

# KiCAD

(/'ki: ,kæd/ KEE-kad)

# What is KiCAD

- KiCAD lets you design your own PCB and is a type of software known as Electronic Design Automation (EDA).
- KiCAD is open source and free.
- Alternatives: Autodesk EAGLE, Altium Designer, EasyEDA

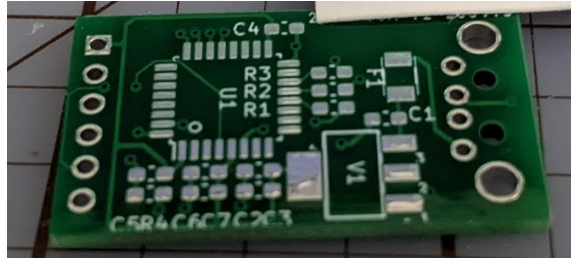
# Fabrication companies

The following are companies I have used. Not an endorsement. Many fabrication companies exists.

- JLCPCB <https://jlcpcb.com/>
- OSHPark <https://oshpark.com/>

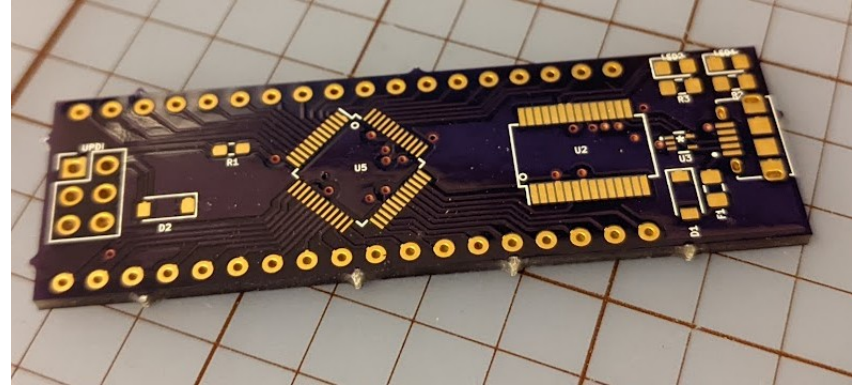
# JLCPCB

- Ships from China
- \$2 USD for 2 layers, up to 100mmx100mm, 5 pieces
- FR-4 Tg140, HASL plating
- Multiple board thickness and solder mask colors to choose from
- Many add-on options
- Order number is printed on PCB (removable with added cost)
- Higher quality boards, plating, options are available, but comes at a higher cost



# OSHPark

- Ships from USA
- \$5 USD / Square Inch, 3 pieces
- Free shipping option (even for Canada)
- FR4 175Tg, ENIG plating (gold)
- 1.6mm, 8mm and flex boards
- Purple solder mask
- Produced boards have tabs that need to be filed off



# KiCAD UI

- KiCAD is a collection of applications
- KiCAD is shortcut key driven, get used to it!
- Shortcut keys are different between programs



## Schematic Editor

*Edit the project schematic*



## Symbol Editor

*Edit global and/or project schematic symbol libraries*



## PCB Editor

*Edit the project PCB design*



## Footprint Editor

*Edit global and/or project PCB footprint libraries*



## Gerber Viewer

*Preview Gerber files*



## Image Converter

*Convert bitmap images to schematic symbols or PCB footprints*



## Calculator Tools

*Show tools for calculating resistance, current capacity, etc.*



## Drawing Sheet Editor

*Edit drawing sheet borders and title blocks for use in schematics and PCB designs*

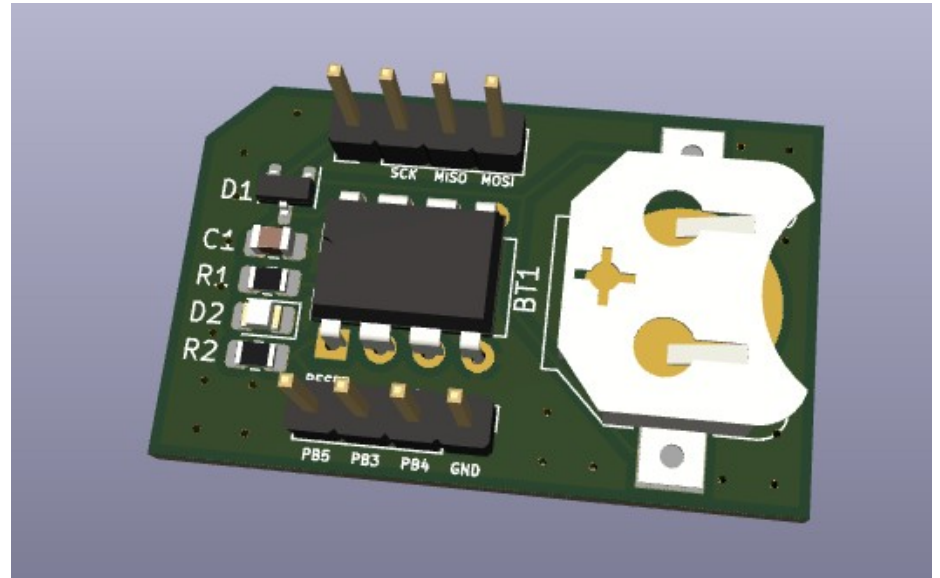


## Plugin and Content Manager

*Manage downloadable packages from KiCad and 3rd party repositories*

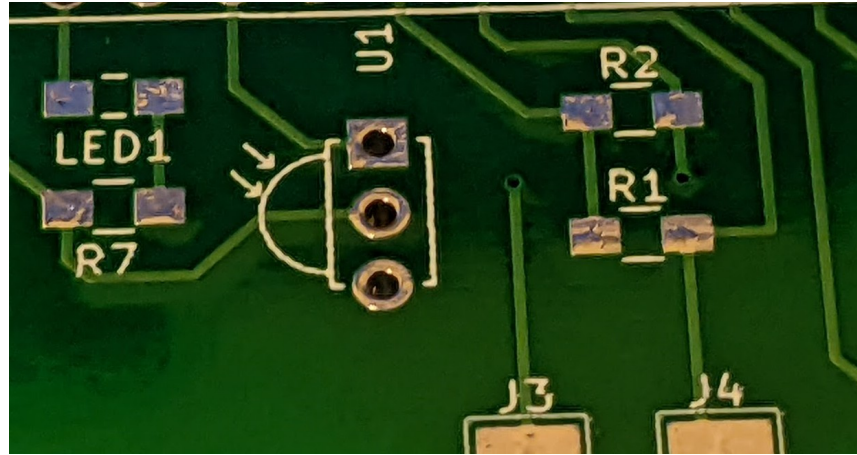
# Demo

- ATtiny85 breakout board
- 1 LED
- Breadboard compatible headers



# Reference Designators

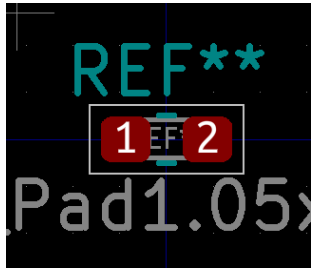
- Uniquely identify components on a board
- Usually consists of one or two letters and a number
- Common examples:
  - R1, R2, ... Resistors
  - D1, D2, ... Diodes
  - C1, C2, ... Capacitors
  - Q1, Q2, ... Transistors
  - U1, U2, ... ICs
  - F1, F2, ... Fuses



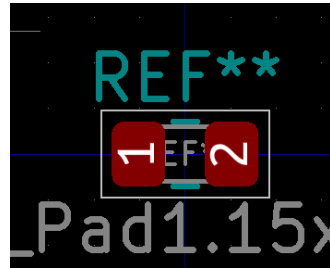


# Footprints

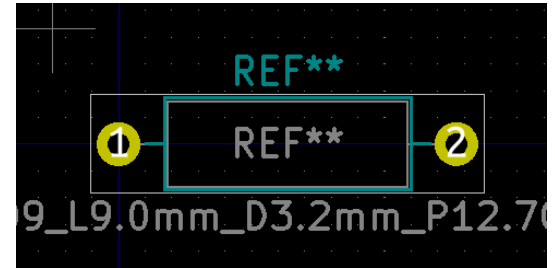
- Template that defines the physical layout of the pads in a component
- Components have many variations: THT, SMD, DIP, etc.



0603 resistor



1206 resistor



Through hole resistor

# Electrical Rules Checker (ERC)

- Basic check on your schematics for errors
- Doesn't catch everything. Rely on yourself to avoid mistakes.

# Design Rules

- Enforce basic rules to meet the design requirements by the PCB fabrication company.
- Rules are different from company to company. Obtain the design rules from the fabrication company by checking their website.
- Even within the same fabrication company, the rules may be different based on the type of board. Always check the website.

# Netclass Editor

- Specify clearances, trace widths, via diameters by defining net classes

The screenshot displays the Netclass Editor interface. At the top, there is a table defining net classes with columns for Clearance, Track Width, Via Size, Via Hole,  $\mu$ Via Size, uVia Hole, DP Width, and DP Gap. Below this table are controls for adding (+) and deleting (trash) net classes. The main area is divided into two sections: 'Filter Nets' and 'Assign Net Class'. The 'Filter Nets' section includes input fields for 'Net class filter' and 'Net name filter', along with 'Show All Nets' and 'Apply Filters' buttons. The 'Assign Net Class' section includes a 'New net class' dropdown and 'Assign To All Listed Nets' and 'Assign To Selected Nets' buttons. On the right side, a list of nets is shown with their corresponding assigned net classes.

Net Class	Clearance	Track Width	Via Size	Via Hole	$\mu$ Via Size	uVia Hole	DP Width	DP Gap
Default	0.2 mm	0.25 mm	0.8 mm	0.4 mm	0.3 mm	0.1 mm	0.2 mm	0.25 mm
Power	0.2 mm	0.3 mm	0.8 mm	0.4 mm	0.3 mm	0.1 mm	0.2 mm	0.25 mm

Filter Nets

Net class filter:

Net name filter:

Show All Nets Apply Filters

Assign Net Class

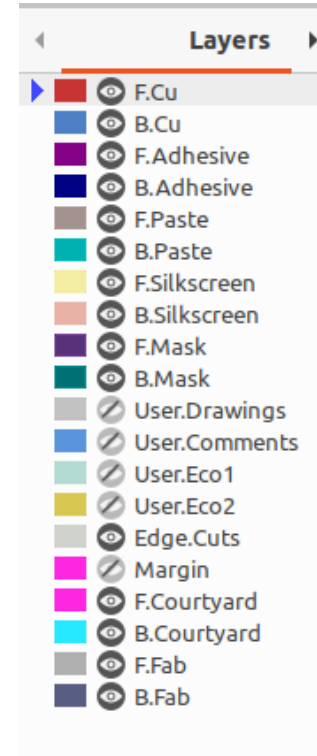
New net class:

Assign To All Listed Nets Assign To Selected Nets

Net	Net Class
/5V	Power
/PB0	Default
/PB1	Default
/PB2	Default
/PB3	Default
/PB4	Default
/PB5	Default
GND	Power
Net-(BT1-Pad1)	Default
Net-(C1-Pad1)	Default

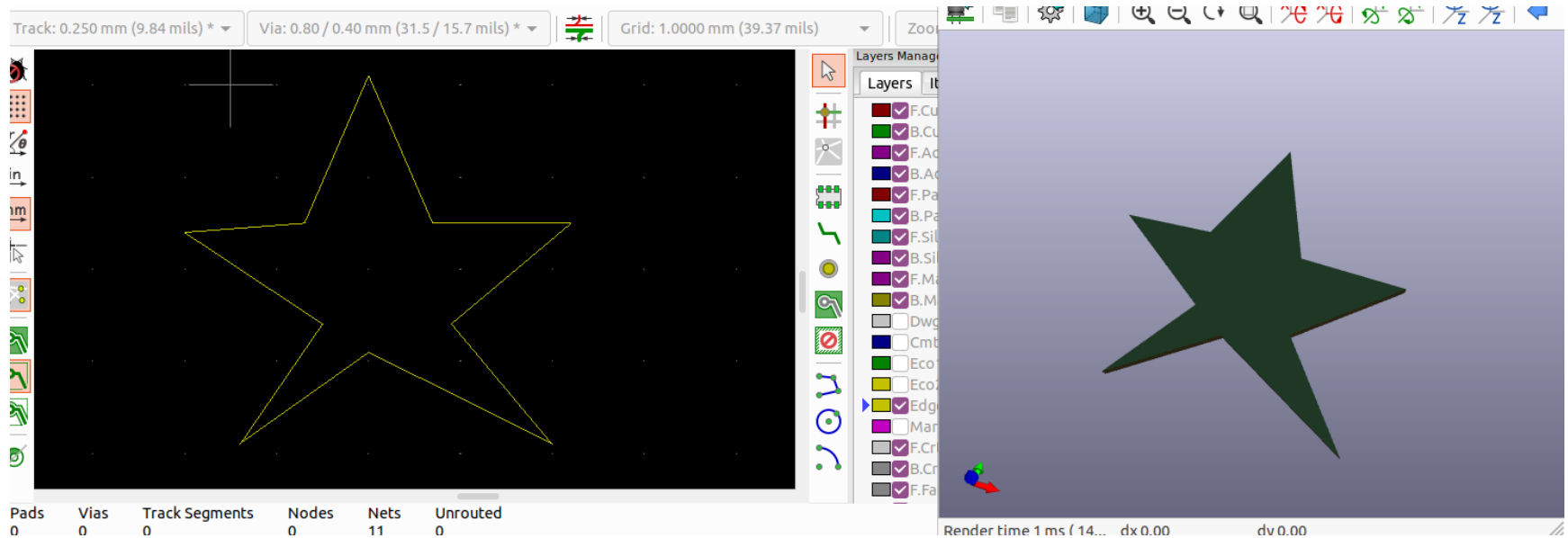
# Layers (2-layer PCB)

- F.Cu, B.Cu  
Copper layers
- F.Paste, B.Paste  
Solder paste layers
- F.SilkS, B.SilkS  
Silkscreen
- F.Mask, B.Mask  
Solder mask
- Edge.Cuts  
PCB board outline
- F.CrtYd, B.CrtYd  
Courtyard layers
- F.Fab, B.Fab  
Fabrication layers



# Edge Cuts

- Define the shape of your PCB



# Autorouting

- KiCAD does not natively support auto-routing
- <https://freerouting.org/> - Standard alone auto-router which can be used
- Download it from: <https://github.com/freerouting/freerouting/releases>

# Gerbers

- Final file format for PCB production
- File → Plot...
- Consult fabrication company for best options

