# KiCad

## 

## Why KiCad?Afbeelding met tekst Automatisch gegenereerde beschrijving

The Rastaban project requires a printed circuit board (PCB) to be used and moved safely from one place to another. Additionally, a PCB is the best option for producing the product in larger quantities. I chose KiCad (6) for several reasons.

First, it is open-source and free, making it accessible to many hobbyists and professionals. Second, it is a good software to learn how to design PCBs. Previously, I used EasyEda, which is proprietary. However, this comes with ads, limited access to certain features, and increased vulnerability to losing or stealing designs. In addition, I was unable to install useful plugins like an interactive BOM file, 3D model archiver, and fabrication toolkits. These limitations led me to stop using EasyEda. However, it should be noted that using EasyEda's library may be the fastest and cheapest way to produce a fully assembled PCB. If you plan to assemble your PCBs yourself, this is no longer an issue. KiCad6 also has plugins that allow you to easily obtain LCSC part numbers for PCB assembly with JLCPCB.

## Plugins

To create an even better PCB and make my workflow easier, I downloaded a few plugins. The plugins I downloaded are as follows:

|  |  |  |
| --- | --- | --- |
| Name | Function | Download location |
| KiCad JLCPCB tools | This plugin allows you to search the JLCPCB parts database, assign LCSC article numbers to your parts, generate production files for JLCPCB and much more. | <https://github.com/Bouni/kicad-jlcpcb-tools> |
| Interactive HTML BOM | This plugin generates convenient BOM listing with ability to visually correlate and easily search for components and their placements on the pcb. | Build into KiCad6 |
| PCB action tools | Annular Ring Checker, Snap Selected Footprint(s) to Grid, Fabrication Footprint Position, Move Selected Drawings to chosen Layer, Export pcb technical layers to DXF, Checking 3D missing models | Build into KiCad6 |
| Archive 3D models | Copies footprint models to the project local subfolder and remaps all the links within the used footprints. | Build into KiCad6 |
| Place Footprints | Arrange sequentially numbered footprints or footprints from multiple hierarchical sheets in linear, circular or matrix arrangement. This plugin works on footprints already present in the layout, so that layout and schematics stay in sync. | Build into KiCad6 |
| Round Tracks | Algorithmically smooth tracks in a predictable manner. Useful for flex PCBs, or just because it looks cool. | Build into KiCad6 |
| Length matching | Track Length Calculator | Build into KiCad6 |
| Freerouting | Auto router for KiCad. It draws all the connections between components for you. Be warned: Auto routing should never be used carelessly, always check the results. | Build into KiCad6 (requires Java) |

## How to learn KICAD6

I learned KICAD6 by following [an online course.](https://techexplorations.com/so/kicad-like-a-pro-3rd-edition/) But I could also have used tutorials on YouTube or read books on it. KICAD itself is not difficult to use, but there are a lot of buttons, some of which are important, some are a bit redundant, which can be confusing. The best way to learn KICAD is to just start creating a schematic with some (maybe 4) parts and connecting them together. Then you can start creating a PCB in the “PCBNEW” section.

The design approach  
Creating a PCB is always following trough 2 stages

1. Create the schematic
2. Create the PCB

After having done this, your design is finished. Of course there are more details to both design stages an those are described in the following pictures.

