KiCAD

(/ˈkiːˌkæd/ 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, Altinum Designer, EasyEDA

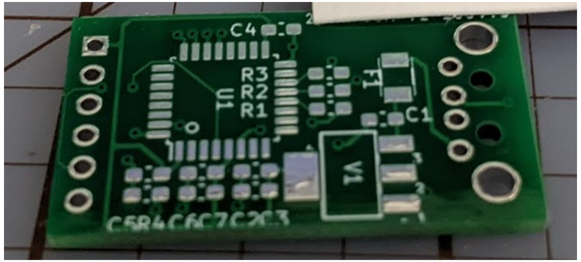
Fabrication companies

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

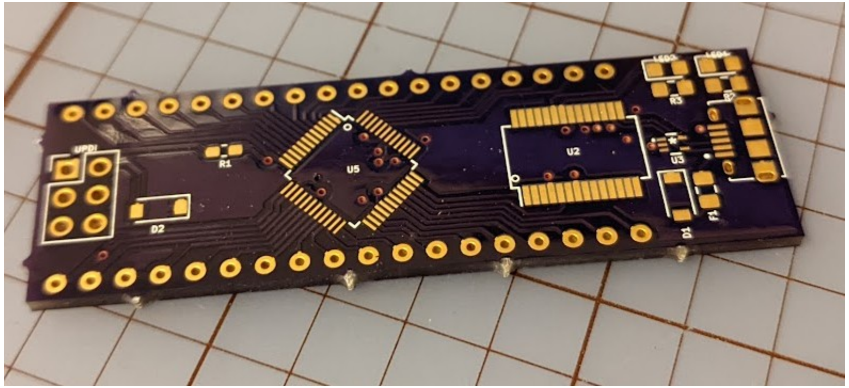
- JLCPCB https://jlcpcb.com/
- OSHPark https://oshpark.com/

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



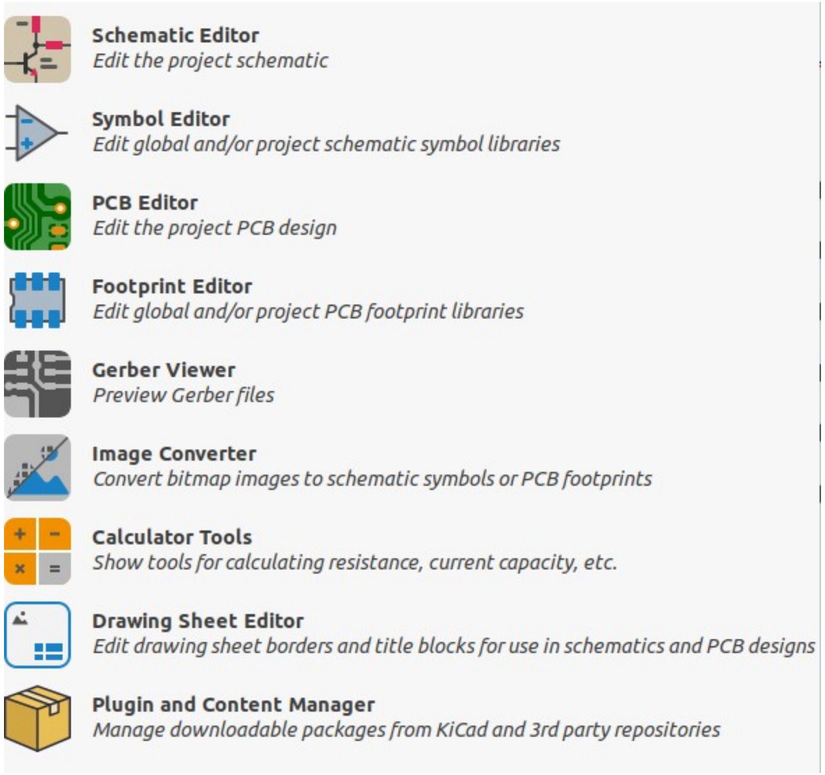
OSHPark



- 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

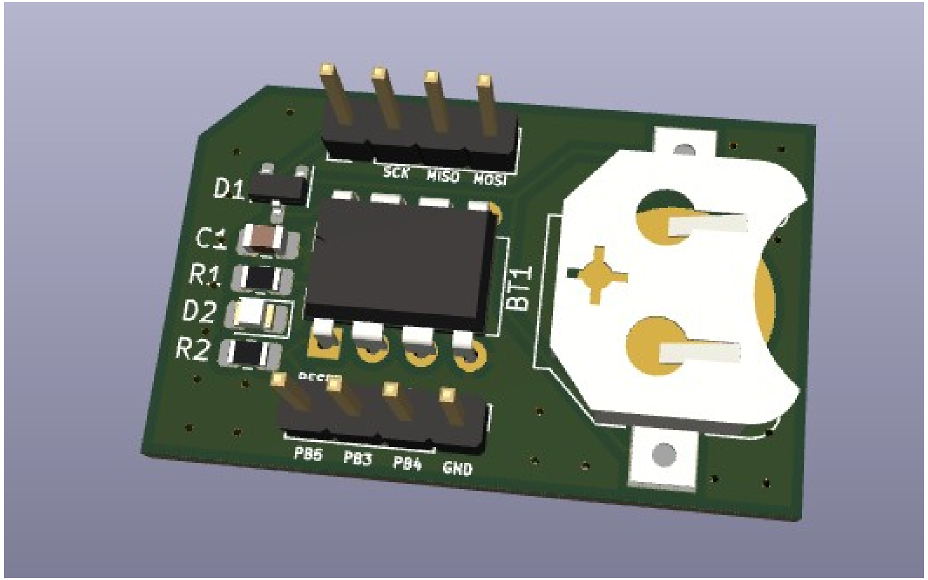
KiCAD UI

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



Demo

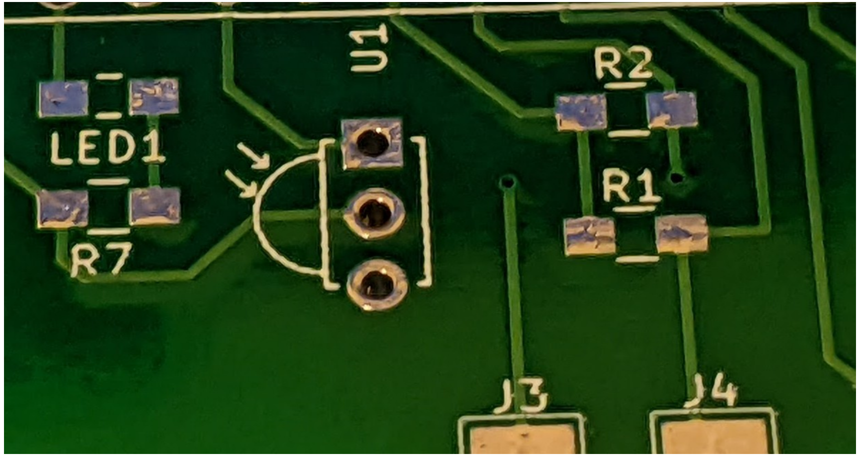
- ATTiny85 breakout board
- 1 LED
- Breadboard compatible headers



Reference Designators

- Uniquely identify components on a board
- Usually consists of one or two letters and a number
- Common examples:

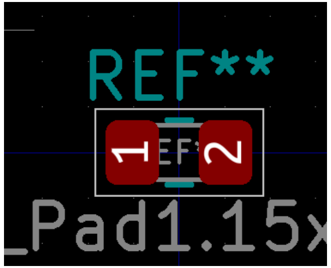
- R1, R2, …	Resistors
- D1, D2, …	Diodes
- C1, C2, …	Capacitors
- Q1, Q2, …	Transistors
- U1, U2, …	ICs
- F1, F2, …	Fuses



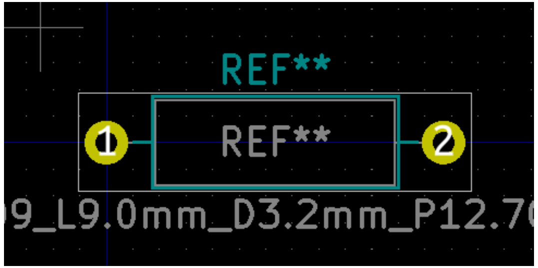
Footprints

- 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)

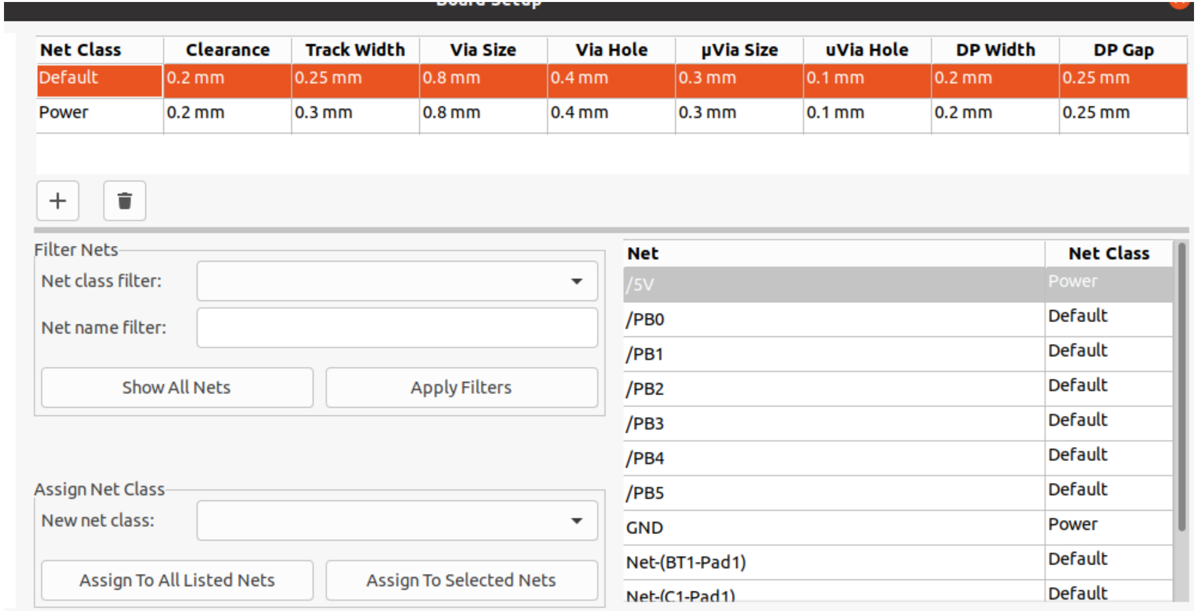
- Basic check on your schematics for errors
- Doesn’t catch everything. Rely on yourself to avoid mistakes.

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.

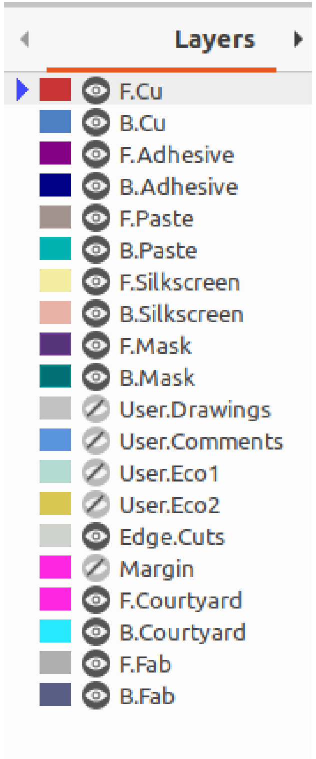
Netclass Editor

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



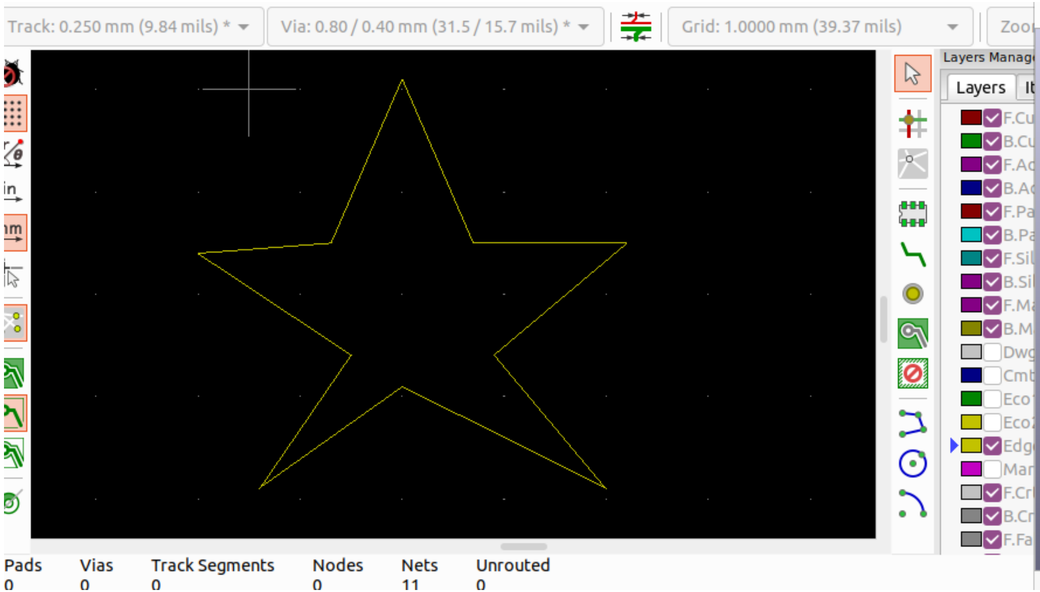
Layers (2-layer PCB)

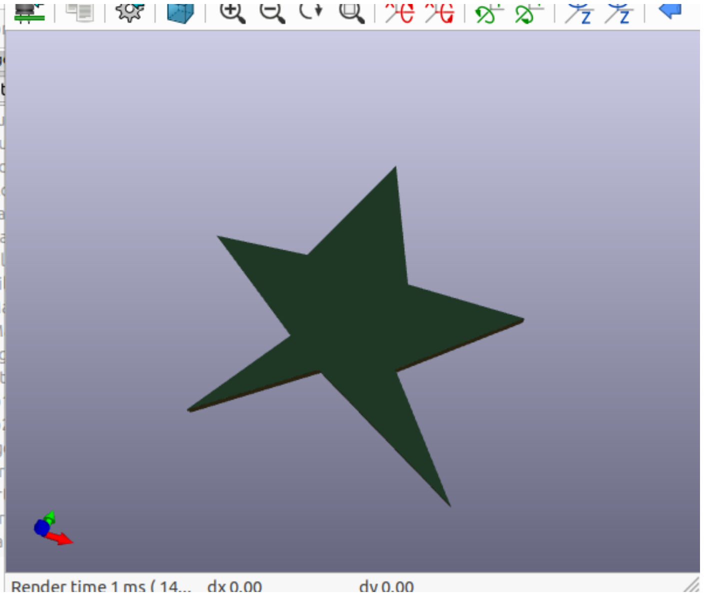
• F.Cu, B.Cu	Copper layers
• F.Paste, B.Paste	Solder paste layers
• F.SilkS, B.SilkS	Silkscreen
• F.Mask, B.Mask	Solder mask
• Edge.Cuts	PCB board outline
• F.CrtYd, B.CrtYd	Courtyard layers
• F.Fab, B.Fab	Fabrication layers



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