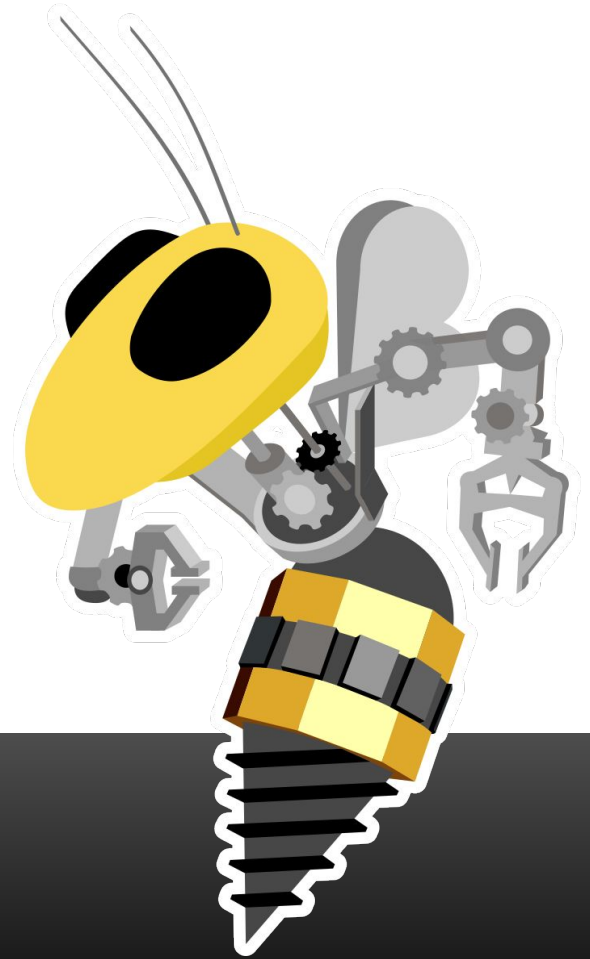


Welcome!

Electrical Training
Week 2

ROBOJACKETS
COMPETITIVE ROBOTICS AT GEORGIA TECH

www.robojackets.org



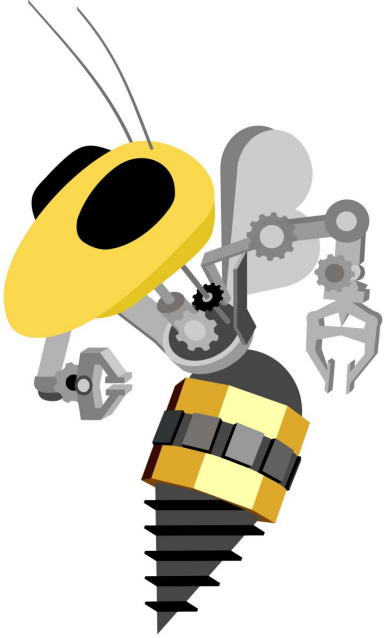
Last Week!

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

This Week!

- Introduction to PCBs
- Introduction to KiCAD
- Parts and Libraries in KiCAD
- Configuring KiCAD Setup
- Adding parts to KiCAD

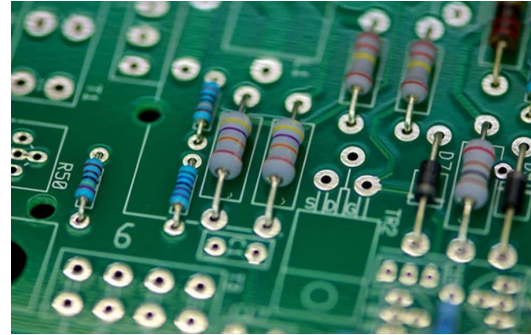
TLDR, we are going to import our parts into KiCAD from SnapEDA.



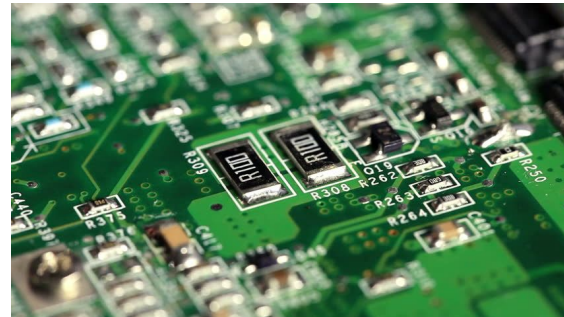
What are PCBs?

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



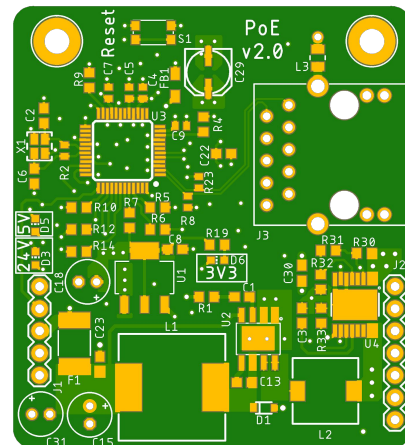
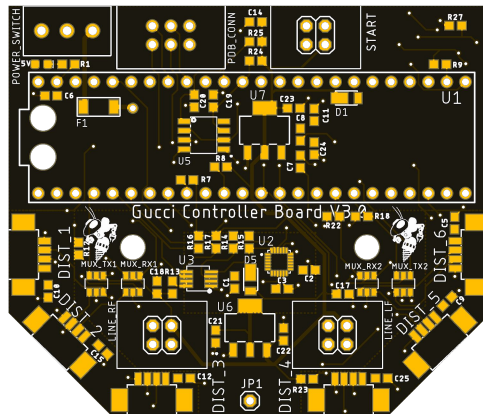
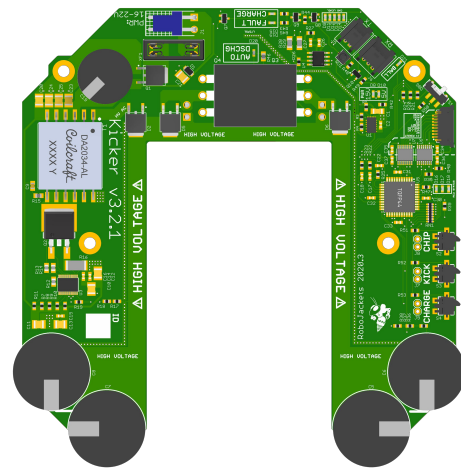
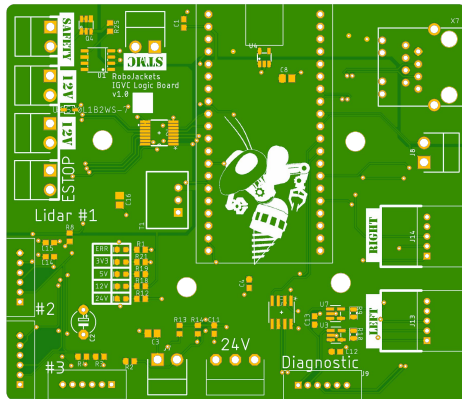
Through Hole

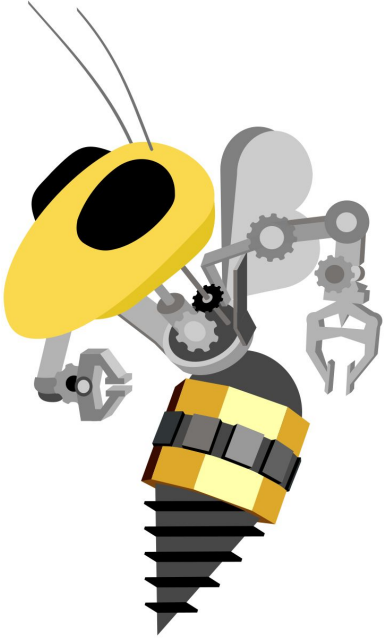


Surface Mount

Team Examples

*We use PCBs
for a wide
range of
problems*

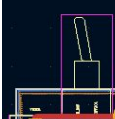


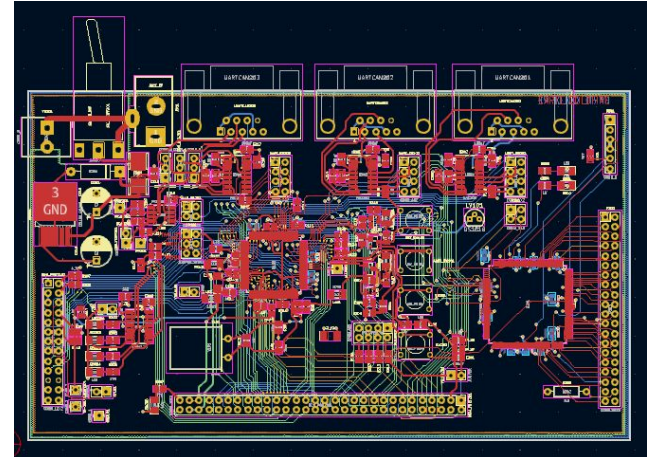
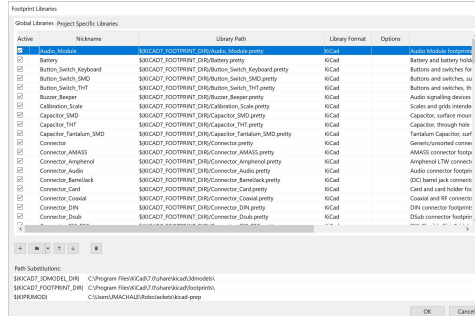
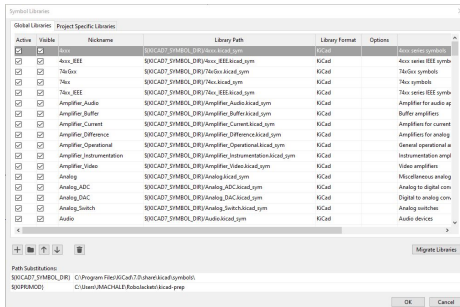
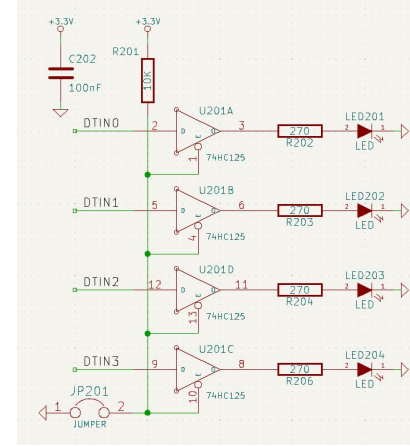


What is KiCAD?

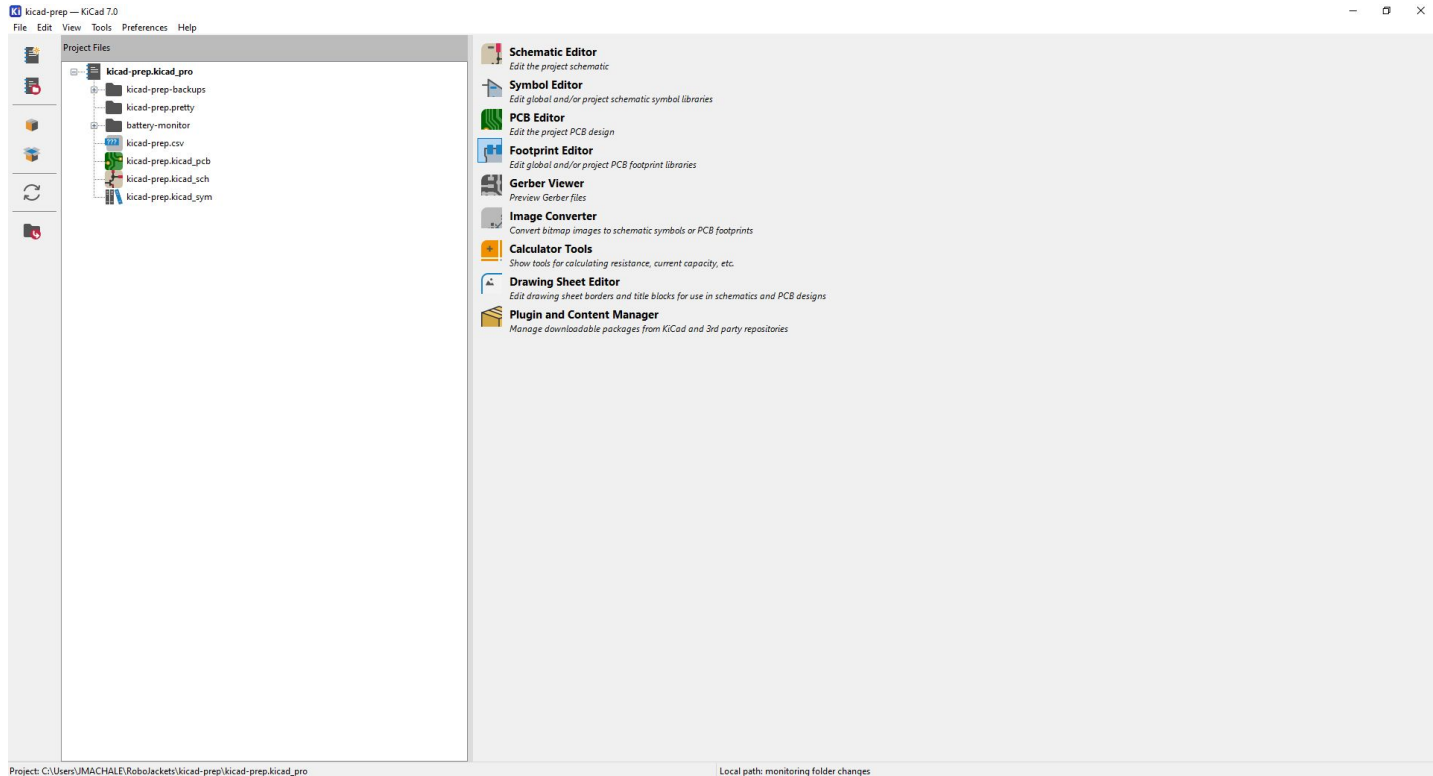
Where the suffering begins

KiCAD

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

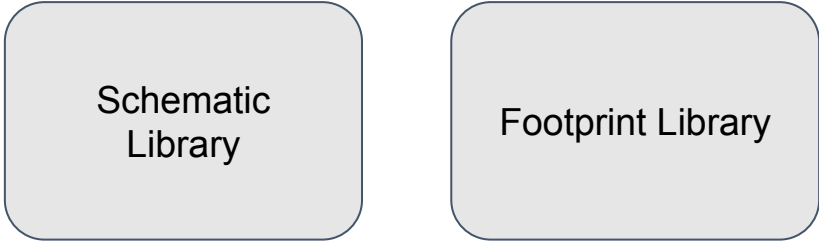


Project Manager



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

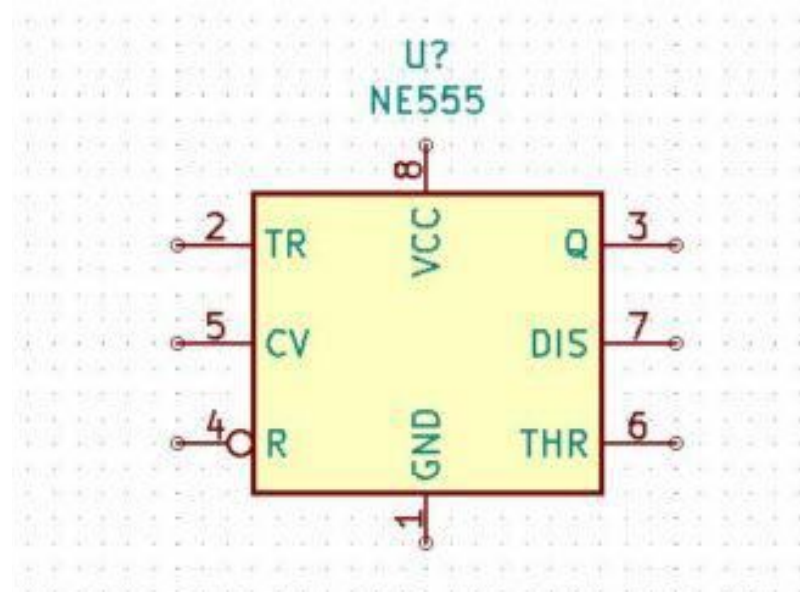


Schematic
Library

Footprint Library

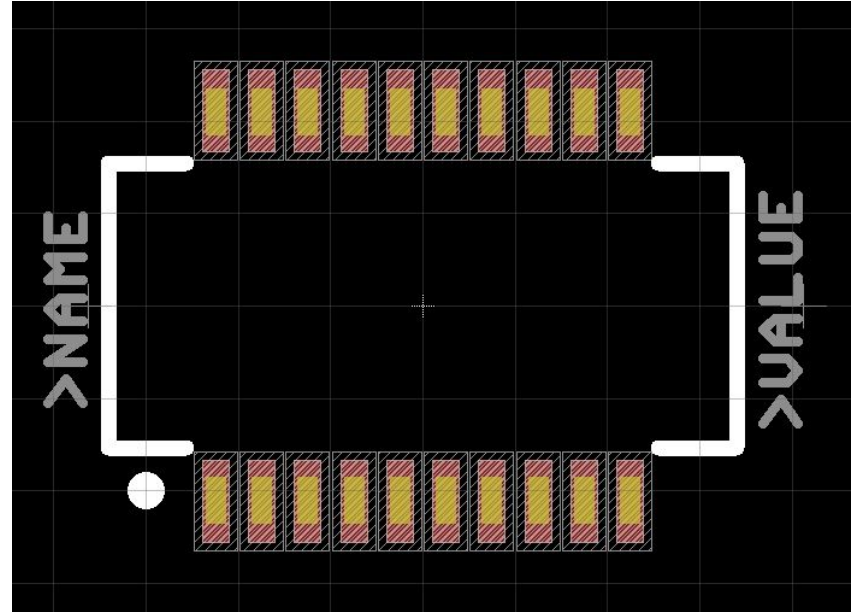
Symbol

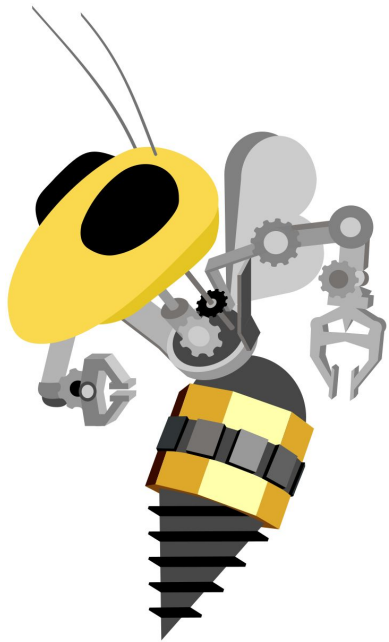
- Used in the schematic view to make circuit connections with other component symbols
- “Symbolizes” the connections made on the PCB



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

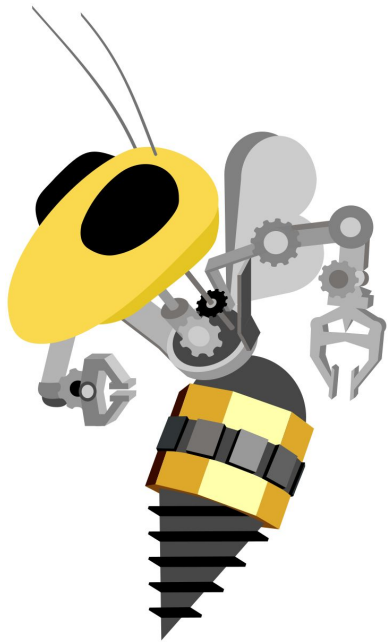
Live Demonstration

- 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 Form

