

A decorative graphic on the left side of the slide, consisting of a network of orange lines and circles of varying sizes, resembling a printed circuit board (PCB) layout. The lines are primarily vertical and horizontal, with some diagonal segments. The circles are placed at various points along these lines, some at the ends and some in the middle. The overall pattern is dense and intricate, extending from the top to the bottom of the slide.

EAGLE PCB Design

Shardul Jade

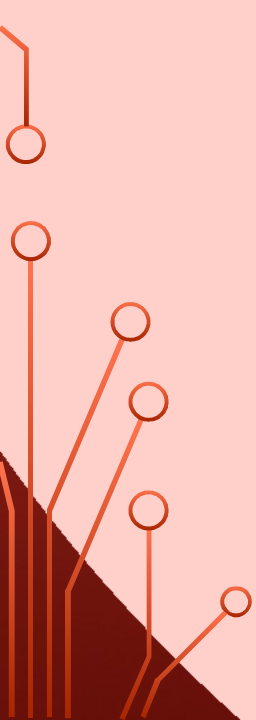


Tools

- 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

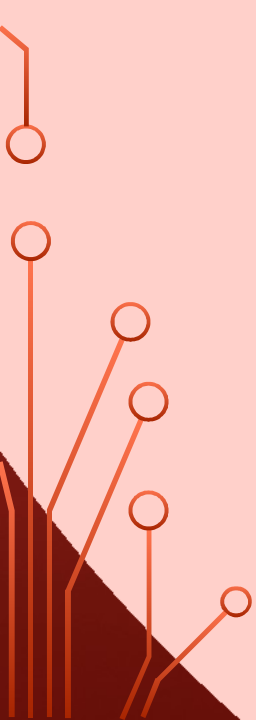
- Track width: units mils, mm
 - Spacing
 - PCB Thickness: 0.8mm/1.6mm/2.4mm
 - Reference:
 - Board Guidelines
<http://www.pcbpower.com/content/article-board-specifications.aspx>
 - File Format Guidelines:
<http://www.pcbpower.com/content/article-file-format-for-submitting-data.aspx>
- 
- 
- 



Demo

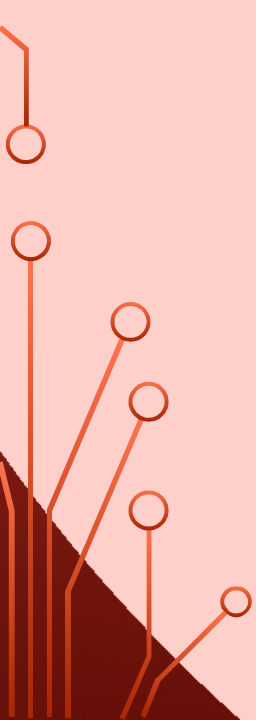
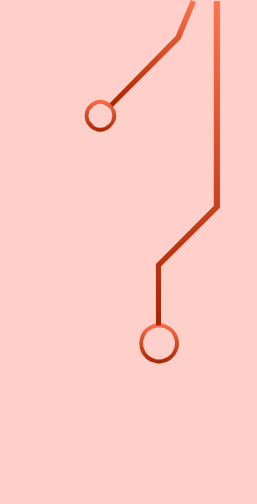
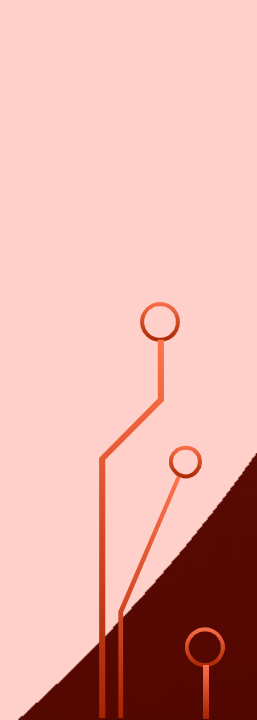


Study Material

- 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

