



*KiCad
workshop
for*



HAN_UNIVERSITY
OF APPLIED SCIENCES

Workshop KiCad7

INTRODUCTION WORKSHOP

WRITTEN BY: CASPER R. TAK

Contents

Workshop Version management	2
Workshop Summary: What will you learn in this workshop?.....	3
Getting Started with KiCad for PCB Design.....	4
Step 1: Download and Install KiCad	4
Step 2: Starting Your First Project	5
Step 3: Creating a New Schematic	6
Step 4: Adding Components to the Schematic	7
Step 5: Wiring Components.....	9
Step 6: Assigning Footprints	9
Step 7: Error Checking (ERC).....	10
Step 8: Starting PCB Design	11
Step 9: Creating a Board Outline.....	12
Step 10: Wiring the PCB	13
Step 11: Review and Visualization	15
Step 12: Finishing up	15
Why use KiCad?	16
In short:	16
Detailed:	16

Workshop Version management

Version	Date	Changes/notes
V0.1	xx-8-2023	First version of the document. Barebones version.
V0.2	05-12-2023	Many improvements on the global layout including: added title page, author, date, index, version management etc.

Workshop Summary: What will you learn in this workshop?

In this workshop, you will learn the fundamental steps to get started with KiCad, a powerful tool for designing printed circuit boards (PCBs). The workshop will cover the following key topics:

1. **Downloading and Installation:** You'll learn how to download KiCad and install it on your Windows-computer, ensuring you have the right software to start PCB design.
2. **Creating a New Project:** You'll discover how to create a new project, set up a schematic, and organize your work effectively.
3. **Adding Components:** You'll learn how to add electronic components to your schematic, including resistors, switches, LEDs, and batteries, and arrange them in your design.
4. **Wiring Components:** We'll guide you through the process of connecting the components with wires, ensuring proper connections for your circuit.
5. **Assigning Footprints:** You'll understand how to associate real-world footprints with your schematic symbols, a crucial step for designing PCBs accurately.
6. **Error Checking:** You'll explore how to perform error checks to identify and resolve issues in your design, ensuring it's ready for PCB layout.
7. **PCB Design:** We'll introduce you to PCB layout, where you'll transfer your schematic into a physical board, positioning and connecting components effectively.
8. **Board Outline:** You'll create a board outline, defining the shape and dimensions of your PCB.
9. **Routing:** Learn to route connections on your PCB, ensuring electrical connectivity between components while adhering to good design practices.
10. **Final Review:** We'll show you how to review your PCB design and visualize it in 3D to ensure it meets your project requirements.

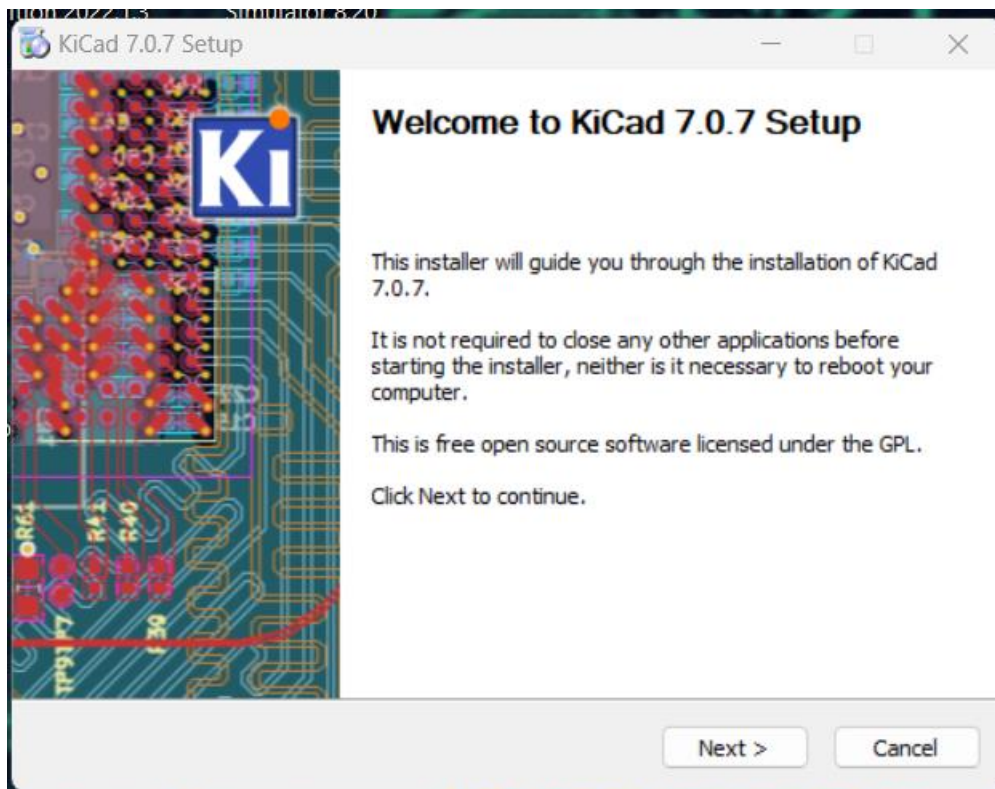
By the end of this workshop, you will have acquired the foundational knowledge and practical skills needed to begin designing your own PCBs using KiCad. Whether you're a beginner or have some experience, this workshop will set you on the path to creating functional and well-designed circuit boards for your projects.

Getting Started with KiCad for PCB Design

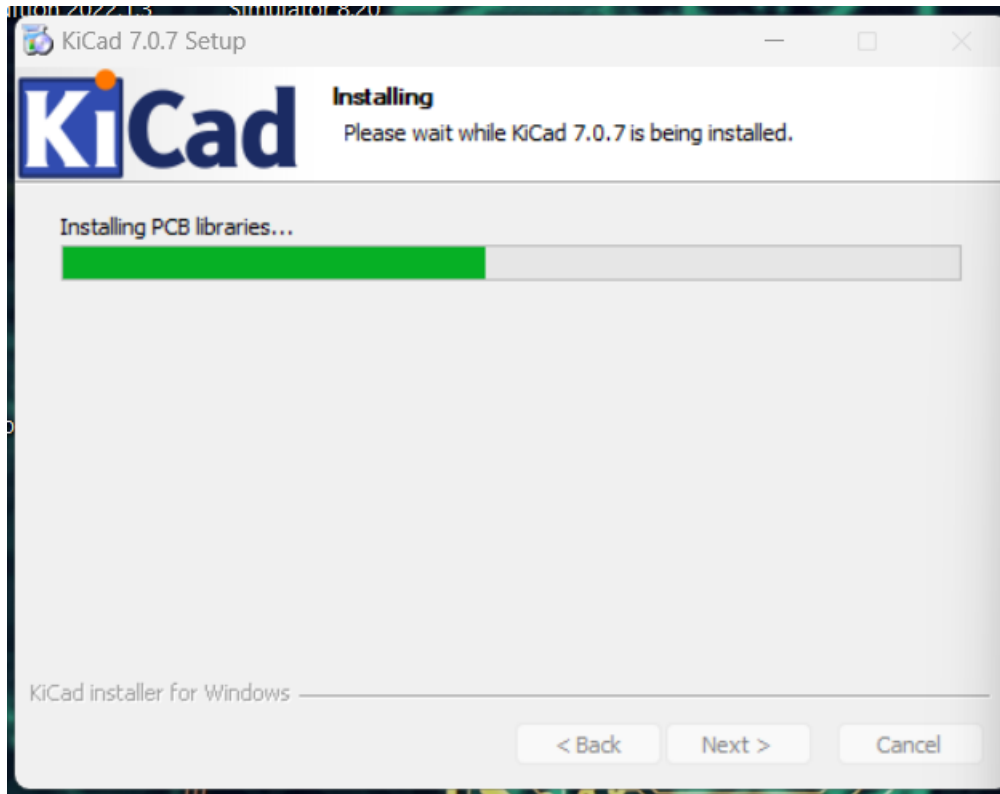
Welcome to the world of PCB design with KiCad! This tutorial will guide you through the process of downloading and setting up KiCad on a Windows computer, creating a simple schematic, assigning footprints, checking for errors, and starting the PCB design. Let's get started!

Step 1: Download and Install KiCad

1. Visit the KiCad website at <https://www.kicad.org/download/>.
2. Choose the operating system you are using (for this tutorial, we'll select Windows).
3. Click on the stable version (e.g., 7.0.7 at the time of writing).
4. Click on "worldwide" (GitHub button) to start the download (approximately 1.1GB in size).
5. Run the downloaded installer. You don't need to change any installation settings.



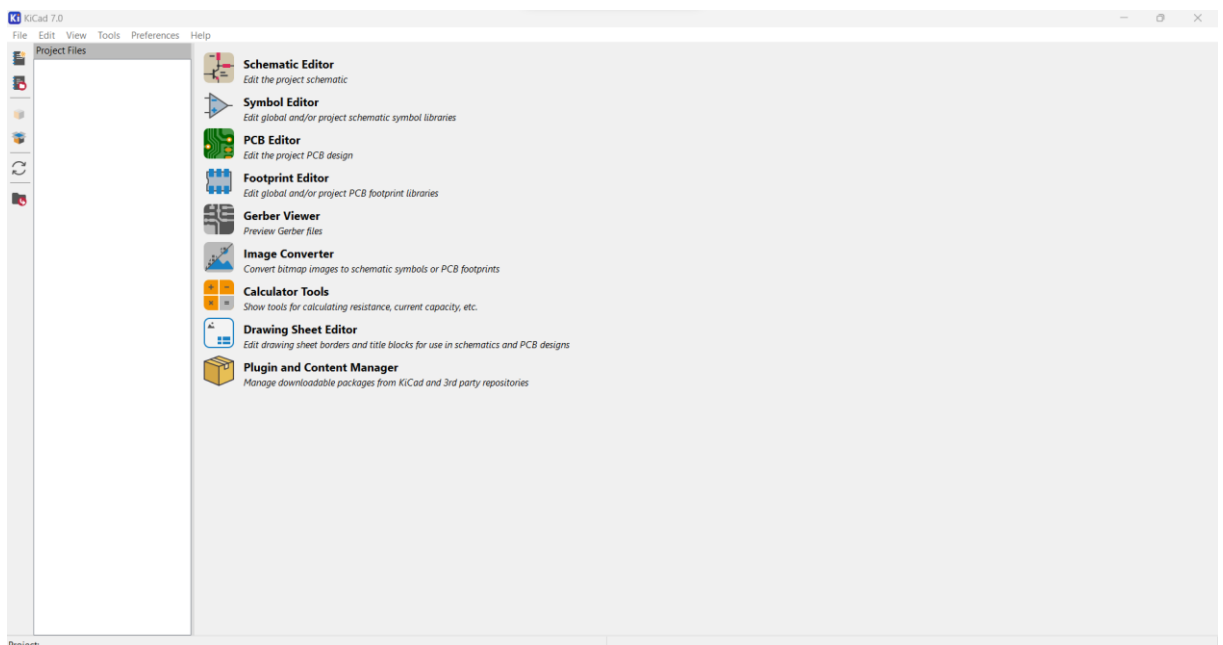
- 6.
7. Click "Next" three times to proceed with the automatic installation (this may take about 3 minutes).



- 8.
9. Click "Finish" when the setup is completed.
10. You should now have a KiCad desktop shortcut.

Step 2: Starting Your First Project

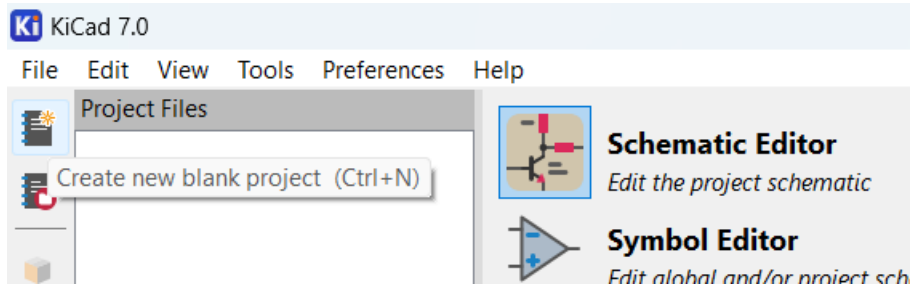
1. Click the KiCad desktop icon to open KiCad.
2. If you receive any notifications about auto-updating, choose your preference.
3. You'll be greeted with the main page.



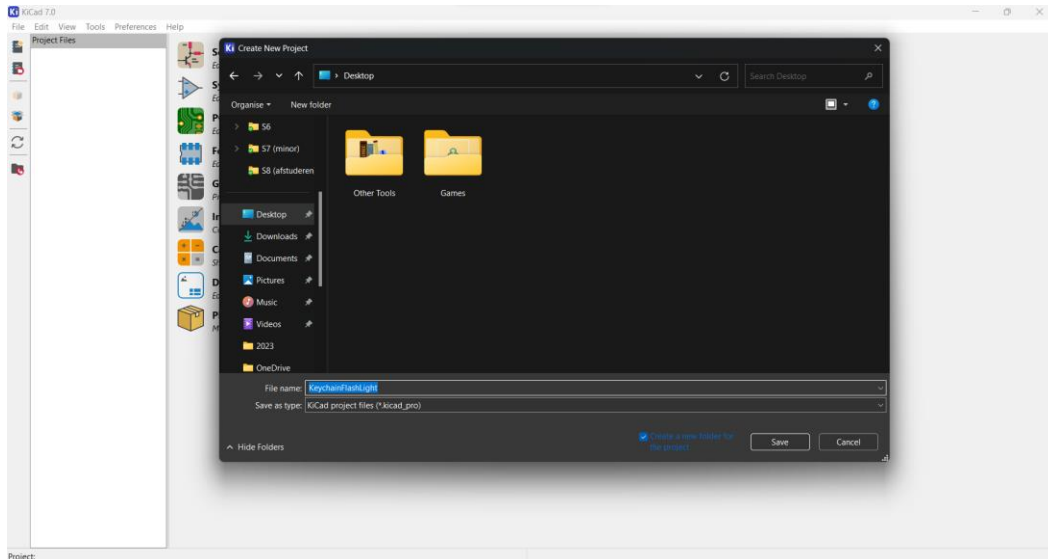
- 4.

Step 3: Creating a New Schematic

1. Create a new project by choosing a destination to save it and giving it a name.

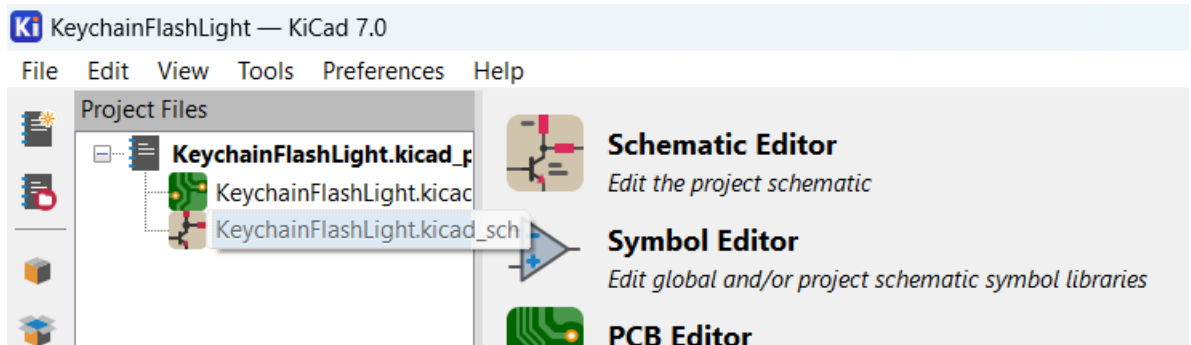


2.



3.

4. Two files will be created; open the schematic (the bottom one) by double-clicking it.

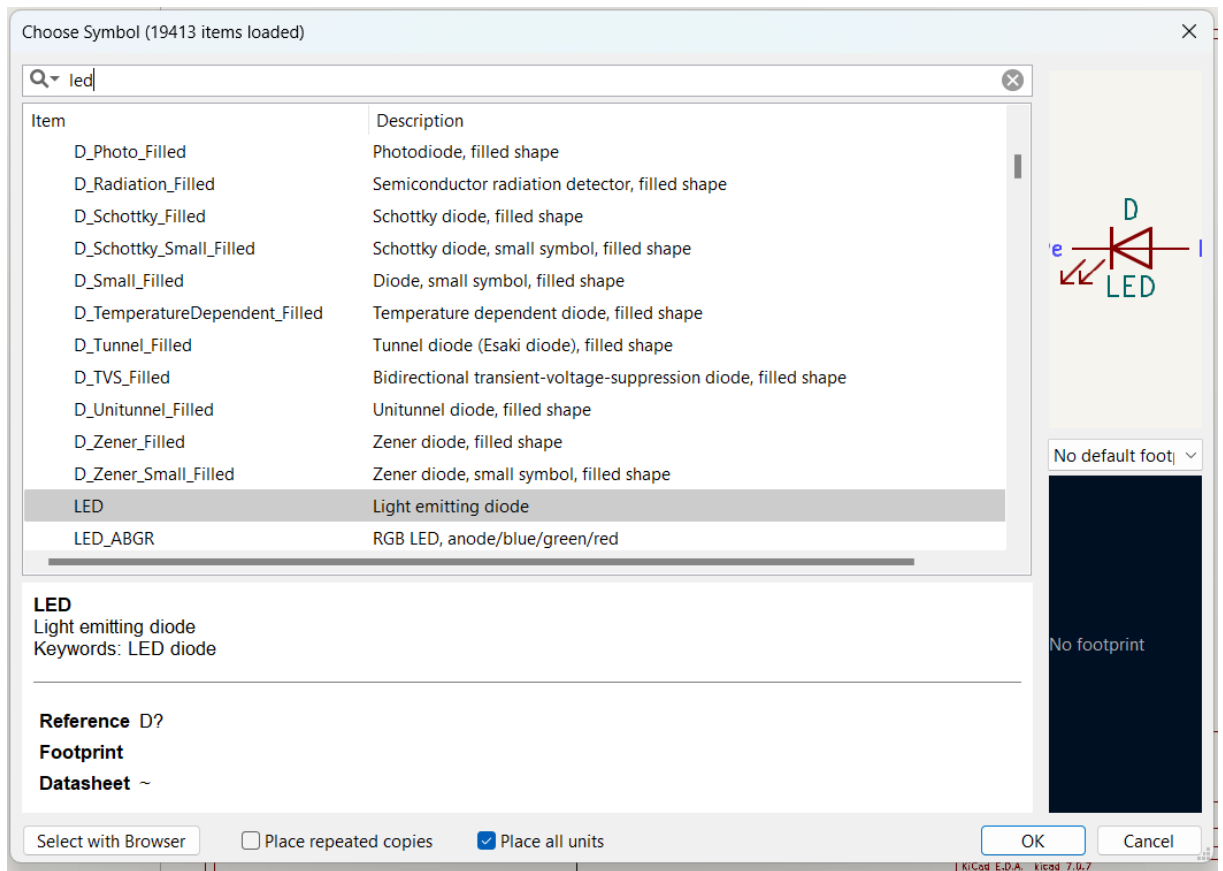


5.

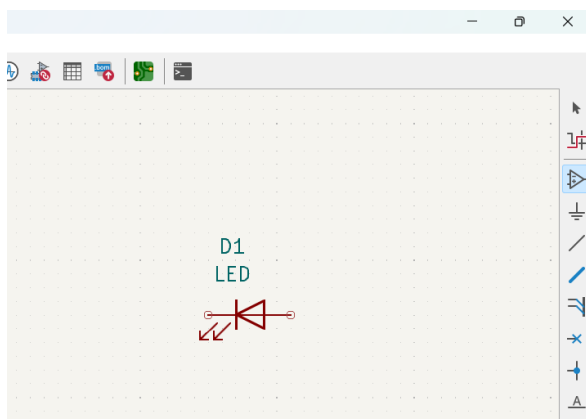
6. You are now in the schematic designer/editor.

Step 4: Adding Components to the Schematic

1. Press "A" on your keyboard to open the symbol selector.
2. Type "LED" in the filter bar, select the LED.



- 3.
4. You can place the resistor on the schematic by pressing the left mouse button.

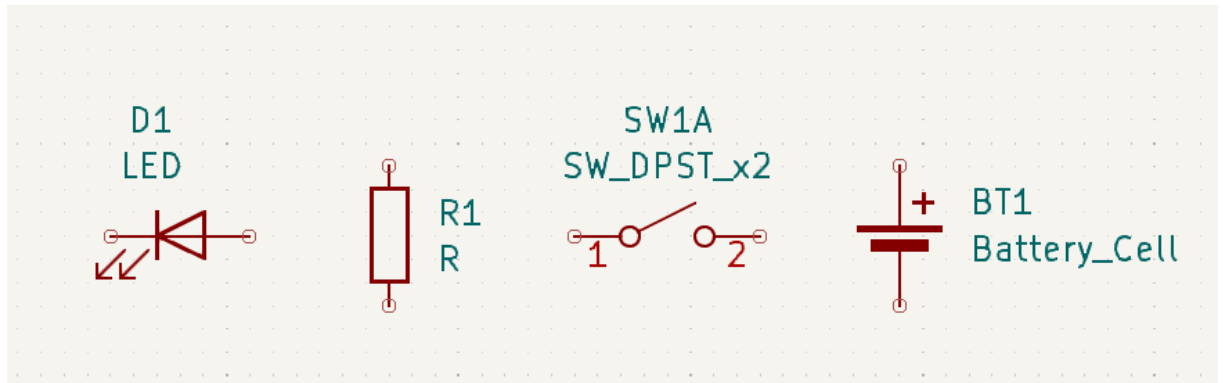


- 5.
6. Press "escape" to stop the component selector. It is still used as you can see in the picture above where the component icon is now blue. It will return to the mouse icon after pressing "escape".

Press "A" again to go back into the component selector and add the following components:

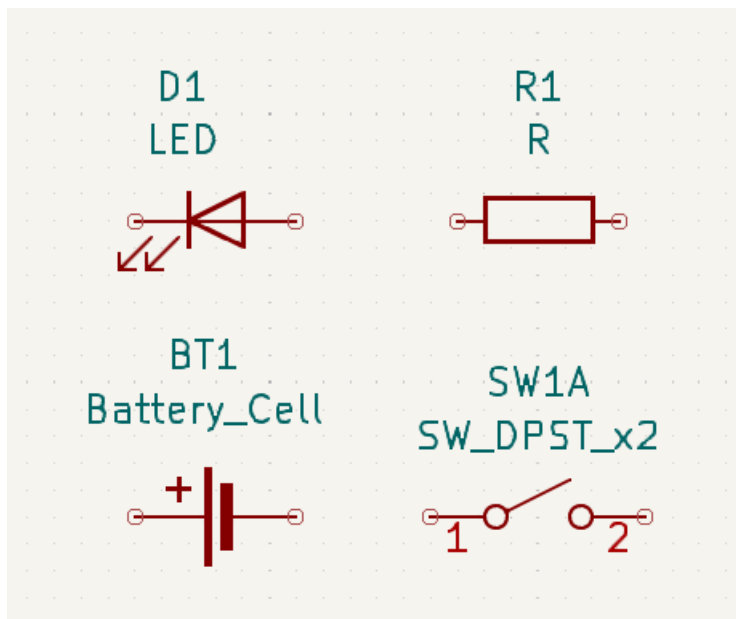
- "R" (resistor)
- "SW_DPST_x2" (Button switch)
- "Battery_Cell" (button cell battery).

7. You will have the following components afterward:



8.

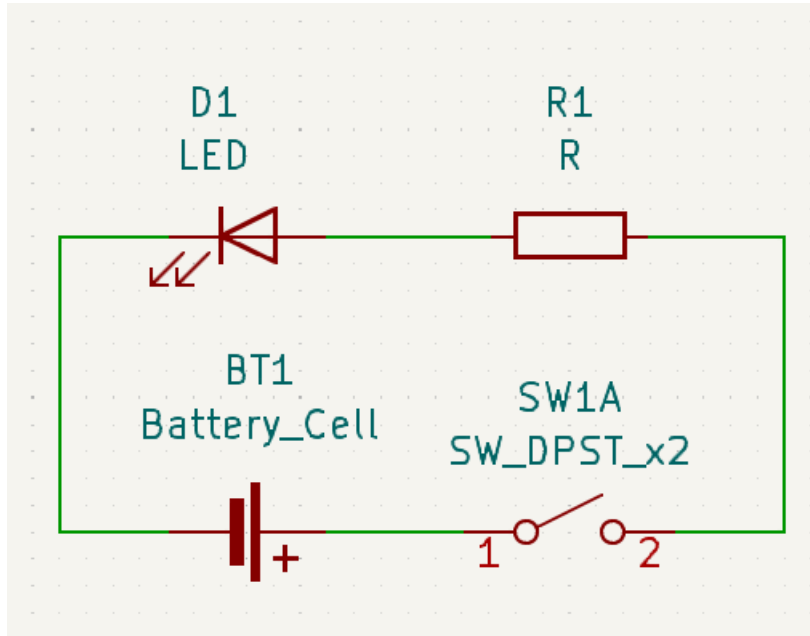
9. Organize these components on the schematic. You can rotate components by using "R" when a component is selected.



10.

Step 5: Wiring Components

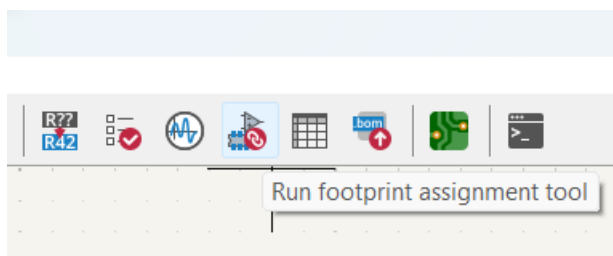
1. Press "W" to start wiring components together. You can start anywhere on the schematic.
2. Connect the components by drawing wires between them. You can end a wire by pressing escape or by connecting a wire to another component.



- 3.
4. Ensure correct connections. As you can see, I had the battery connector backwards before. Rotating it fixed the polarity issue.

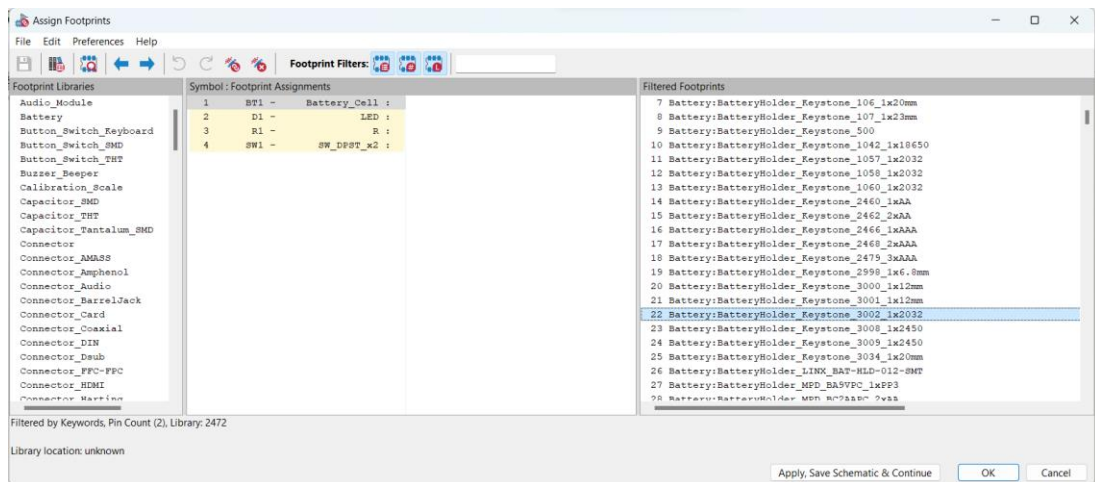
Step 6: Assigning Footprints

1. Click the "footprint assignment tool" button.

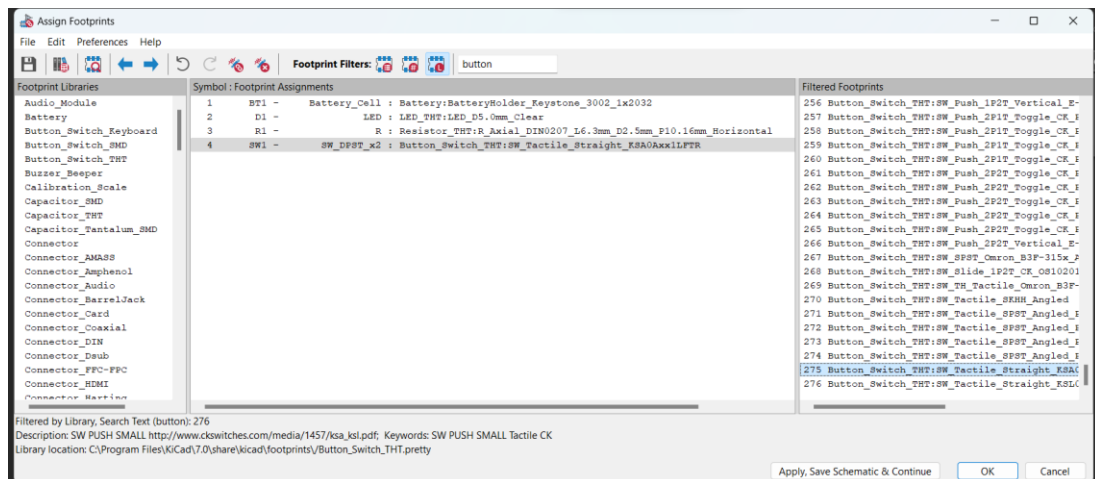


- 2.

3. Select the three footprint filters above for easier component selection.



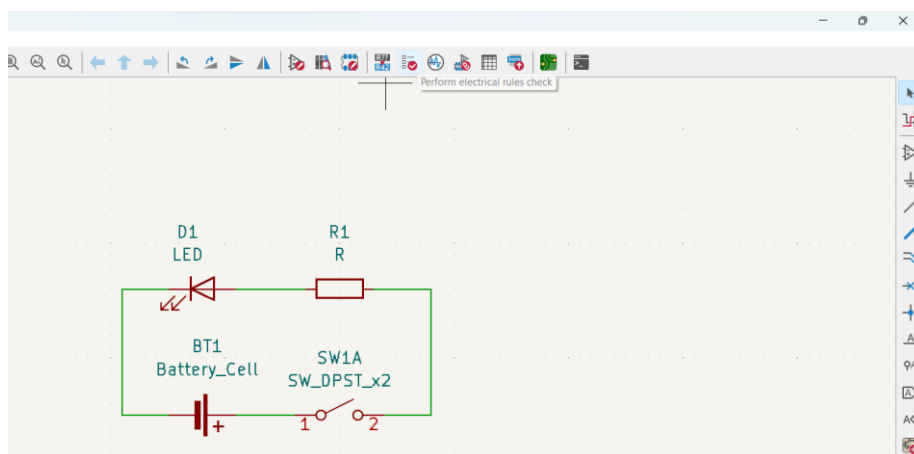
- 4.
5. Double-click a component to assign a footprint (e.g., Batteryholder: 27, LED: 68, Resistor: 57, and for the button switch: 275). I disabled the footprint filters to find the last part.



- 6.

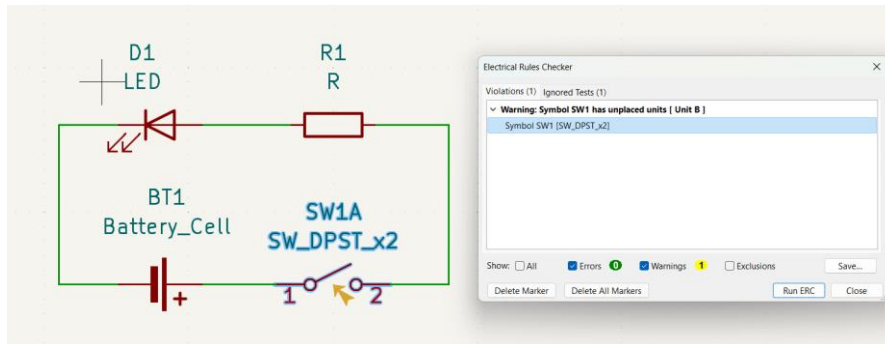
Step 7: Error Checking (ERC)

1. Press the DRC button.



- 2.
3. Click "Run ERC."

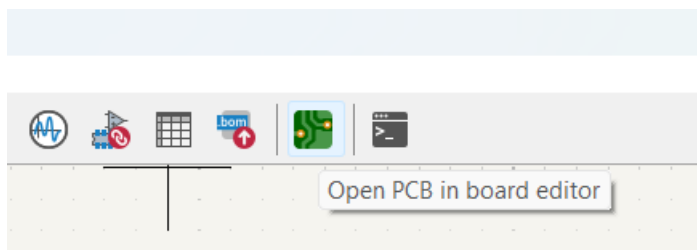
- Address any warnings or errors if necessary. In this tutorial, a library-related warning is expected, but can be ignored.



5.

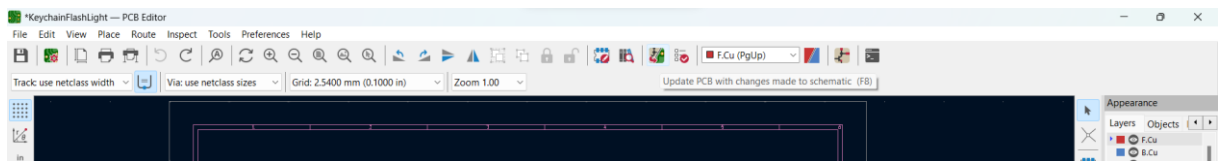
Step 8: Starting PCB Design

- From the schematic editor, open the PCB editor.



2.

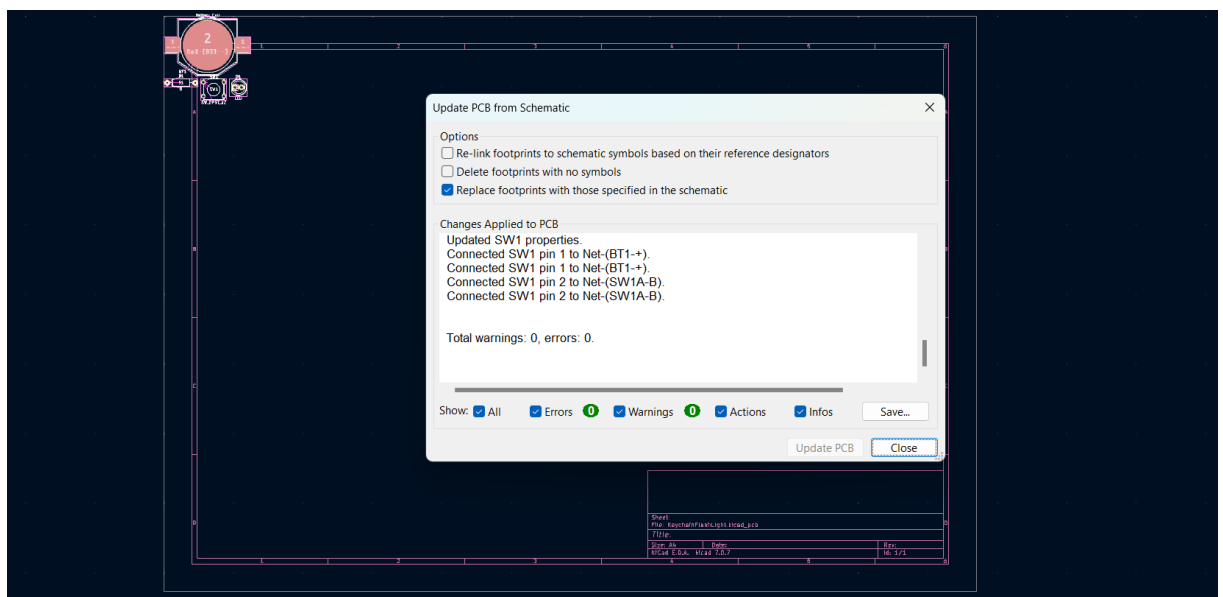
- Click the "Update PCB with changes made to schematic" button (or press F8).



4.

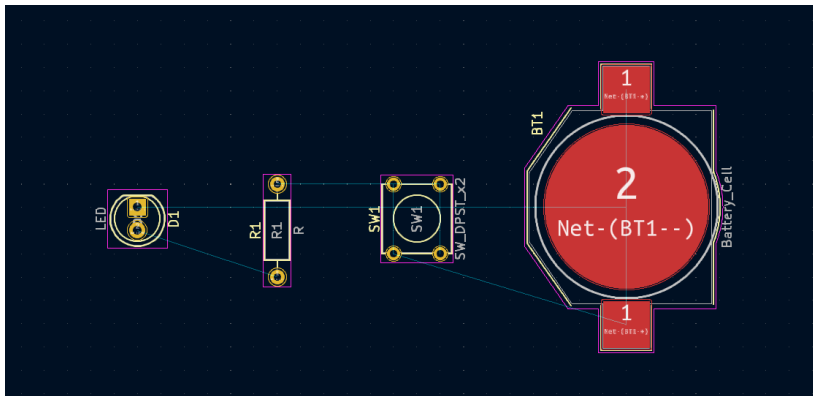
- A window will pop up. Select "update PCB".

- Close the window.



7.

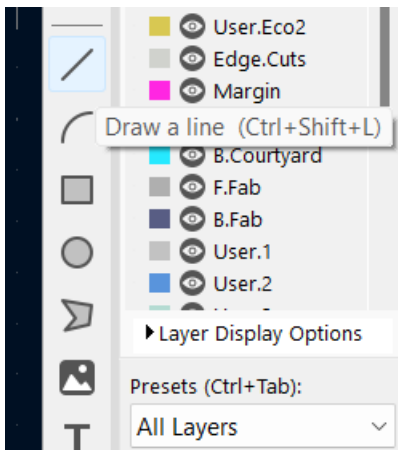
8. Move and rotate components on the PCB layout screen as desired.



9.

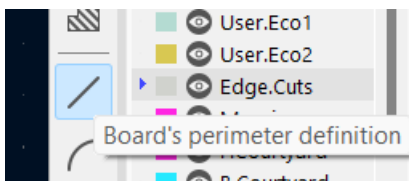
Step 9: Creating a Board Outline

1. Select "Draw a Line" on the right side of the screen.



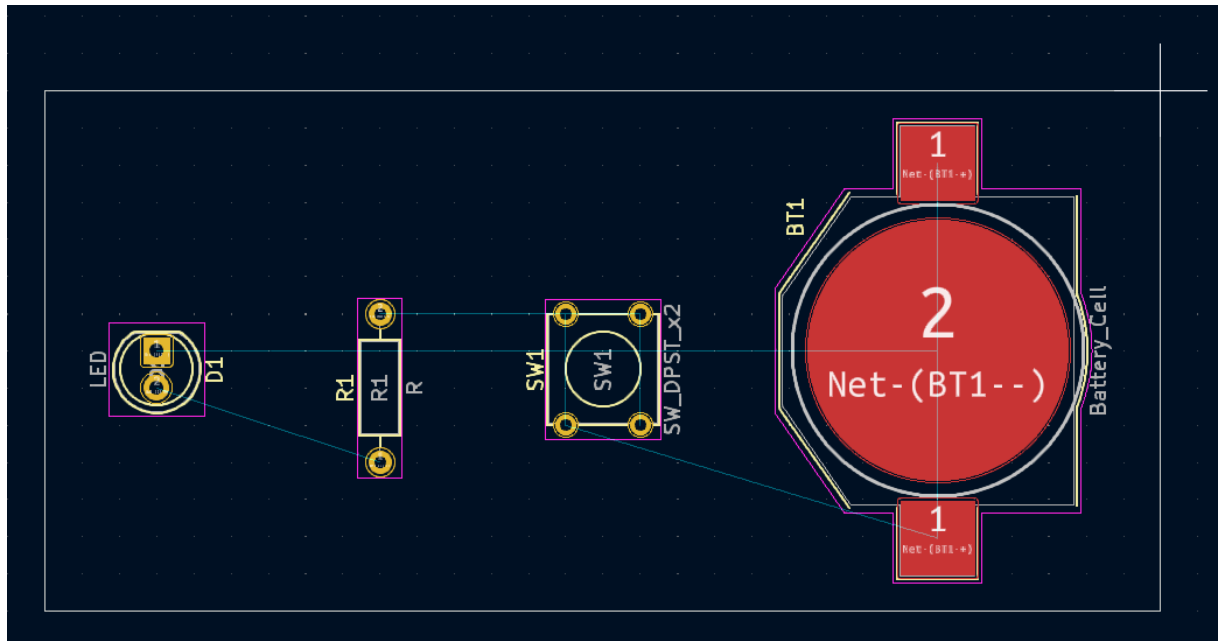
2.

3. Choose "Edge Cuts" in the selector.



4.

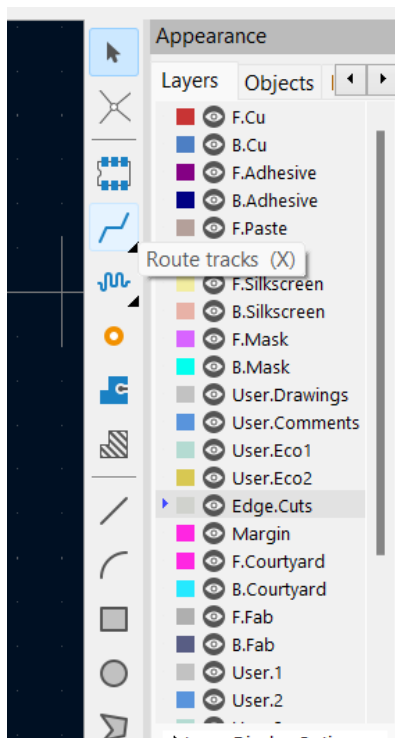
5. Create a rectangular board outline around the components.



- 6.

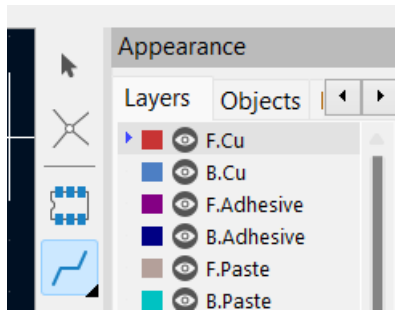
Step 10: Wiring the PCB

1. Select the wiring tool (or press "X").

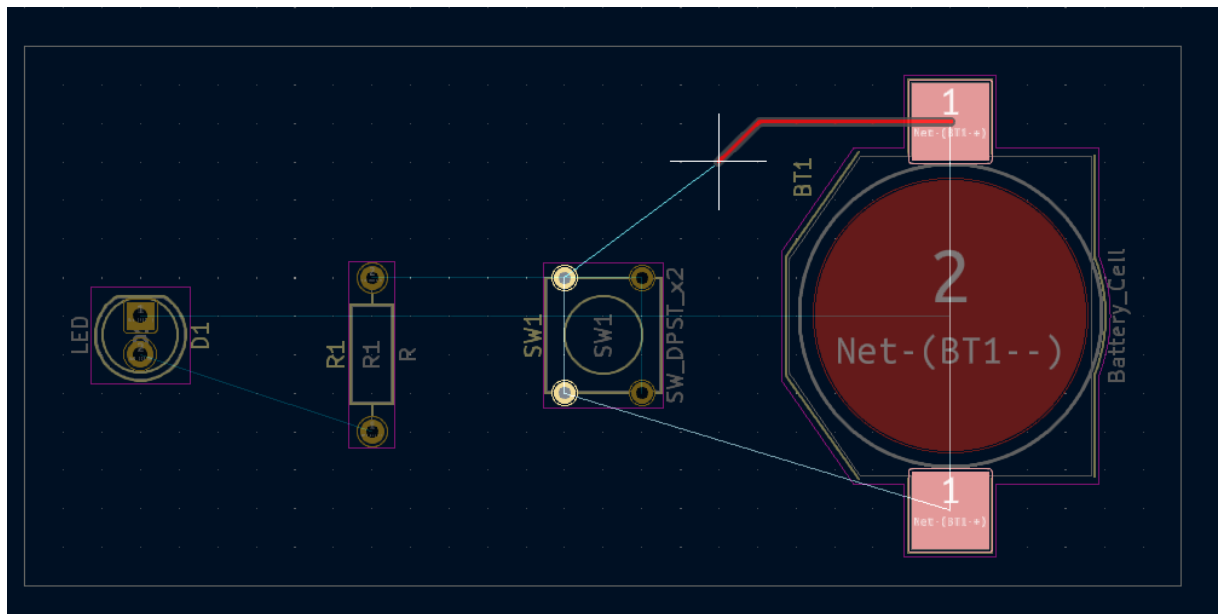


- 2.

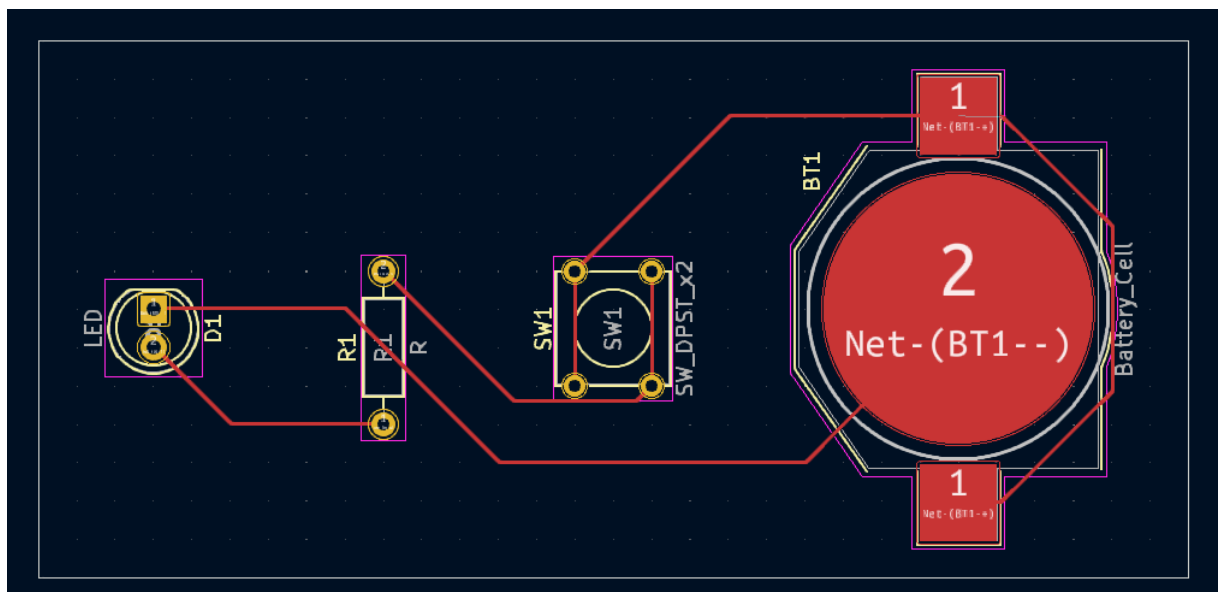
3. Choose the F. Cu (front copper) layer.



- 4.
5. Connect the pads by clicking them. KiCad will guide you through the connections.



- 6.
7. If a mistake is made, press escape, or use Ctrl + Z to undo.



- 8.

Step 11: Review and Visualization

1. Review your PCB design in the 3D viewer (press ALT + 3).
2. Ensure your design meets your requirements.

Step 12: Finishing up

You have successfully created your first PCB design using KiCad. You can save the file or export it to whatever format you like. While this design is basic, it provides a solid foundation for more advanced projects. In future tutorials, you can explore advanced PCB design techniques to make your designs more personalized and elegant. Good luck with your PCB design journey!

Why use KiCad?

In short:

In summary, KiCad's open-source nature, cross-platform support, active community, regular updates, and comprehensive features make it an excellent choice for PCB design, especially for those who prefer free and versatile tools. While other PCB design software like Eagle, EasyEDA, or Altium may have their strengths, KiCad's combination of features, community, and cost-effectiveness makes it a compelling option for many designers.

Detailed:

Using KiCad for PCB design, as described in this tutorial, has several advantages over other popular tools like Eagle, EasyEDA, or Altium. Here's a small reasoning to choose KiCad:

1. **Open Source and Free:** KiCad is open-source software, which means it's free to use without any licensing costs. This accessibility makes it a great choice for hobbyists, students, and small businesses with limited budgets.
2. **Cross-Platform Compatibility:** KiCad is available for Windows, macOS, and Linux, making it versatile and accessible no matter what operating system you prefer. This cross-platform support ensures a consistent experience for users on different systems.
3. **Community and Support:** KiCad has a growing and active user community. You can find plenty of tutorials, forums, and user-contributed libraries, which can be invaluable for beginners and experienced designers alike. This strong community support ensures you can easily find help when needed.
4. **Regular Updates:** KiCad receives regular updates and improvements, as seen in the tutorial mentioning version 7.0.7. This means that the software stays up to date with industry standards and technologies.
5. **Integration:** KiCad offers a full suite of tools for schematic capture, PCB layout, and even 3D visualization. All these tools are integrated seamlessly within the same software, reducing the need to export and import data between different programs.
6. **Customization and Extensibility:** KiCad is highly customizable. You can create custom footprints, symbols, and even scripts to automate repetitive tasks, tailoring the software to your specific needs.
7. **No Vendor Lock-In:** Unlike some proprietary tools like Altium, KiCad doesn't lock you into a specific vendor's ecosystem. You have full control over your designs, and you're not reliant on a single company's continued support.
8. **Reliability and Stability:** KiCad is known for its stability. It rarely crashes or encounters bugs, ensuring that your work is safe, and your design process is efficient.
9. **Large Component Library:** KiCad has a vast and growing component library, and its open-source nature means that users continually contribute to it. This means you'll likely find the components you need for your projects.
10. **Export Options:** KiCad provides a variety of export options for Gerber files, which are essential for manufacturing your PCBs. It supports industry-standard formats, making it compatible with most PCB manufacturers.