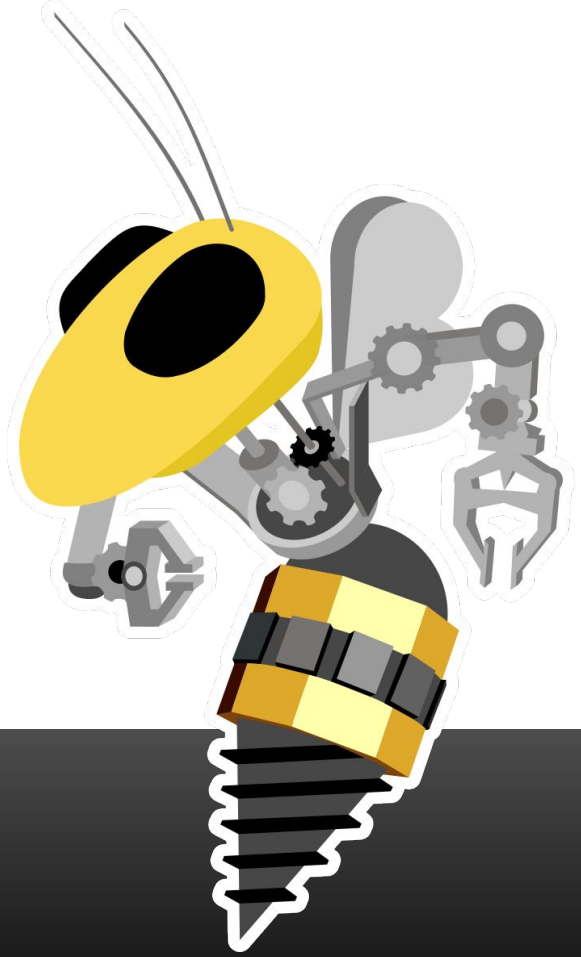
Welcome!

Electrical Training
Week 2



RoboJackets

Competitive Robotics at Georgia Tech

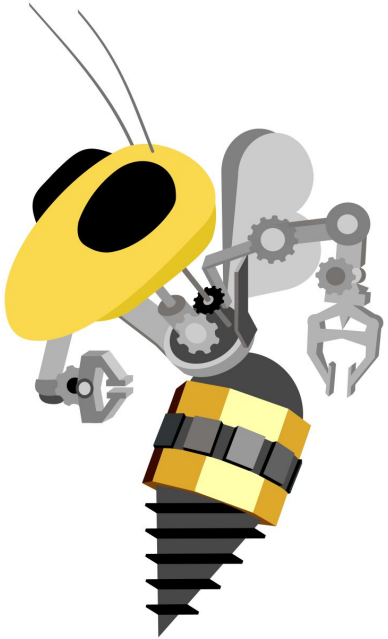
www.robojackets.org

Last Week!

- What are Microcontrollers?
- Intro to C++
- Prototyping

This Week!

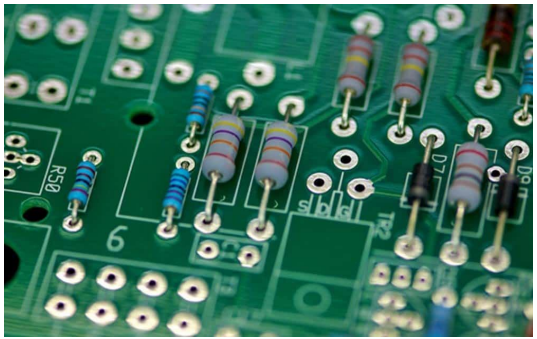
- Introduction to PCBs
- Introduction to KiCAD
- Parts and Libraries in KiCAD
- Configuring KiCAD Setup
- Making a Part in KiCAD



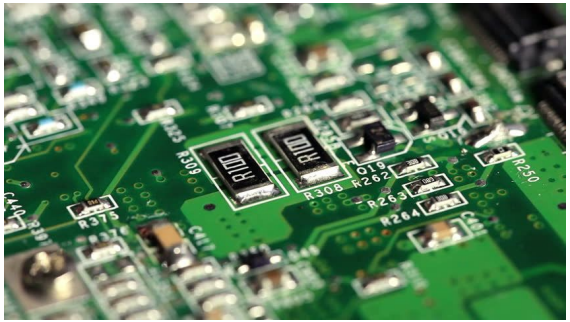
What are PCBs?

Printed Circuit Boards (PCBs)

- A way to construct more electrically complex circuits that are impractical for a breadboard
- Have a wide range of components (sensors, MCUs, power circuit components) that are often surface mount (SMD) rather than through hole



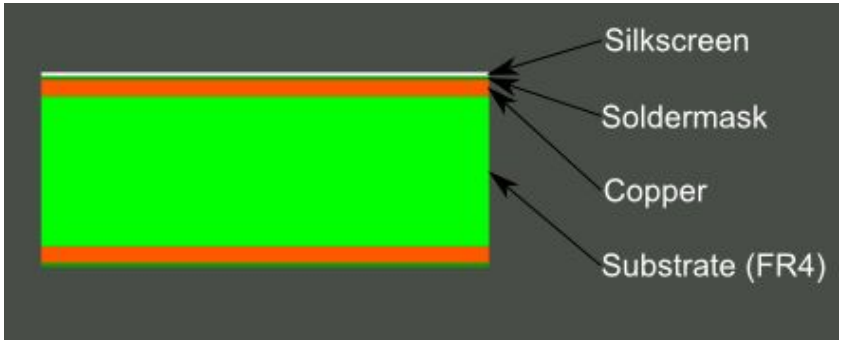
Through Hole



Surface Mount

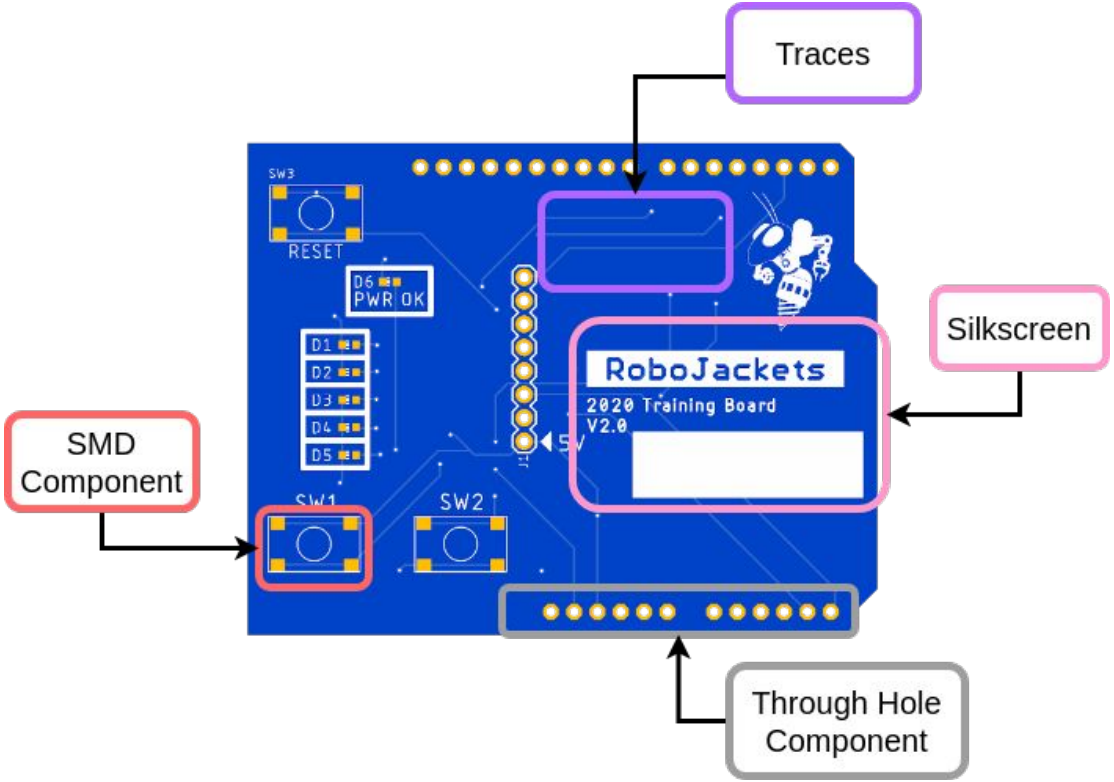
Layers

- Work by having multiple different layers
- Silkscreen - Text, labels, and graphics
- Soldermask - repels solder, usually colored green
- Copper - conductive materials
- Substrate - for structure



PCB Features

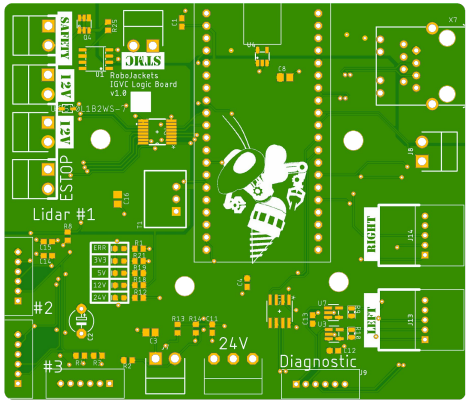
Training Board

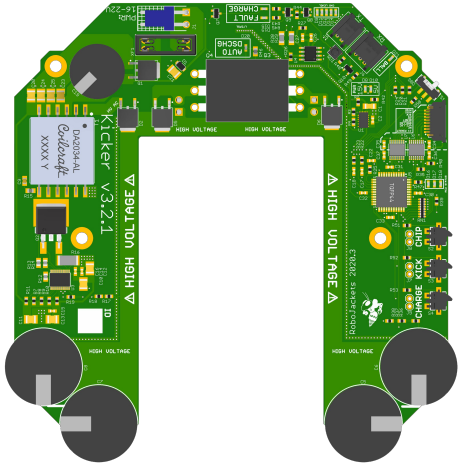


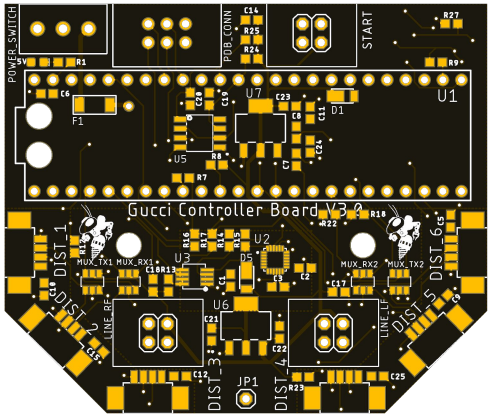
ROBOJACKETS
COMPETITIVE ROBOTICS AT GEORGIA TECH

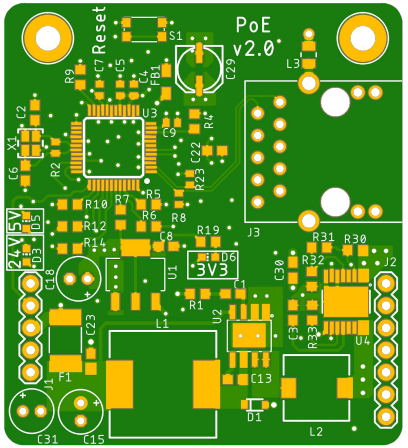
Team Examples

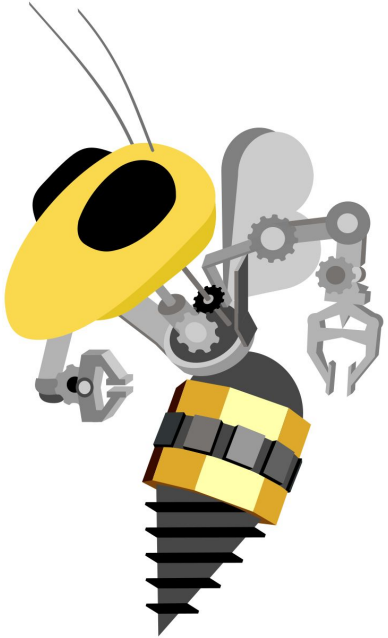
We use PCBs for a wide range of problems









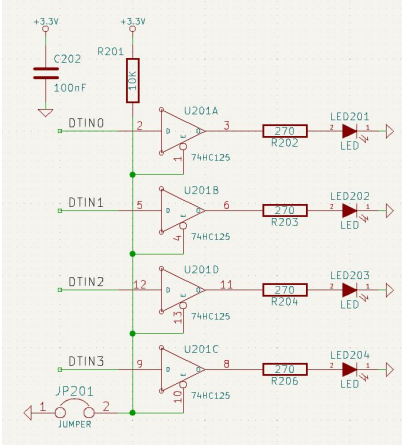


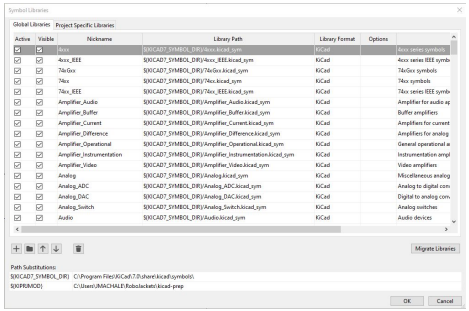
What is KiCAD?

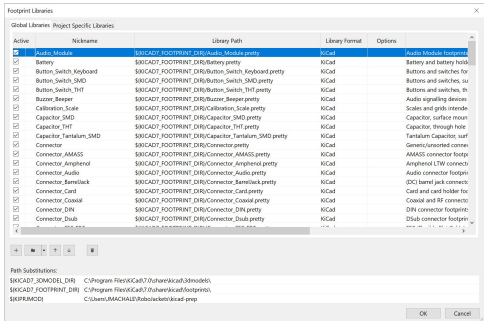
Where the magic happens

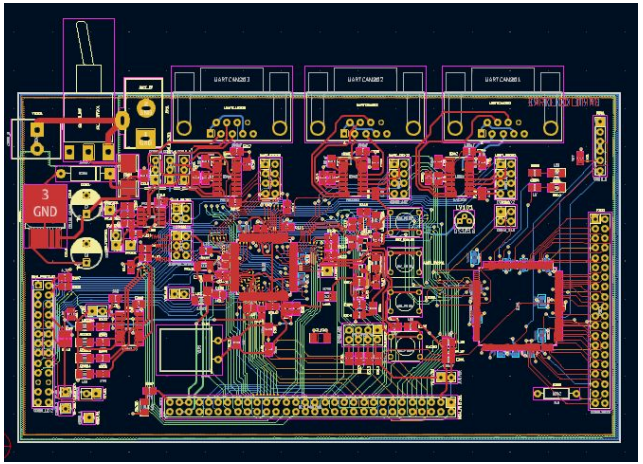
KiCAD

- Computer software to design PCBs
- Three Stages of Development
  - Libraries & Parts
  - Schematics
  - Board Layout

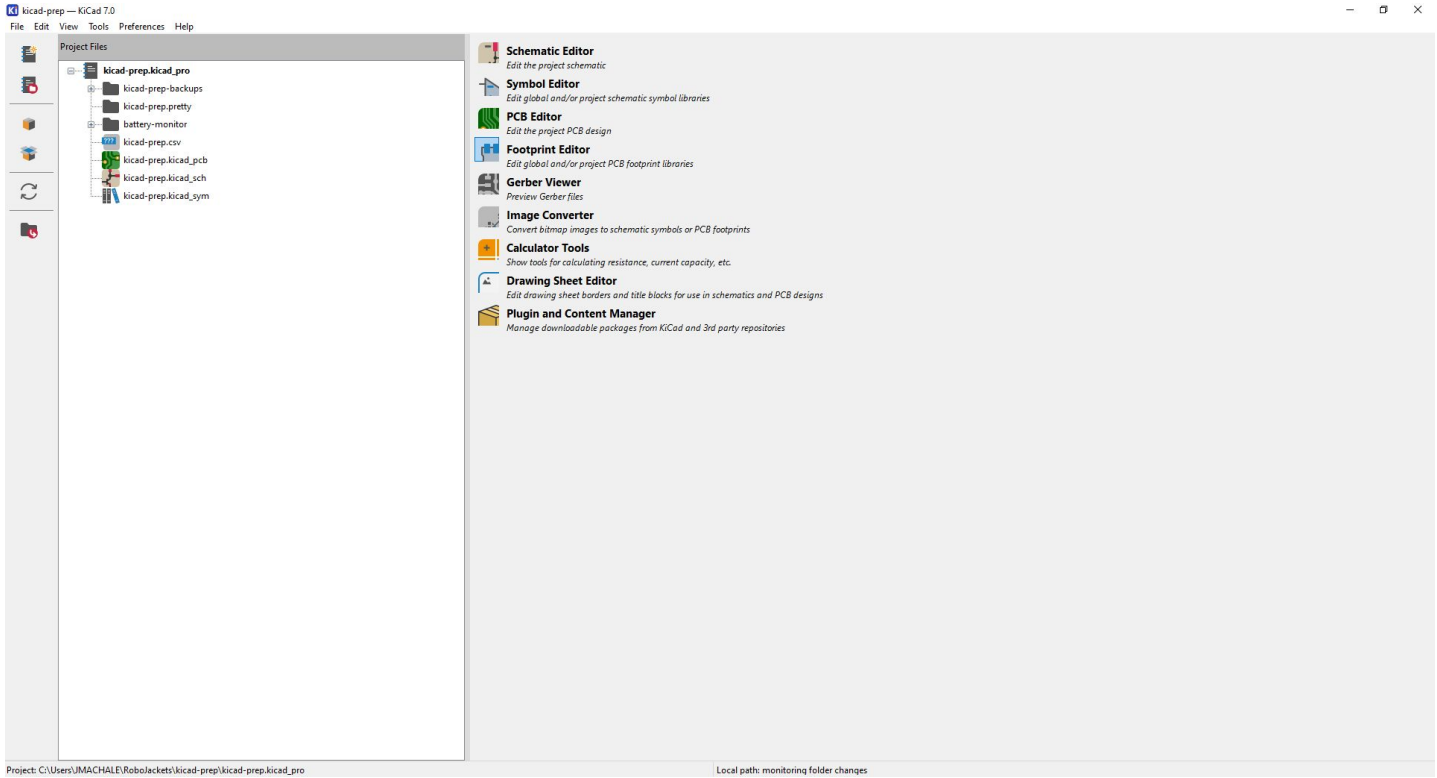






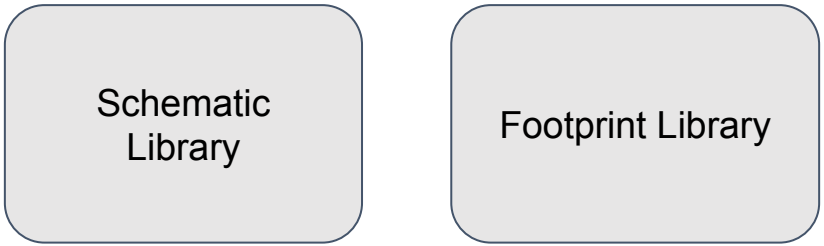


Project Manager



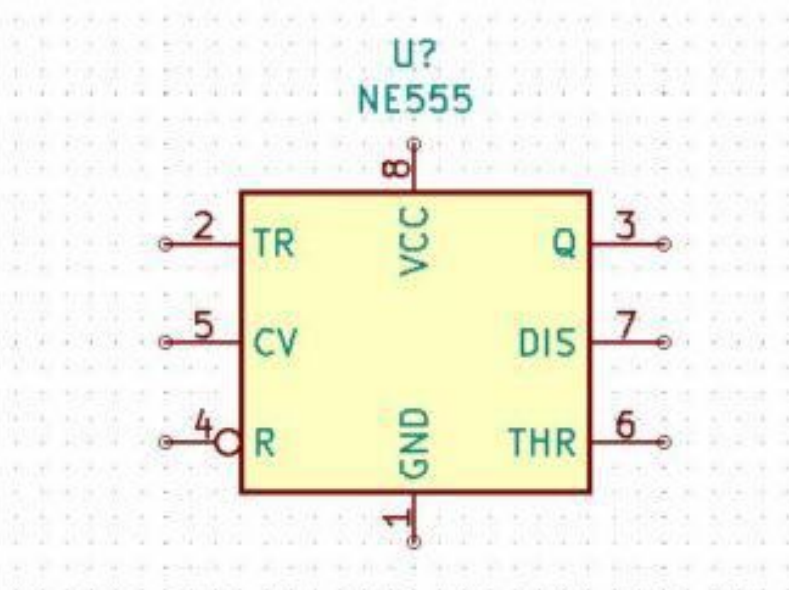
Libraries

- Store various components used in projects
- Schematic and footprint libraries are separate for electrical components in KiCAD



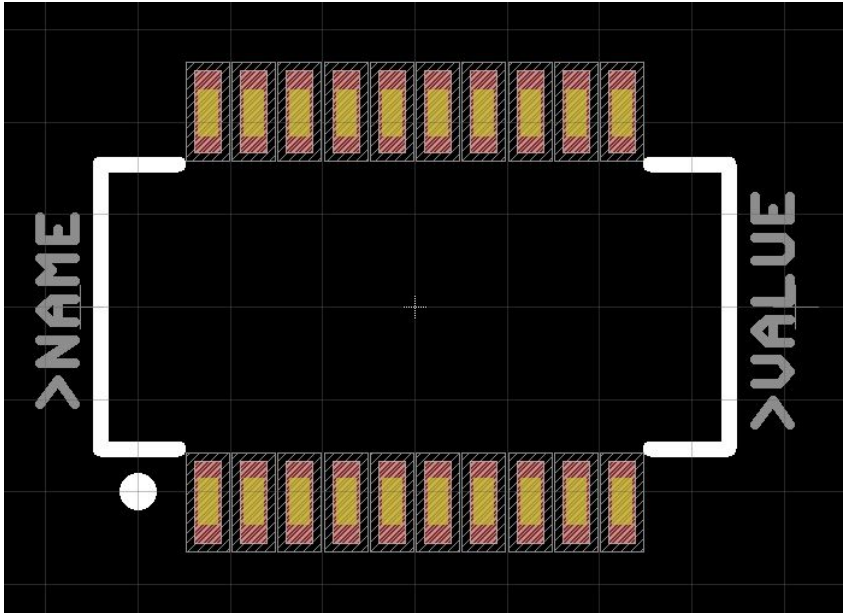
Symbol

- Used in the schematic view to make circuit connections with other component symbols



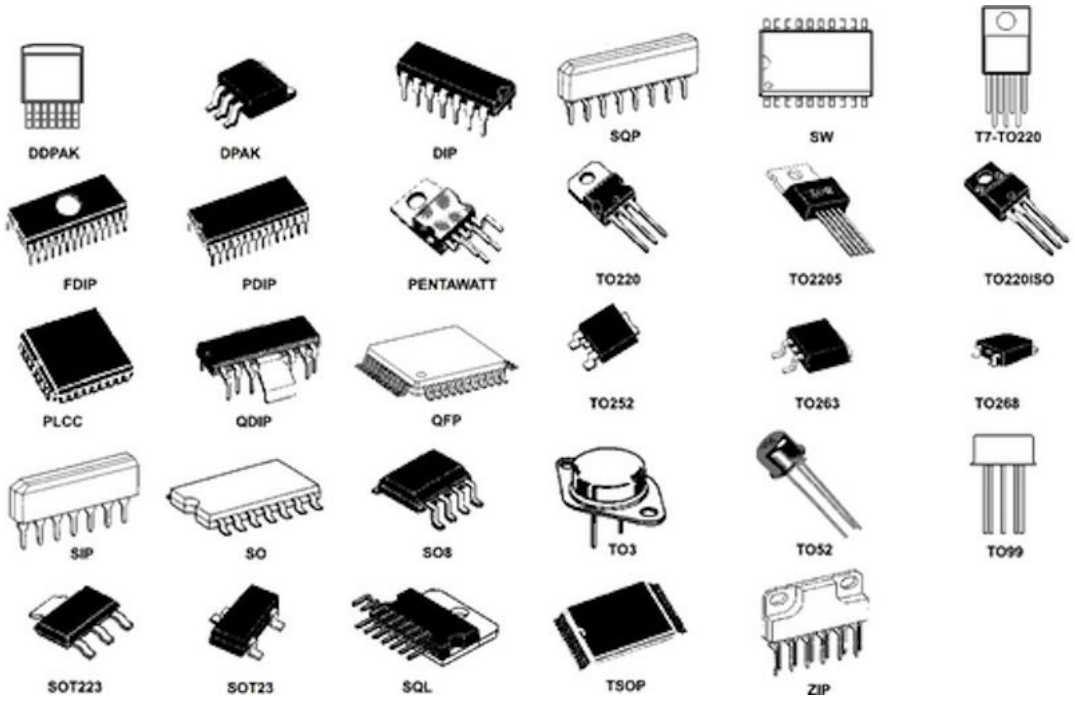
Footprint

- Used in the the board layout view to make the actual physical connections between components



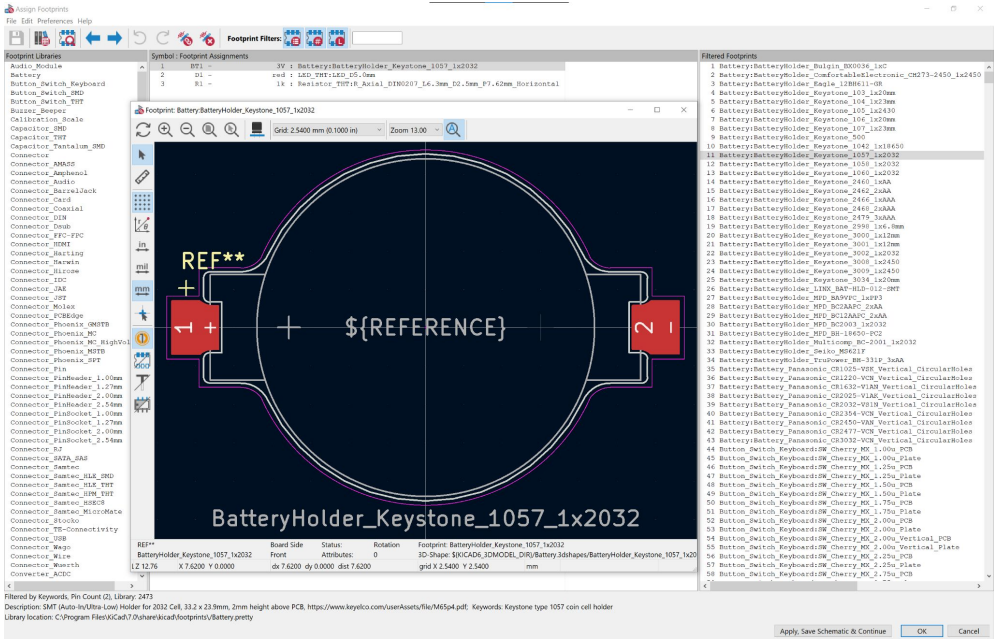
Common Packages

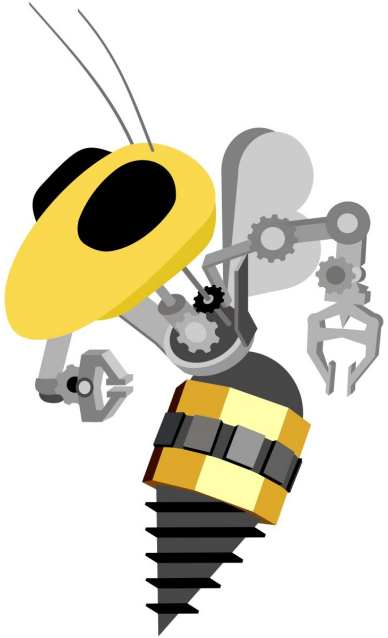
- Many parts utilize industry standardized footprints so many different parts can have the same footprint



Choosing a footprint for a symbol

- You can choose any footprint for a symbol since some components come in multiple different packages (common with ICs)





Lab!

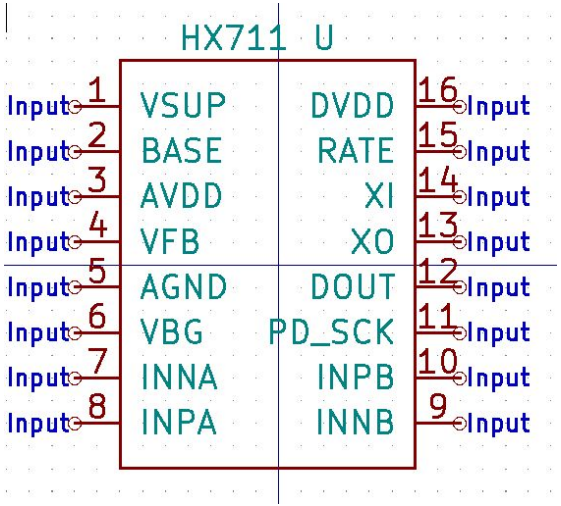
Setup + Making a Part in KiCAD

Installing Software

- KiCAD
  - Download here
    - https://www.kicad.org/download/

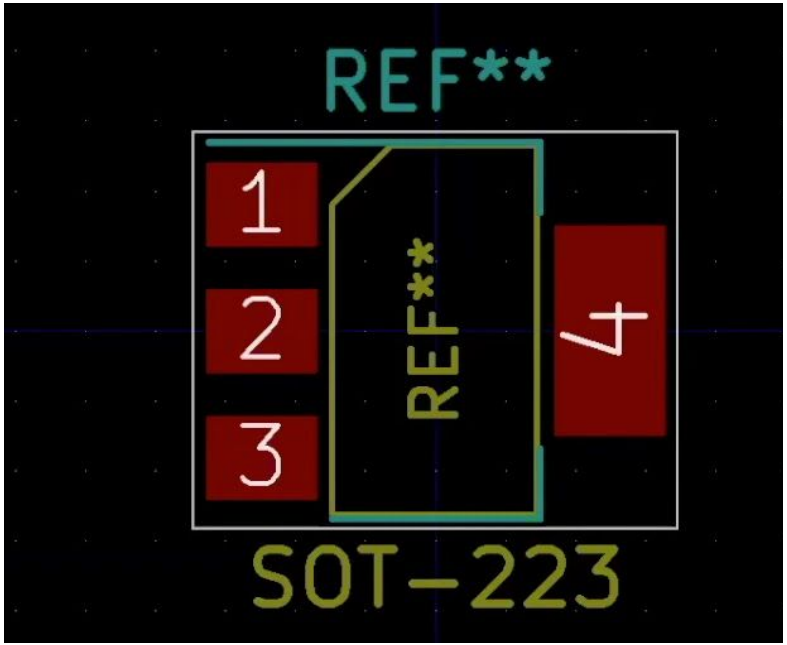
Making a Part - Symbol

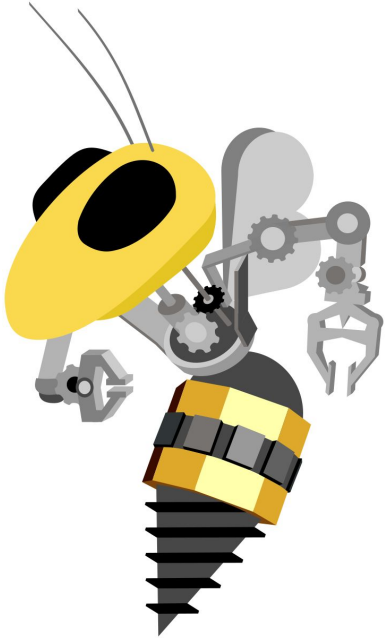
- Based on pin configuration in the datasheet, create similarly named pins and create an outline



Making a Part - Footprint

- Create a physical representation of the the part by reading the package outline details in the datasheet and then placing correctly sized pads/drills at the appropriate location





Thank You!

Any Questions?