# Welcome!

Electrical Training

Week 2

# 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

- Adding parts to KiCAD
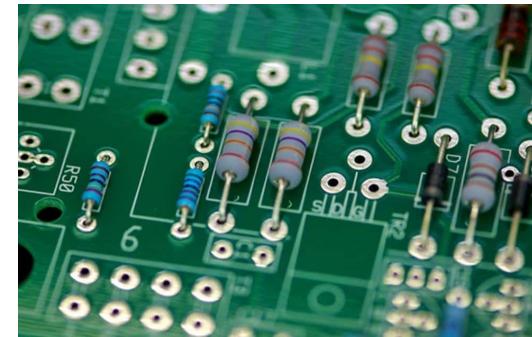
TLDR, we are going to import our parts into KiCAD from SnapEDA.
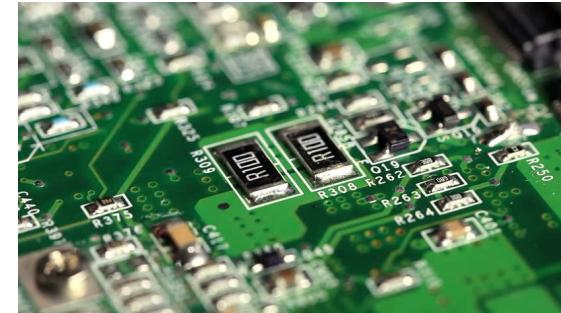
# What are PCBs?

# 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
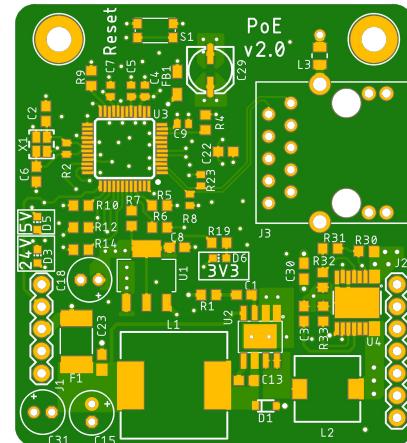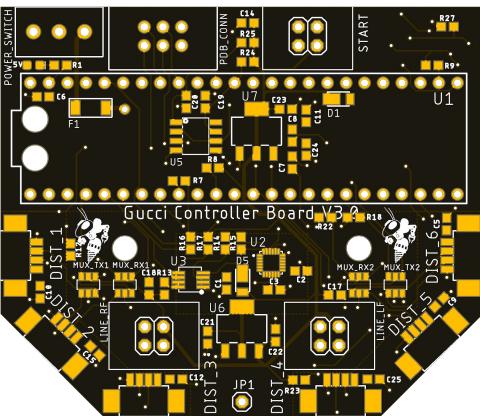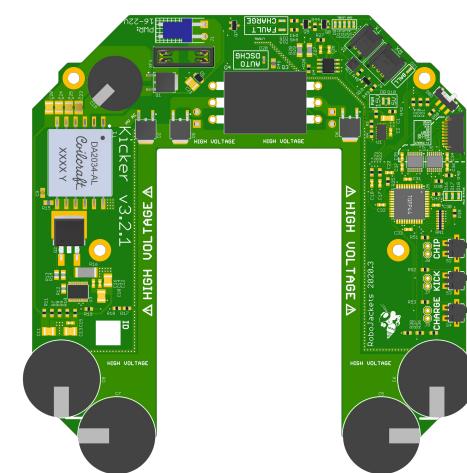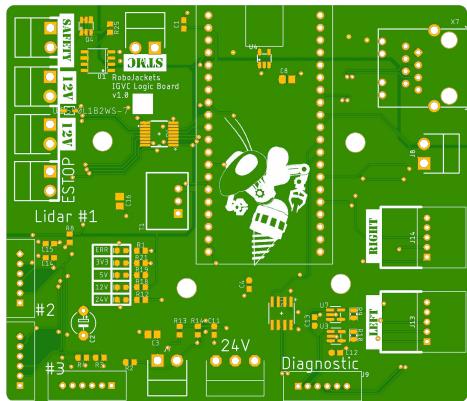


Through Hole



Surface Mount

# RoboJackets
## Competitive Robotics at Georgia Tech
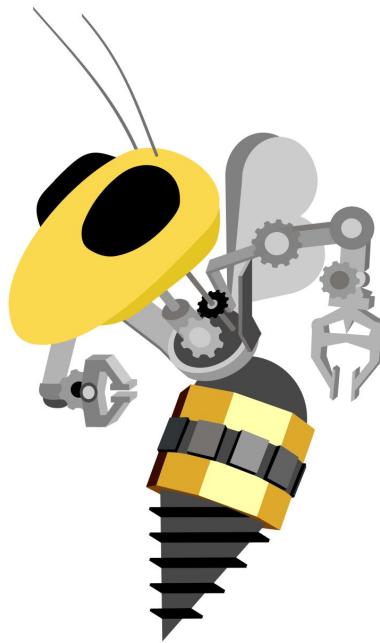
# Team Examples

*We use PCBs for a wide range of problems*
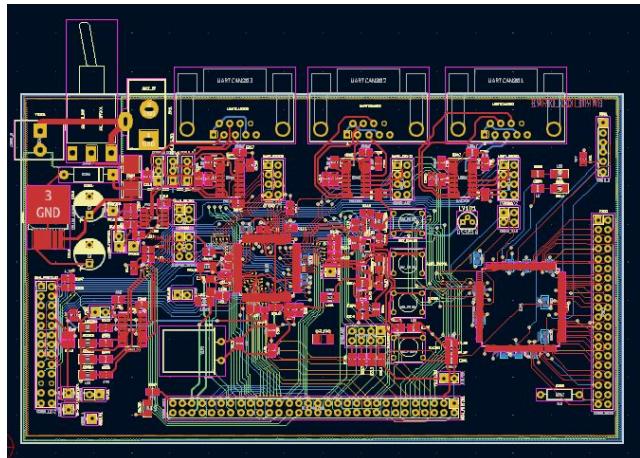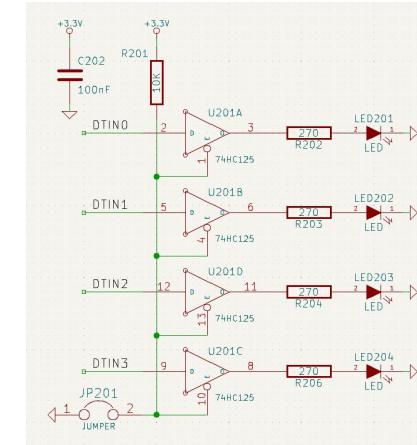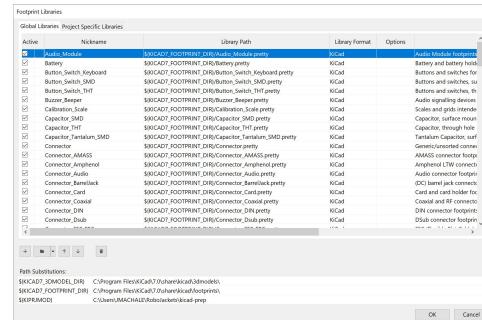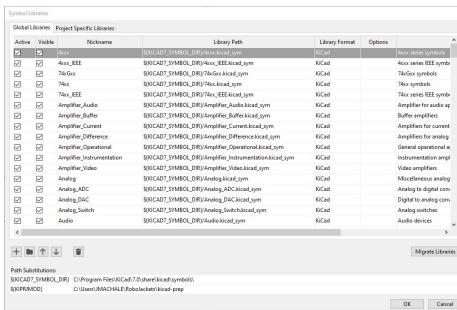
# What is KiCAD?
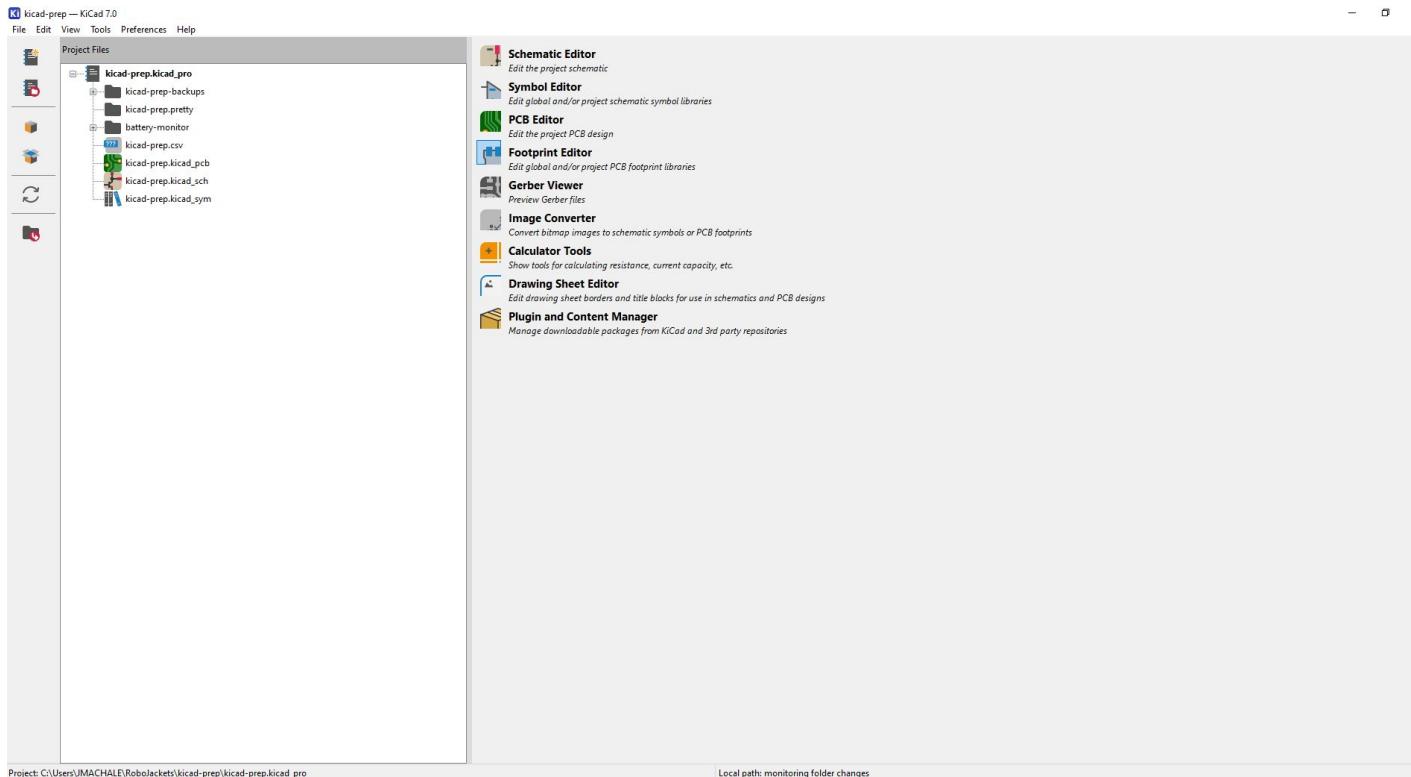
Where the suffering begins

# KiCAD

- Computer software to design PCBs
- Three Stages of Development
    - Libraries & Parts (focus on this today!)
    - Schematics
    - Board Layout

# Project Manager

# Libraries
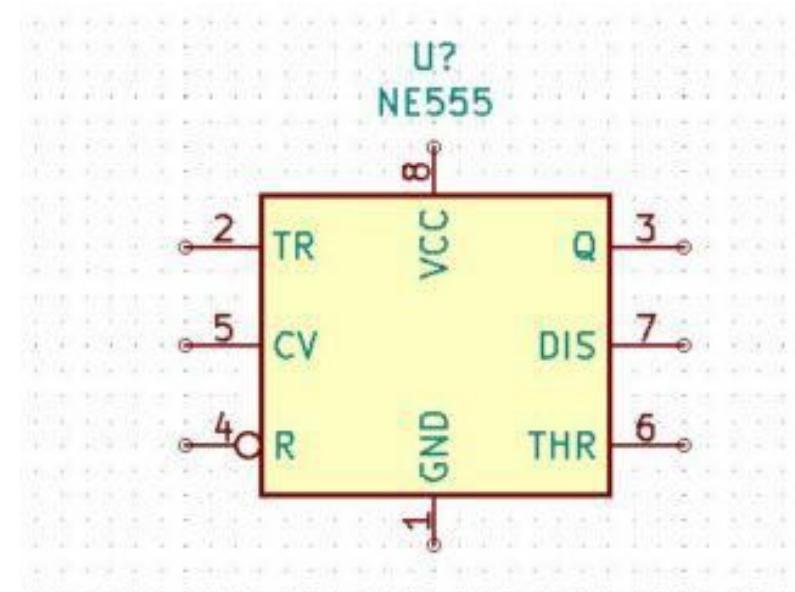
- Store various components used in projects
- Schematic and footprint libraries are separate for electrical components in KiCAD
- We will focus on the schematic library today

Schematic Library

Footprint Library
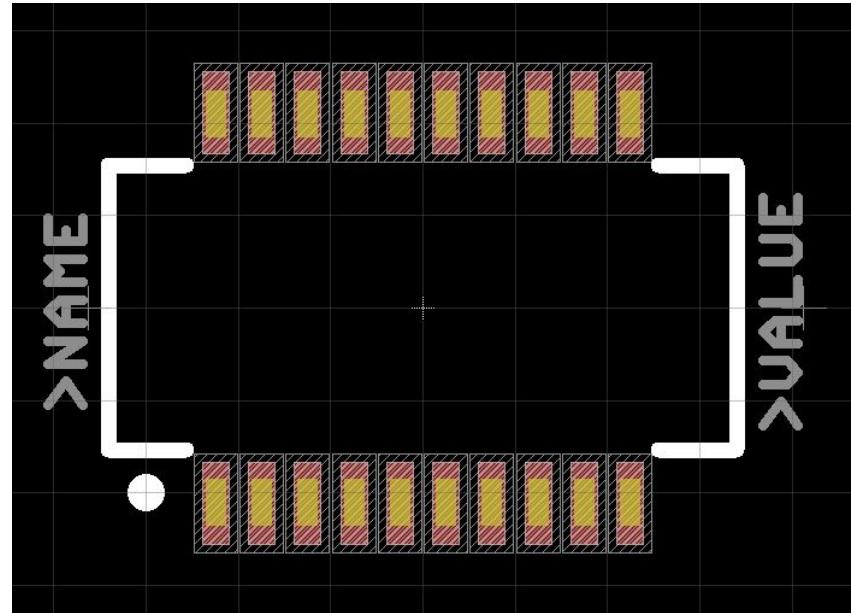
# Symbol
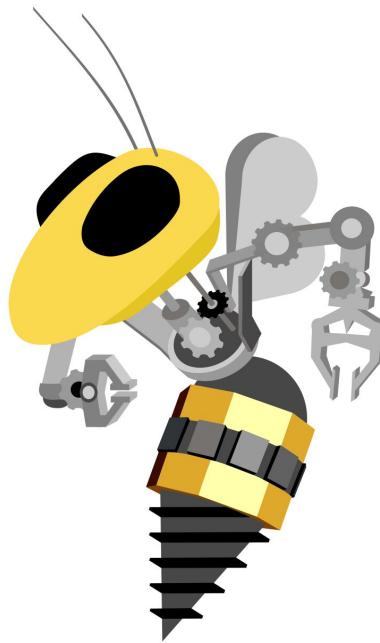
- Used in the schematic view to make circuit connections with other component symbols
- "Symbolizes" the connections made on the PCB

# Footprint

- Used in the the board layout view to make the actual physical connections between components
- The component will actually somewhat look like the footprint

# Lab!

KiCad Setup + Adding Parts to Schematic/Footprint Library

# Installing Software

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

# Downloading Parts from SnapEDA

- You could make your own parts, but why do that when someone else has already done it?
1. Go to snapeda.com (make an account if you haven't done so)
2. Search up your part
3. Download the symbol and footprint and click the KiCad option
4. Inside the zip folder, there is a tutorial on how to import the symbol, follow the instructions for KiCad (or click on this link)
- You will have to import the symbol and footprint separately
- Note: For footprint, you select the folder, not an actual file
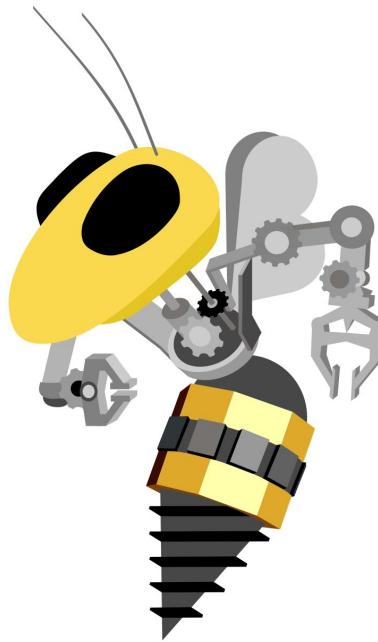
# Live Demonstration

- watch me embarrass myself

# What if I can't find my part?

- There are other symbol/footprint websites out there, so look around the internet!
- If you genuinely cannot find it, you can make your own inside KiCAD
- It's a little advanced and somewhat niche, so I will not be going over it, but here is a [link](#) if you're interested

# Thank You!

Any Questions?

# For next time…

KiCAD Schematic! Be sure to bring:

- Your laptop

- Mouse (highly recommended)

- A Coke for your instructor (me)

Location:

- Electrical: Skiles 169

- Firmware: Van Leer C457

# Feedback/Attendance