

# Introduction to KiCad: Schematic and PCB Editor Tools

## Schematic Editor Tools

1. **Add Symbol**
  - Use this tool to place electronic components (like resistors, capacitors, ICs) from the symbol library onto your schematic. Each symbol represents a part of the circuit.
2. **Wire Tool**
  - The wire tool connects components together by drawing wires between pins. This is how you represent electrical connections in the schematic.
3. **Add Power Port**
  - Power ports are symbols like GND (ground) and VCC (power supply) that connect components to power. This tool helps you add them to your schematic.
4. **Annotate Schematic**
  - This tool automatically assigns unique reference designators (like R1, C1) to each component, making your schematic organized and clear.
5. **ERC (Electrical Rules Check)**
  - The ERC tool checks your schematic for electrical errors, such as unconnected pins or incorrect connections. This helps ensure that your design is correct before moving to the PCB stage.
6. **Generate Netlist**
  - The netlist is a file that describes the connections between components in your schematic. It's essential for transferring your design to the PCB layout.
7. **Assign Footprints**
  - After creating the schematic, each symbol needs to be assigned a physical footprint (the actual shape and size of the component on the PCB). This tool lets you match symbols with their corresponding footprints.
8. **Add No Connect Flag**
  - This tool is used to mark any pins that should not be connected in your design, helping to avoid confusion during the design review.
9. **Add Text and Graphics**
  - Add notes or graphical elements to your schematic for clarity or documentation purposes.
10. **Bill of Materials (BOM) Generation**
  - Automatically generate a list of all the components used in your schematic, which can be useful for purchasing and assembly.

## PCB Editor Tools

1. **Add Footprint**
  - In the PCB editor, this tool allows you to place the physical component footprints (which were assigned in the schematic editor) onto the board.
2. **Route Tracks**

- The track routing tool is used to draw electrical traces that connect the footprints on the PCB. These traces represent the wires that carry electrical signals between components.
- 3. **Add Via**
  - Vias are small holes that connect tracks on different layers of a multi-layer PCB. This tool lets you place vias to facilitate connections between layers.
- 4. **Zone Fill (Copper Pour)**
  - Copper zones are large areas of copper on the PCB, typically used for ground or power planes. This tool creates these fills, helping with signal integrity and thermal management.
- 5. **Design Rules Check (DRC)**
  - The DRC tool checks your PCB layout for rule violations, such as too-close traces or incorrect layer usage. It's important to run DRC before finalizing your design.
- 6. **Measure Tool**
  - This tool allows you to measure distances between different features on the PCB, ensuring that components are placed correctly and that clearances are respected.
- 7. **3D Viewer**
  - The 3D viewer lets you see a three-dimensional representation of your PCB, including components and traces. This is helpful for checking the physical layout and identifying any potential issues before manufacturing.
- 8. **Add Text and Graphics**
  - Similar to the schematic editor, you can add text labels and graphical elements to your PCB layout. This can include part numbers, company logos, or other annotations.
- 9. **Footprint Editor**
  - If you need to modify or create custom footprints for components, the footprint editor provides tools for designing and saving these footprints.
- 10. **Gerber File Generation**
  - Once your PCB design is complete, use this tool to generate Gerber files, which are the industry-standard files needed for PCB manufacturing.