, aiding assembly and troubleshooting.
5. **Pads**: Exposed copper areas for soldering components.
6. **Vias**: Plated holes connecting traces between Copper layers.

3 PCB Design Process

The design of a PCB begins with a clear understanding of the requirements and functionality of the circuit. Here is the key steps involved:

1. **Schematic Design**: The first step is to create a schematic, which is a graphical representation of the electrical connections of the circuit. **Choose components** that meet the electrical and mechanical requirements of the circuit. Consider factors such as size, power rating, and availability when selecting components. Use PCB design software (e.g., JLCPCB, KiCad, Altium Designer) to **define the components and their interconnections**. Ensure that the schematic is accurate and includes all necessary components, such as resistors, capacitors, integrated circuits (ICs), and connectors.
2. **PCB Layout Design**: Translate the schematic into a physical layout using the PCB design software. **Place components** on the board in a logical and efficient manner, minimizing the length of electrical connections. **Route the traces** (copper pathways) to connect the components according to the schematic. Pay attention to design rules, such as trace width, spacing, and via placement, to ensure manufacturability and reliability.
3. **Design Verification**: Use design rule checks (DRC) and electrical rule checks (ERC) to identify and correct errors in the layout. Simulate the circuit if possible to verify its functionality before manufacturing.

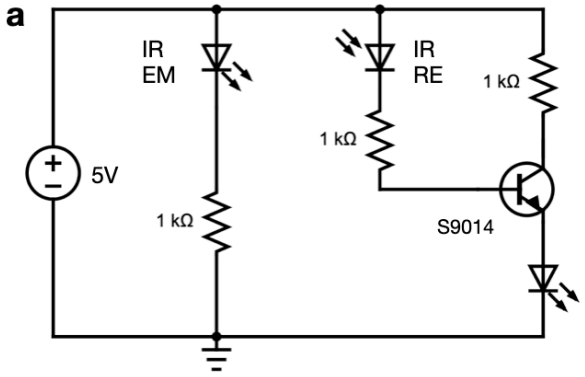
4 PCB Fabrication and Soldering

Once the design is finalized, the next step is to fabricate the PCB. Professional PCB manufacturing is the most common approach for complex designs. Submit the design files to a PCB manufacturer. Choose the appropriate specifications, such as board material, layer count, and surface finish. The manufacturer will produce the board using processes such as etching, drilling, and plating.

Soldering is the process of attaching components to the PCB to create electrical connections. The soldering process varies depending on the type of components and the complexity of the board. Similarly to the pervious laboratory, insert the component leads through the holes in the PCB and bend the leads slightly to hold the component in place. Apply heat to the pad and lead using a soldering iron, then feed the solder into the joint. Finally, trim the excess leads after soldering. Inspect solder joints for defects like bridges, cold joints, or insufficient solder, and use a multimeter or continuity tester to check electrical connections.

For professional results, you should plan with a clear schematic and PCB layout, test design, and function regularly to identify early errors, and document all design, fabrication, and assembly details for future use.

a



b

Component	Model No.
Connector	XFCN PZ254V-11-02P
IR Emitter	Everlight Elec IR333C
IR Receiver	XINGLIGHT XL-504PDC
1k Resistor	TE Connectivity YR1B1K0CC
Transistor	Jiangsu S9014-TA
LED	XINGLIGHT XL-502UGD

Figure 2: **a**, Circuit diagram of the Inferred switch light. **b**, Table of the reference model no. of the components.

5 PCB Design Software

PCB design software allows users to design, simulate, and optimize PCB prior to manufacturing. These tools typically include schematic capture, component placement, routing, and design rule checking (DRC) to ensure that technical and manufacturing requirements are met. Open-source and Web-based tools increase accessibility for beginners while maintaining advanced features for professionals. EasyEDA https://easyeda.com, a free web-based tool, is popular for its ease of use and integration with JLCPCB. It caters to both beginners and professionals, offering schematic capture, PCB layout, and a vast component library. EasyEDA facilitates direct PCB ordering from JLCPCB https://jlcpcb.com, easing the design-to-production process, and supports real-time collaboration. Its accessibility, cost-effectiveness, and manufacturing integration make it a favorite for hobbyists, students, and professionals. We will use EasyEDA in the laboratory. List 2 more PCB design software and the advantages in the answer sheet.

5.1 Table for PCB design software and the advantages.

6 Inferred Switch Light

Figure 2a shows the circuit diagram of the inferred switch light that has been soldered in the last laboratory. The list of the electronic components used is shown in Fig. 2b. Follow the steps state in Sec. 3 to design the PCB using the EasyEDA. You should refer to the video here https://youtu.be/1E2ULNZT76M. After the design, screenshot the schematic, the front and back sides of the PCB design and the 2D preview, and the 3D preview of the frond sides.

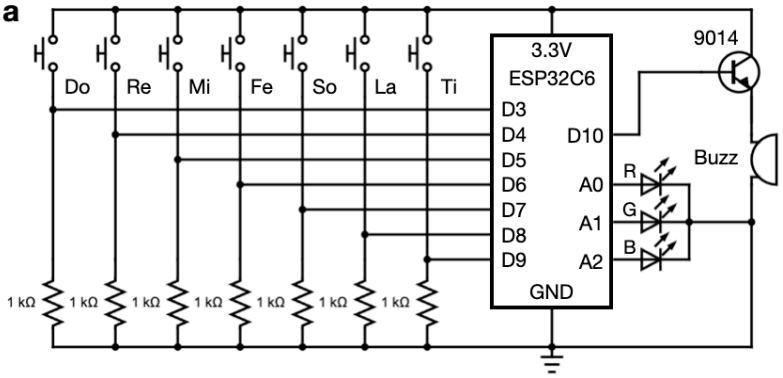
6.1 Screenshots of the schematic, the front and back sides of the PCB design and the 2D preview, and the 3D preview of the front sides.

6.2 Save the project of your PCB design and upload it to Moodle with the file name "lab6_v62".

A PCB of the light has been fabricated. Solder the electronic component to the PCB. Take photos of the front and back sides of the soldered PCB. Demonstrate the function of the PCB and take video of it. Please take photos and fill in the answer sheet ever-though the soldering is not finished or the circuit is not function.

6.3 Photos of your soldered PCB of inferred switch light.

a



b

Component	Model No.
1x7p Connector	PM254V-11-07-H85
RGB LED	TJ-L5FPTXHMCSLCRGB-A5
Button	TC-1103-B-A
1k Resistor	TE Connectivity YR1B1K0CC
Transistor	Jiangsu S9014-TA
Buzzer	FS9055-05

Figure 3: **a**, Circuit diagram of the Digital Piano. **b**, Table of the reference model no. of the components.

6.4 Take a video to demonstrate the inferred switch light and upload it to Moodle with the file name "lab6_v64"

7 Digital Piano

Figure 3a shows the circuit diagram of a digital piano using ESP32C6, RGB LED and Buzzer. The list of the electronic components used is shown in Fig. 3b. Follow the steps state in Sec. 3 to design the PCB using the EasyEDA. After the design, screenshot the schematic, the front and back sides of the PCB design and the 2D preview, and the 3D preview of the frond sides.

7.1 Screenshots of the schematic, the front and back sides of the PCB design and the 2D preview, and the 3D preview of the frond sides.

7.2 Save the project of your PCB design and upload it to Moodle with the file name "lab6_v72".

Also, A PCB of the piano has been fabricated. Solder the electronic component to the PCB. Take photos of the front and back sides of the soldered PCB. After finishing the soldering, check the connection of the board, install the XIAO development board and upload the program "Button_LED_Flowing" downloaded from Moodle. Demonstrate the function of the PCB and take video of it. Please take photos and fill in the answer sheet ever-though the soldering is not finished or the circuit is not function.

7.3 Photos of your soldered PCB of digital piano.

7.4 Take a video to demonstrate the digital piano and upload it to Moodle with the file name "lab6_v74"

8 Conclusion

In this laboratory, you gained hands-on experience in PCB design by assembling the infrared switch light and digital piano. Your understanding of PCB is enhanced and the experiment provides a solid foundation for future electronic projects. Design and fabricate you own PCB.