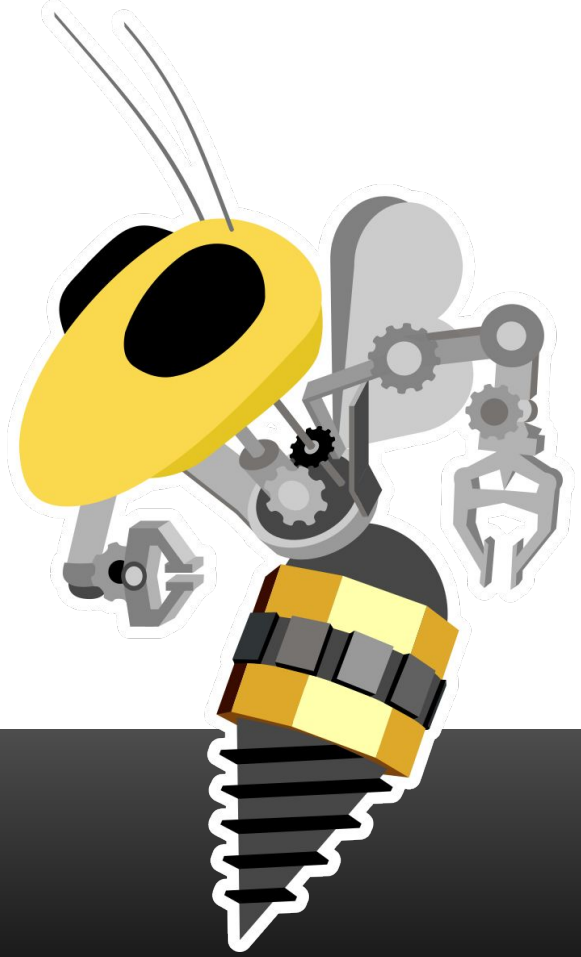
Welcome!

Electrical Training
Week 2



RoboJackets

Competitive Robotics at Georgia Tech

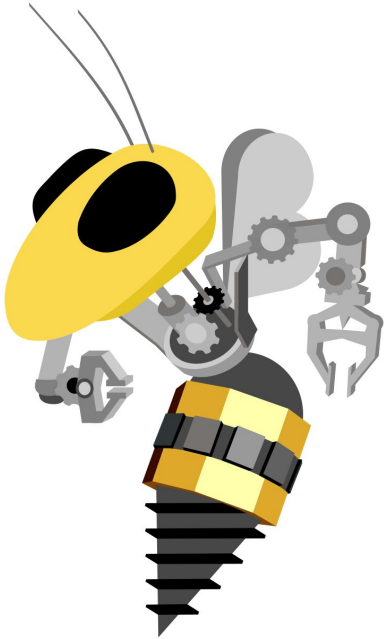
www.robojackets.org

Last Week!

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

This Week!

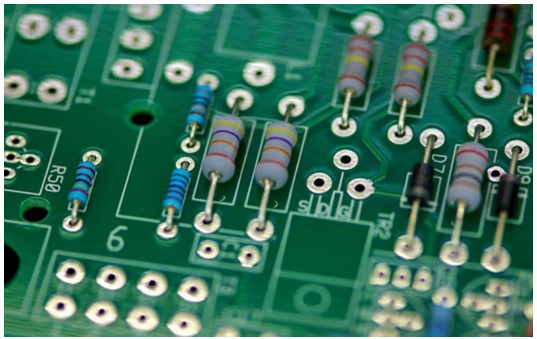
- Introduction to PCBs
- Introduction to EAGLE CAD
- Parts and Libraries in EAGLE
- Configuring EAGLE Setup
- Making a Part in EAGLE



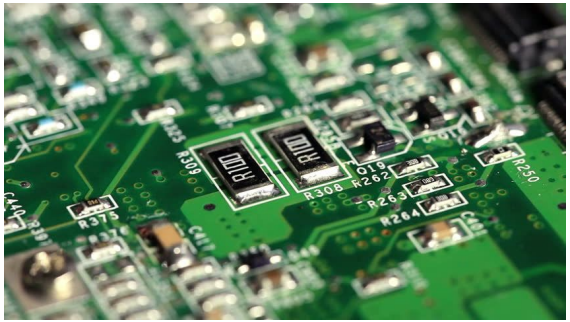
What are PCBs?

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



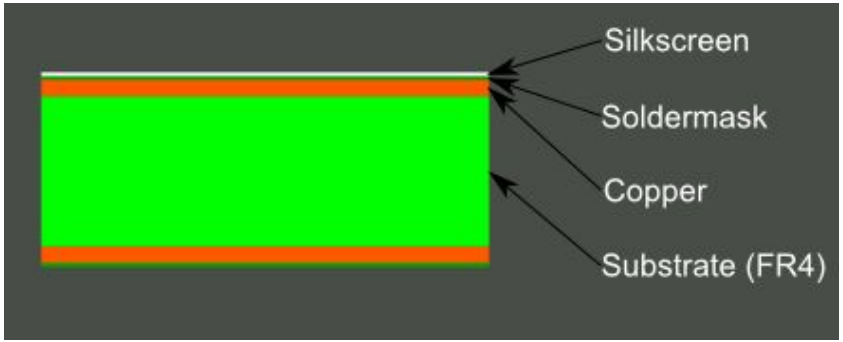
Through Hole



Surface Mount

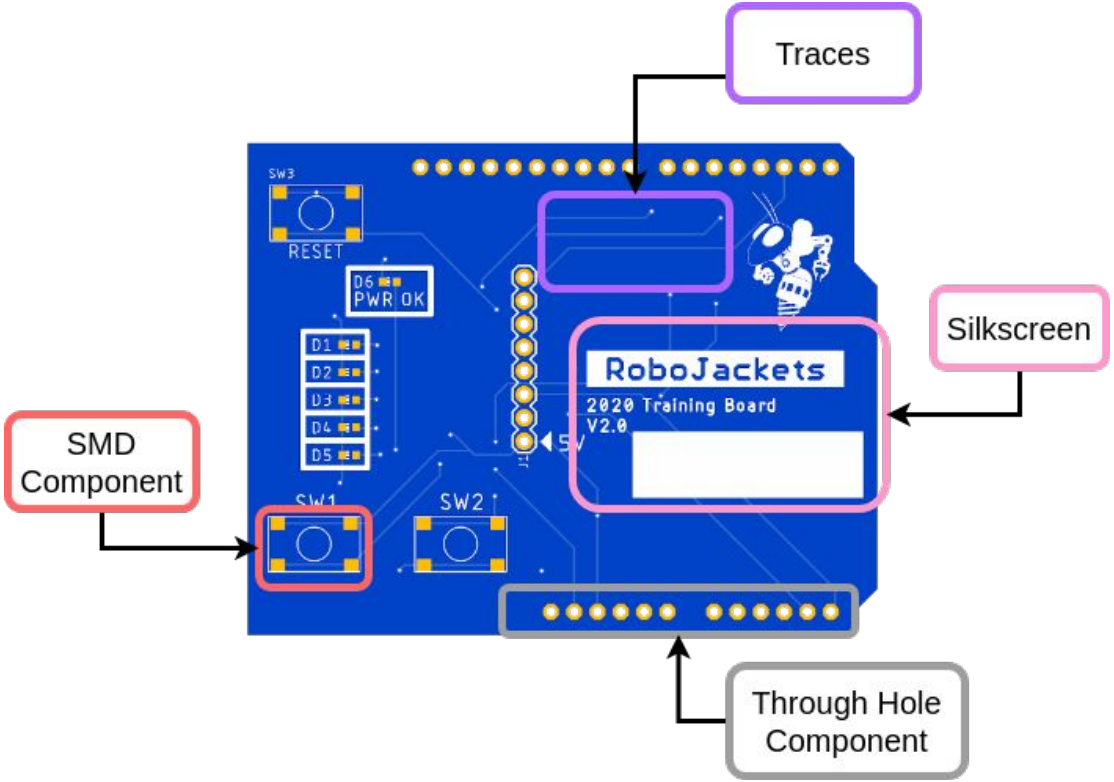
Layers

- 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

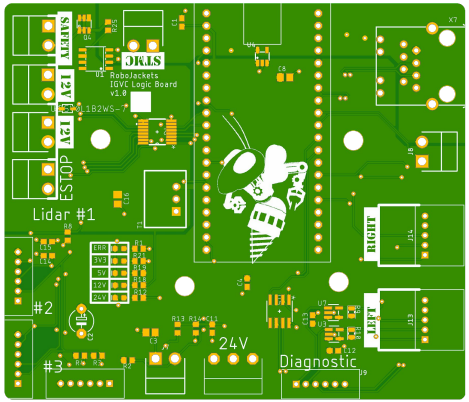
Training Board

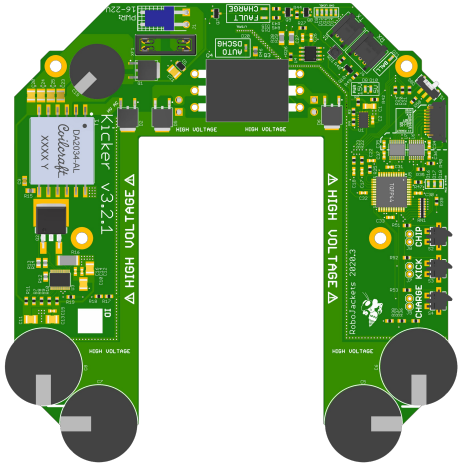


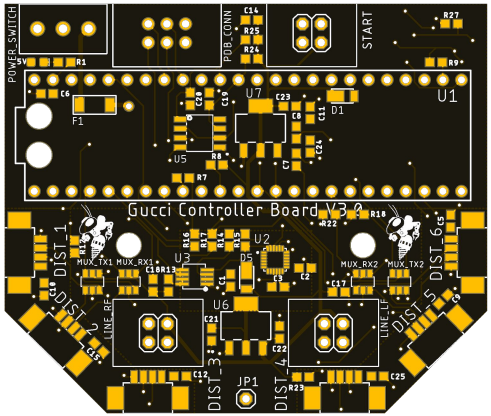
ROBOJACKETS
COMPETITIVE ROBOTICS AT GEORGIA TECH

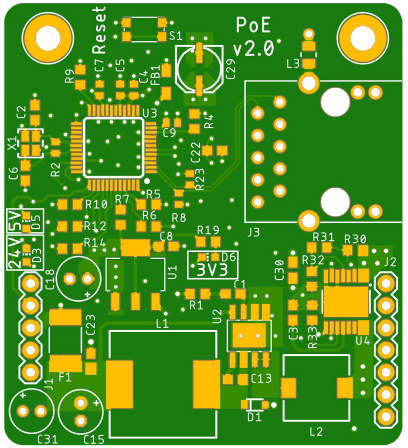
Team Examples

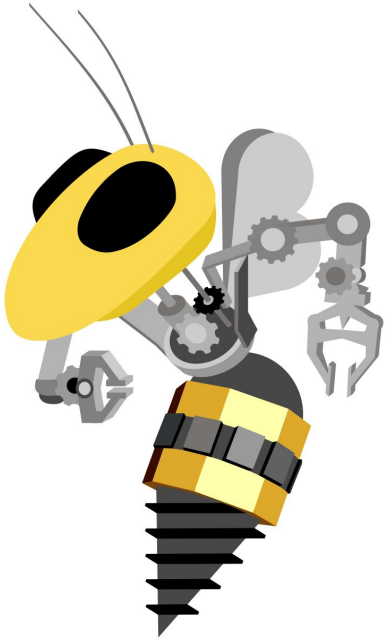
We use PCBs for a wide range of problems







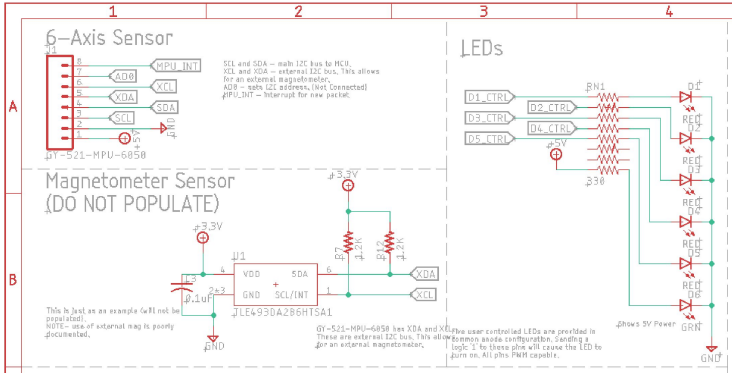


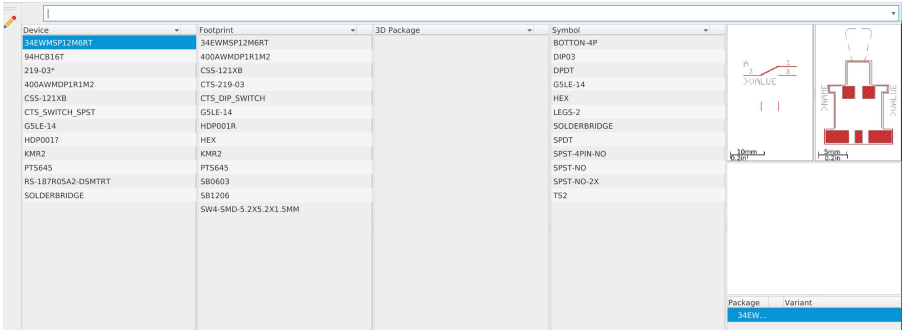


What is EAGLE?

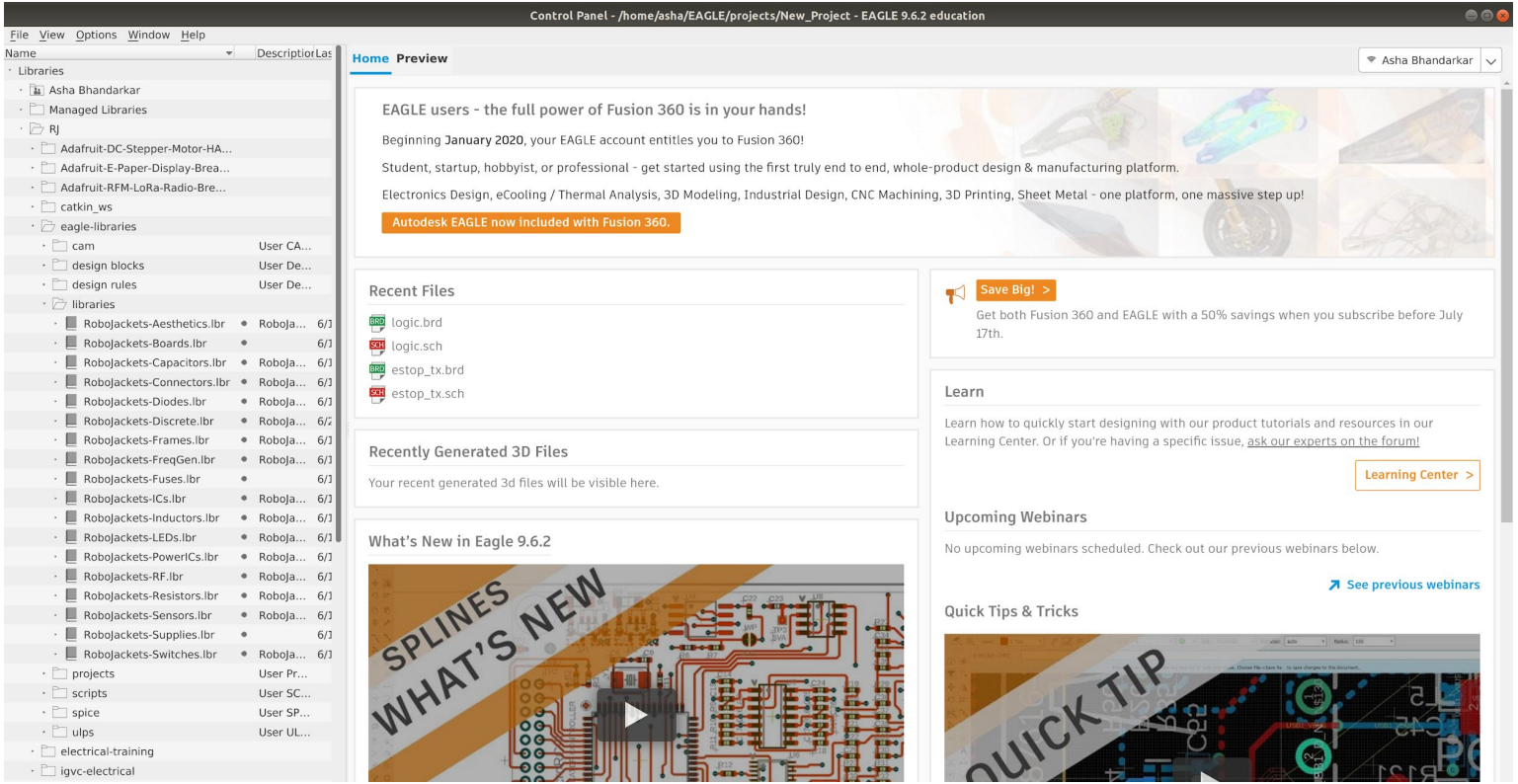
EAGLE

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



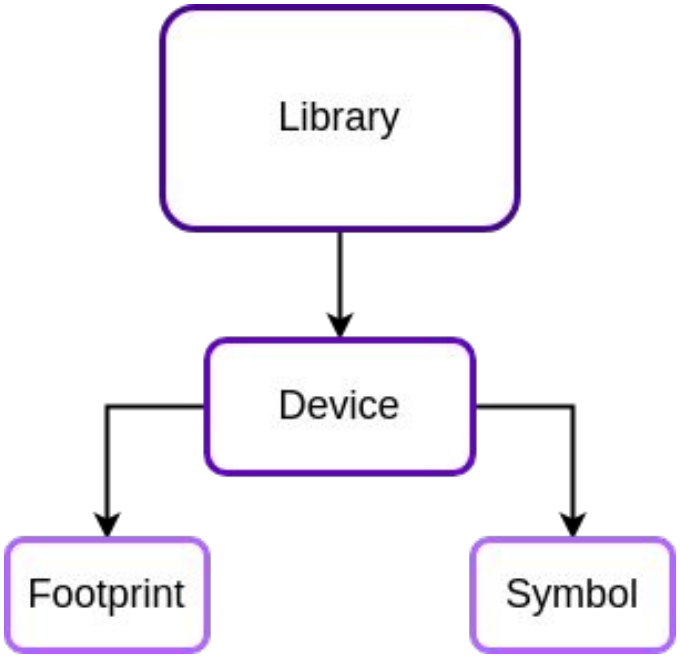


Control Panel



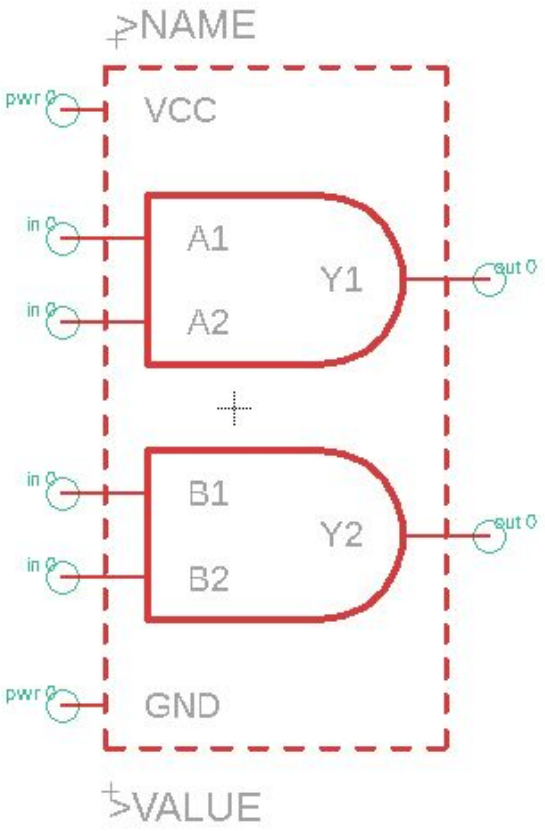
Libraries

- Store various components used in projects
- Libraries organize devices (electrical components) to be comprised of a symbol and footprint



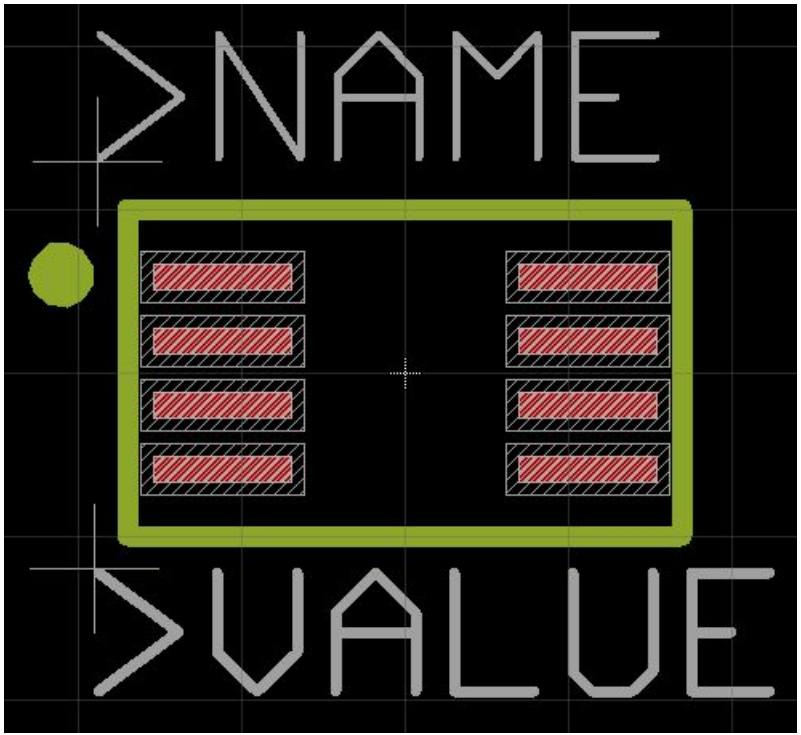
Symbol

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



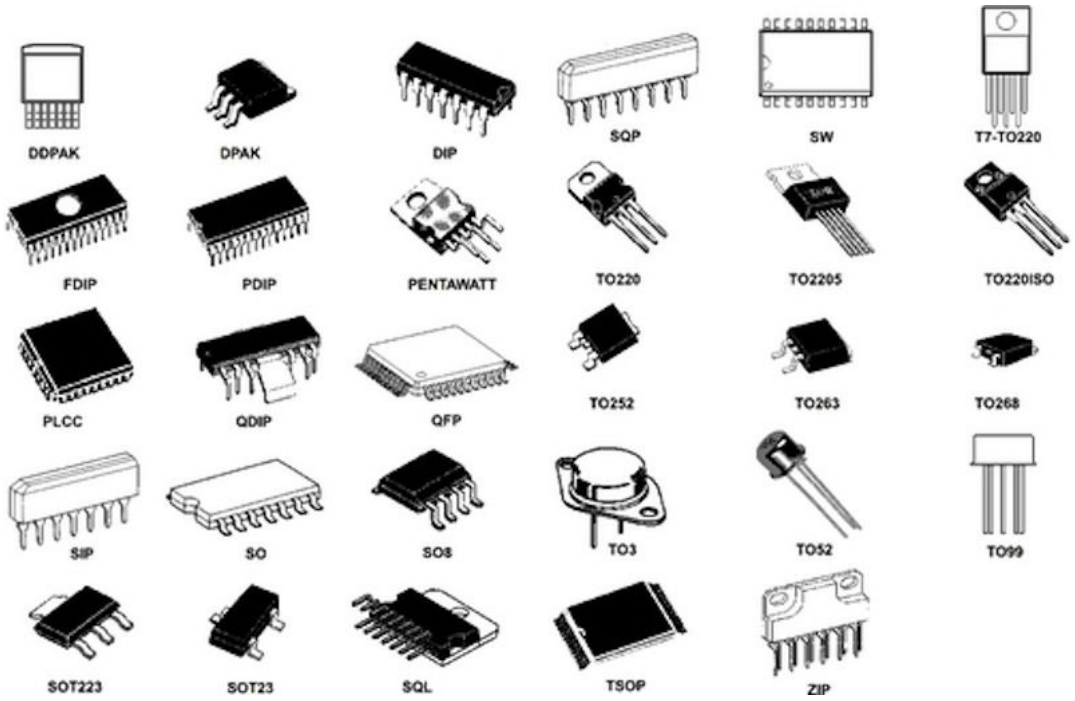
Footprint

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



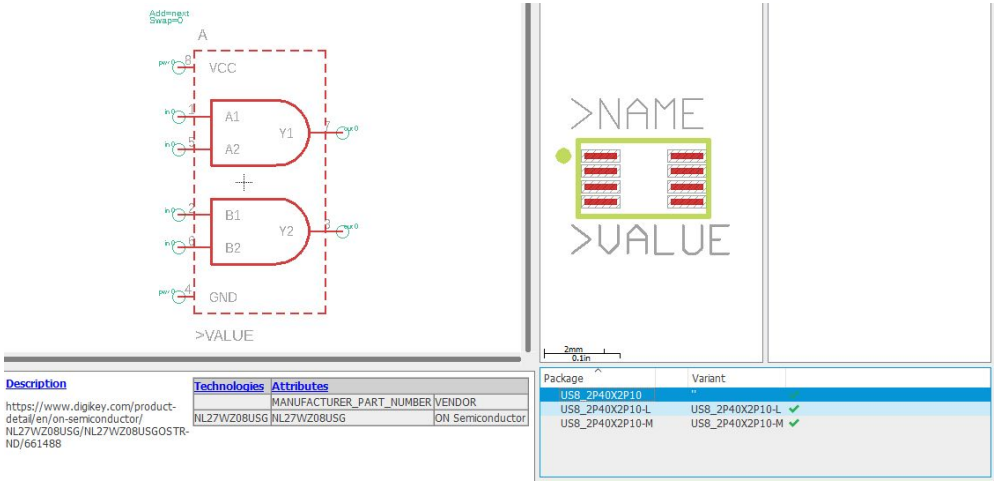
Common Packages

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

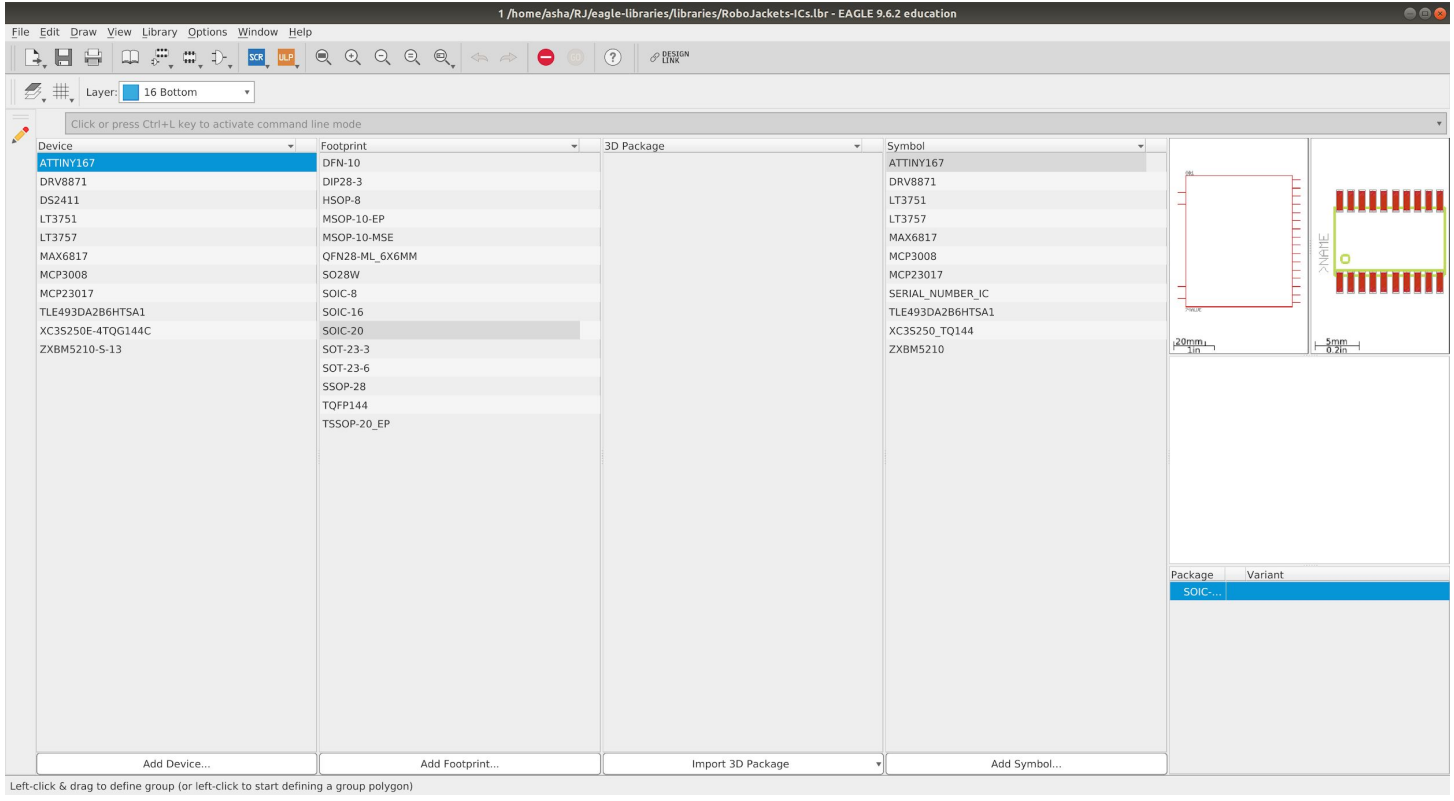


Device

- Links the symbol and footprint together by mapping the symbol's pins to the footprint pins
- A symbol can have multiple footprints if the component comes in multiple different packages (common with ICs)

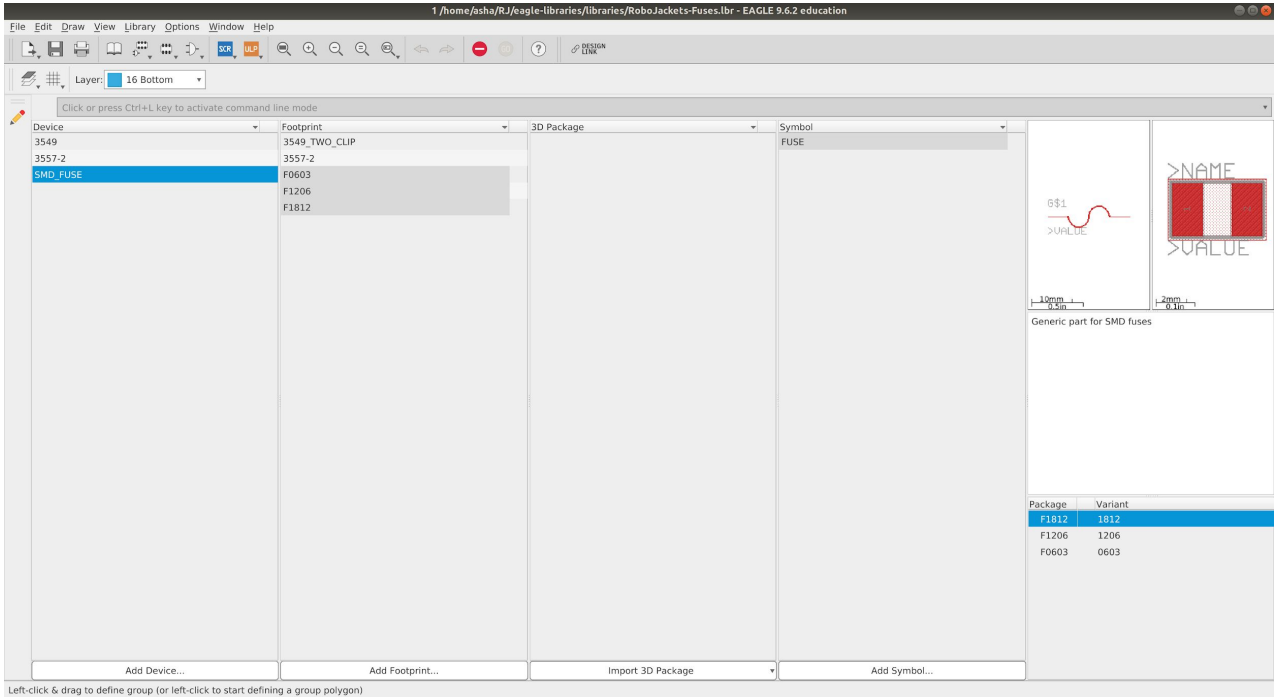


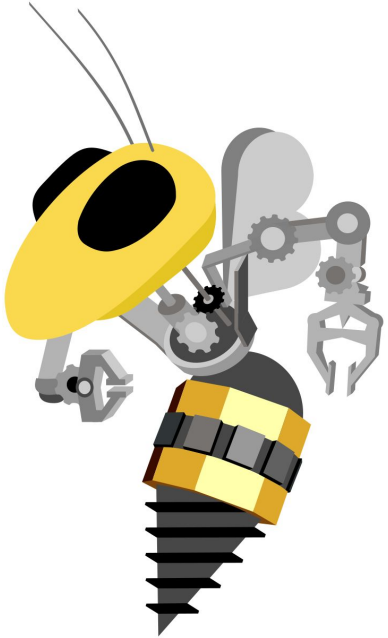
Example Library



Example Library

A component with multiple footprints





Lab!

Setup + Making a Part in EAGLE

Installing Software

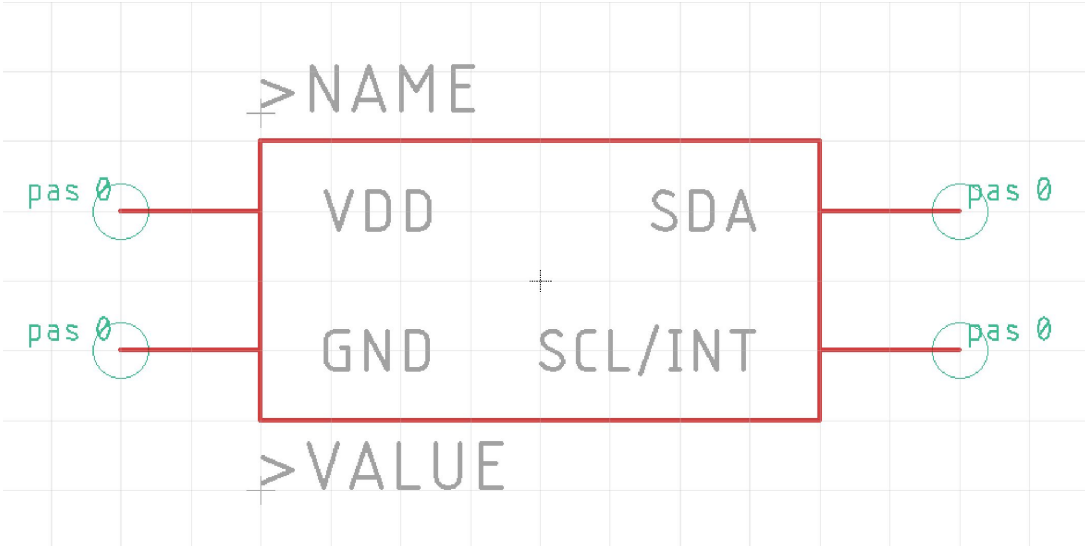
- EAGLE
  - https://www.autodesk.com/products/eagle/free-download
  - Requires Autodesk account registered with GA Tech email
    - https://www.autodesk.com/education/edu-software/overview?sorting=featured&page=1

Adding Repositories

- Adding the #eagle-libraries repository where RJ parts are stored

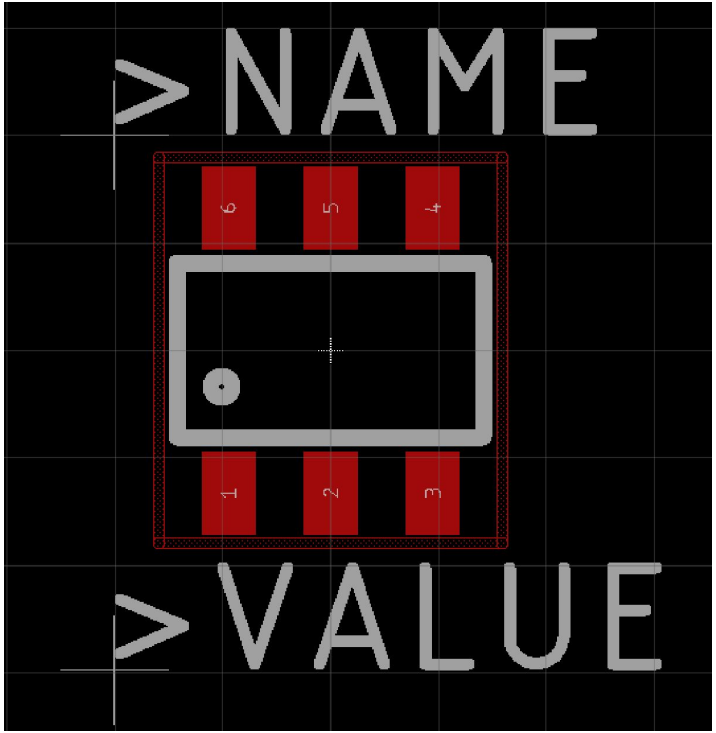
Making a Part - Symbol

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



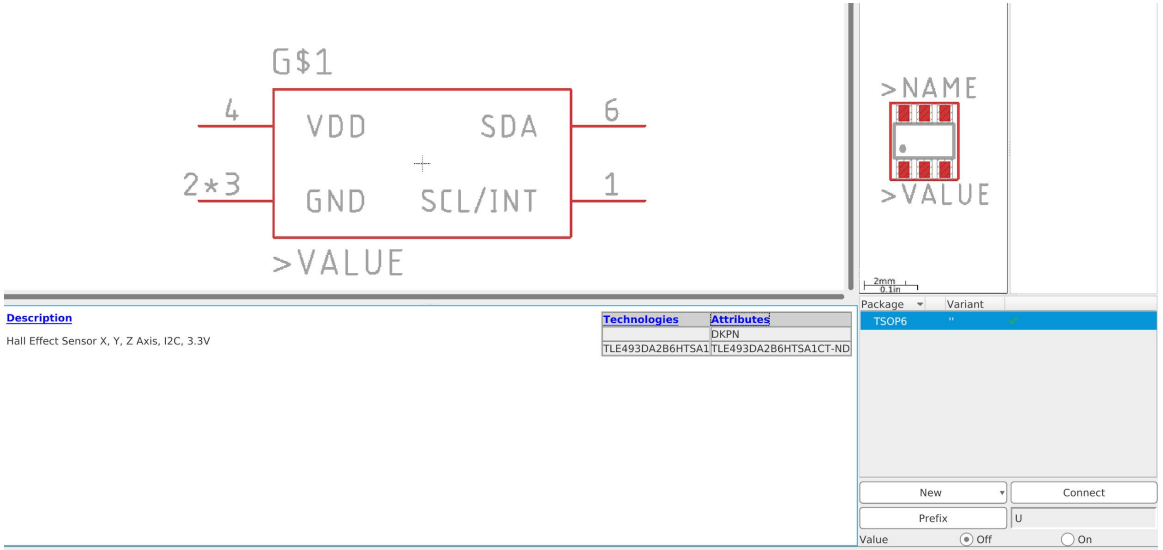
Making a Part - Footprint

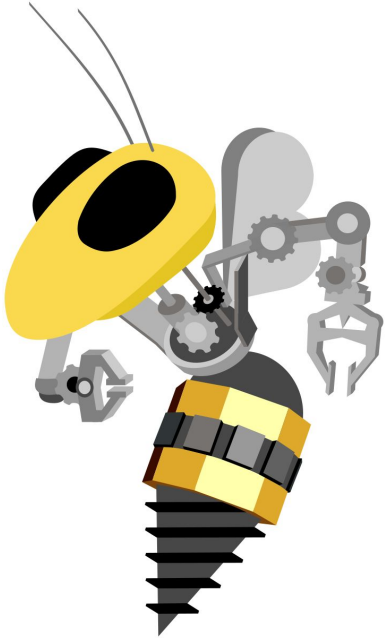
- Create a physical representation of the the part by reading the package outline details in the datasheet and then placing correctly sized pads/drills at the appropriate location



Making a Part - Device

- Map the symbol's pins to the footprint's pads, add prefix, and add descriptive information





Thank You!

Any Questions?