



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**
- 

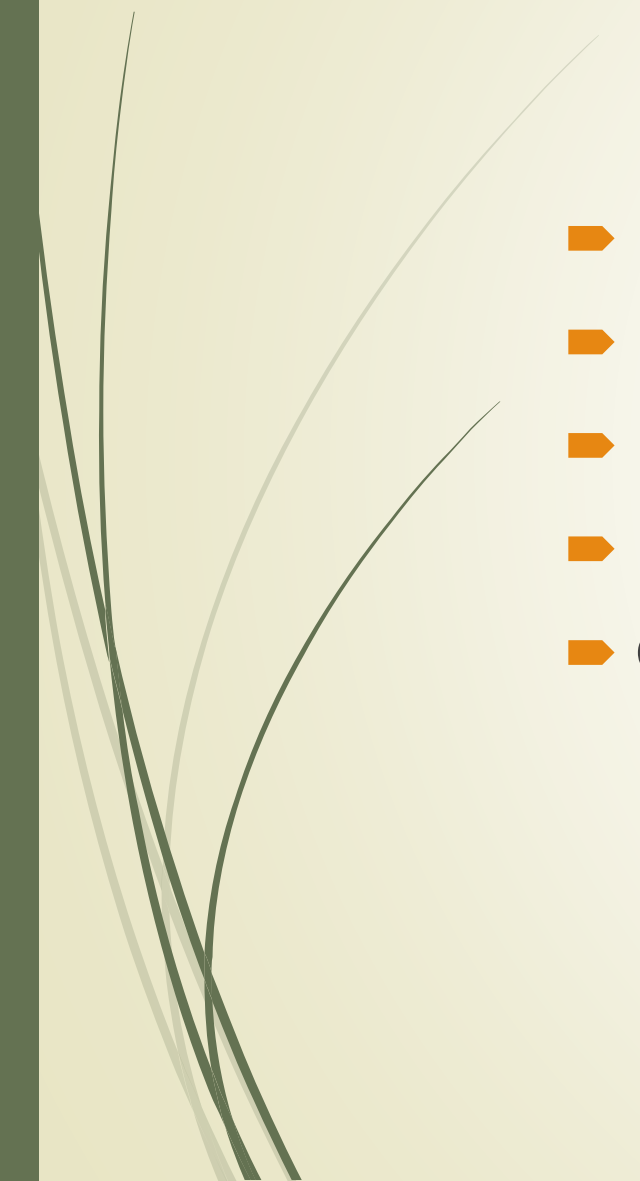


What to look for in a PCB design tool

- ▶ **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, \$\$

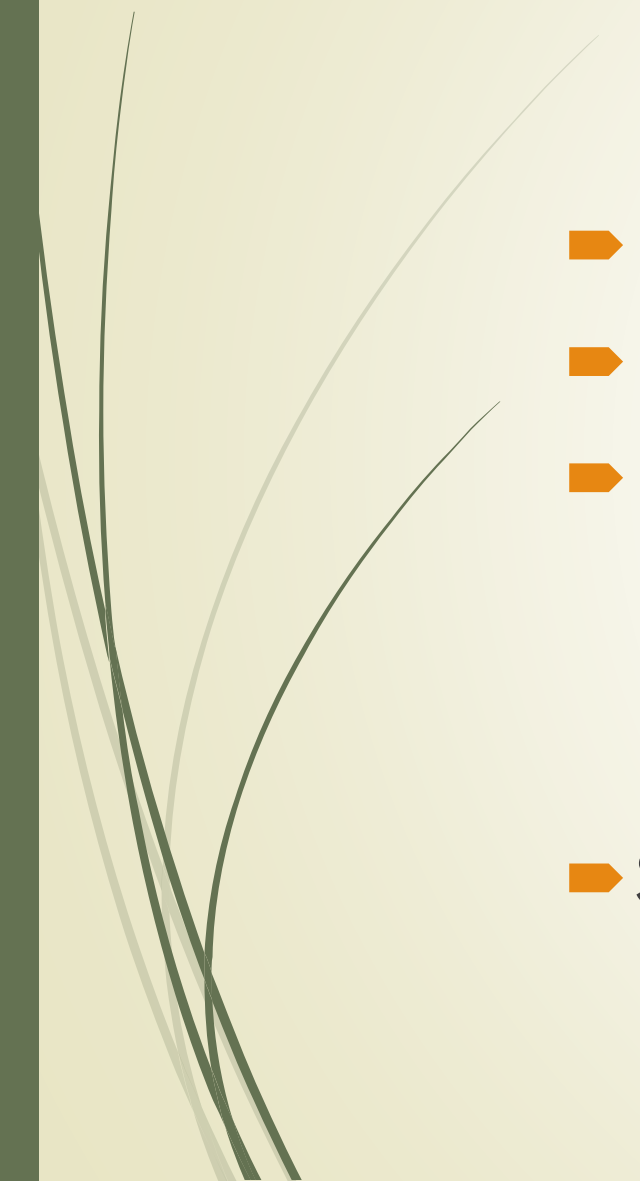
- ▶ **JLCPCB**

- ▶ 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/>