Basic PCB Design using KiCad

USU SPAC spring 2020

Nikolas Clark

Lecture Outline

- Why KiCad? What other software is used in industry?
- PCB Layers and descriptions
- Traces and Vias
- Schematic Design
  - Adding components
  - Creating custom components?
  - Importing components into custom library?
  - Connecting with wires
  - Connecting with labels
  - Associating footprints with Symbols
  - Exporting schematic to a netlist
  - Electrical DRC
- PCB design
  - Importing from a netlist
  - Basic elements of a good design
  - Defining board outline with edge cuts
  - Adding traces/vias
  - Adding planes
  - Design Rules Setup/Check
  - Exporting Gerbers/Feducials
- Ordering a board from Oshpark, or another service