PCB Design, Part 1

Survey and things to know

Part 1 of 2

- **In this presentation we will look at popular EDA packages for circuit and PCB design**
- **In Part 2 at Build-A-Thon, we will walk through the design and PCB generation process for a project for the September Build-A-Thon**

What to look for in a PCB design tool

- Ease of use
- Output (Gerbers) compatibility with popular vendors
- Customization and import capabilities; for example, footprints
- PCB Shape and Silk Screen flexibility
- 2 to (optionally) 4-layer capabilities
- Wide user community support

What are the leading packages?

- KiCAD (CERN)
- ExpressPCBPlus, was Copper Connection
- Eagle (Autodesk)
- PADS (Mentor Graphics)
- CircuitMaker* (Altium)

KiCAD

- Probably the most popular free PCB design tool
- Open source, distributed by CERN, covers virtually all OS's (PC, Mac, Linux flavors)
- Combines some of the best features of Eagle and PADS
- A set of applications working together; the main ones are Eeschema (Eagle) and Pcbnew (PADS), but also has a SPICE simulator
- Has advanced features like Microwave/RF traces, that are usually only available in high $$ Mentor/Altium products

KiCAD

- KiCAD is one of the packages that has community and component vendor (Digikey/Mouser) support for footprint libraries
- Does not require you to start with a schematic, but rewards you if you do because net names and annotations will carry over to the PCB silk screen
- New footprints can easily be added/created, but you should *rarely* need to

ExpressPCBPlus

- Copper Connection, now *ExpressPCB Plus*, is what I have used until recently
- CC supports 4 layers and top and bottom silk screening
- Has a wide array of common component footprints
- New footprints can easily be added/created
- CC generates Gerber files that are compatible with all major facilities
- Faster to learn and use than KiCAD, but not as powerful

Workflow

- Create your schematic design
- Annotate components/nets
- Simulate if desired
- Generate PCB (you usually have the option to disable auto-routing)
- Load Gerber files onto Vendor PCB site and review the layout

Practices (more in Part 2)

- Work in metric
- Footprint additions to your library
  - Mainly these are in the back of component spec sheets
  - Most are specified in mm, but some are in both systems
  - Requires some addition/subtraction in your coordinate system to get spacing correct

PCB Build Vendors and compatibility

- **OSHPark**
  - **Best tools and very easy to use, $$**
- **JLCPCB**
  - **Good tools, easy to use, $**
- **PCBWAY**

PCB Vendor manufacturing options

- PCB Colors
- PCB thickness, usually 1.6 mm for 2 layer,
- Pad finish
  - ENIG - Electroless Nickel Immersion (OSHPark default)
  - HASL – Hot Air Solder Leveling, Lead or Lead-free solder
- Silk screen details

Which PCB Vendor?

- KiCAD comes with >10 completed examples, some of which are excellent test cases to understand a supplier's capabilities and online tools.
- Zip-up your Gerber/drill files, and use this to understand how easy a PCB vendor is to work with.
  - Pricing
  - Display options before you order
  - Solder Mask colors, pad finishing options, etc.

Self Study Resources

- Udemy – www.udemy.com (KiCAD like a Pro, 2ed)
- KiCAD website at http://kicad-pcb.org/