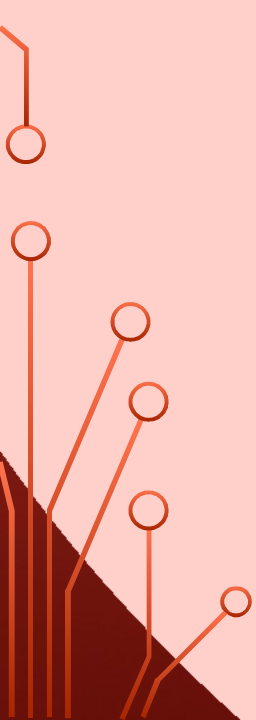# EAGLE PCB Design

Shardul Jade

TOols

- A general EDA tool involves Circuit Design, Simulation, Component Selection, Board Layout, Routing, Gerber Files and 3D CAD.

- Examples of PCB Design:

- Free: KiCAD, EAGLE, DesignSpark, freePCB, expressPCB,

- Paid: EAGLE, PADS, DxDesigner, NI Ultiboard, etc

- There other comprehensive packages which have all the above features. But unfortunately very expensive!

- Reference: http://en.wikipedia.org/wiki/Comparison_of_EDA_software

Some Jargons

- Via fill, or tented vias
- Solder paste
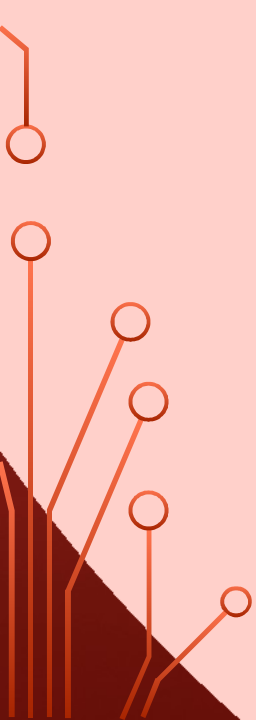- PTH or NPTH
- Soldermask
- Legends

Design Guidelines

- Track width: units mils, mm

- Spacing

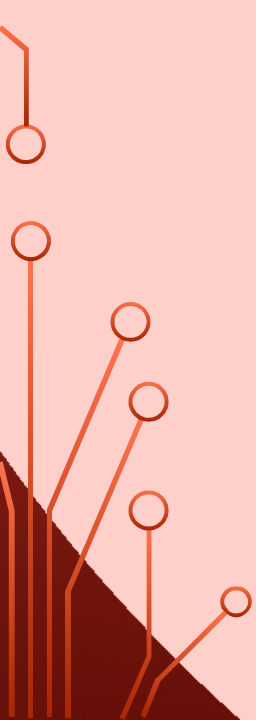- PCB Thickness: 0.8mm/1.6mm/2.4mm

- Reference:
  - Board Guildelines
    http://www.pcbpower.com/content/article-board-specifications.aspx
  - File Format Guidelines:
    http://www.pcbpower.com/content/article-file-format-for-submitting-data.aspx

Demo
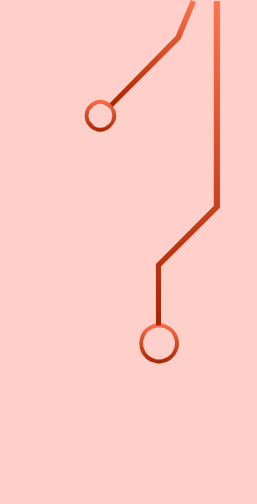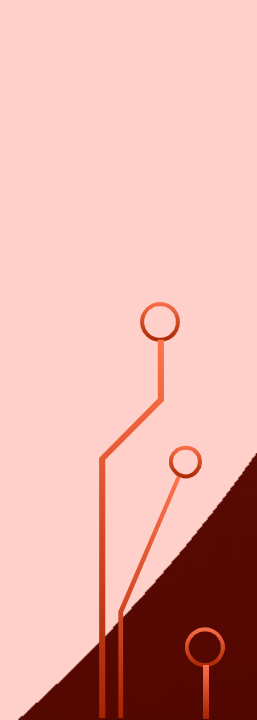
Study Material

- Links to tutorials:
  - https://www.sparkfun.com/tutorials/109
  - https://www.sparkfun.com/tutorials/115
  - http://www.ni.com/white-paper/10492/en/
- External Libraries:
  - Sparkfun
  - Adafruit
  - Element14

General Electronics Tutorials

- Links to tutorials:
  - https://www.sparkfun.com/tutorials/
  - http://doctord.dyndns.org/courses/topics/electronics/alex_pounds/Index.htm
  - http://www.societyofrobots.com/

Thanks you

Shardul Jade

Contact: 9028332193