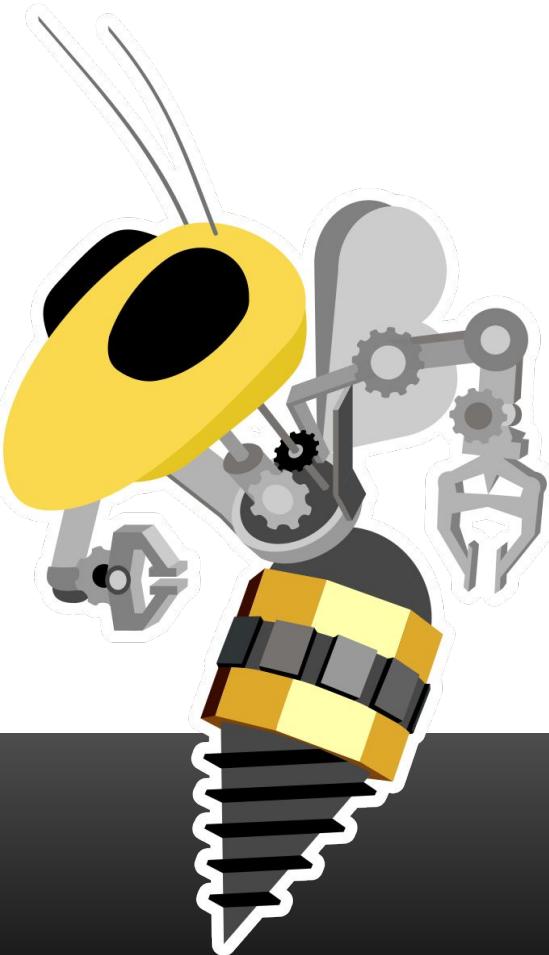


# Welcome!

Electrical Training  
Week 2

**ROBOJACKETS**  
COMPETITIVE ROBOTICS AT GEORGIA TECH

*[www.robojackets.org](http://www.robojackets.org)*

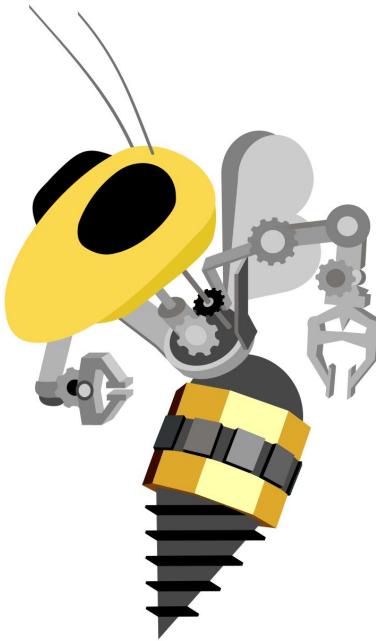


# Last Week!

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

# This Week!

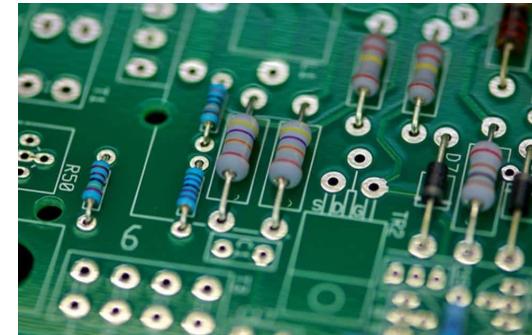
- Introduction to PCBs
- Introduction to EAGLE CAD
- Parts and Libraries in EAGLE
- Configuring EAGLE Setup
- Making a Part in EAGLE



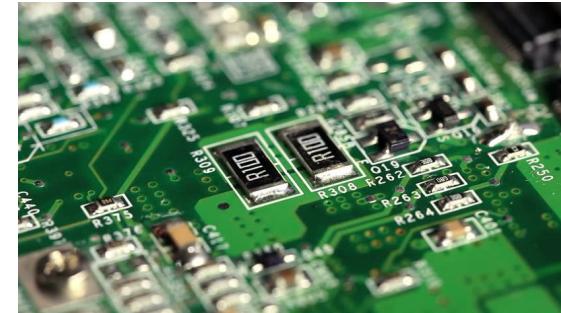
# What are PCBs?

# 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



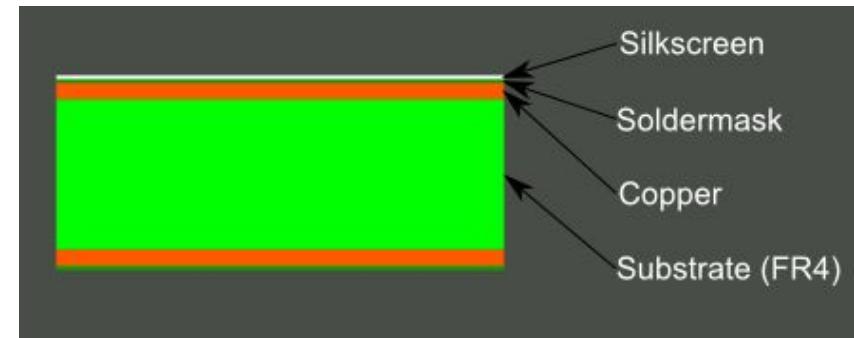
Through Hole



Surface Mount

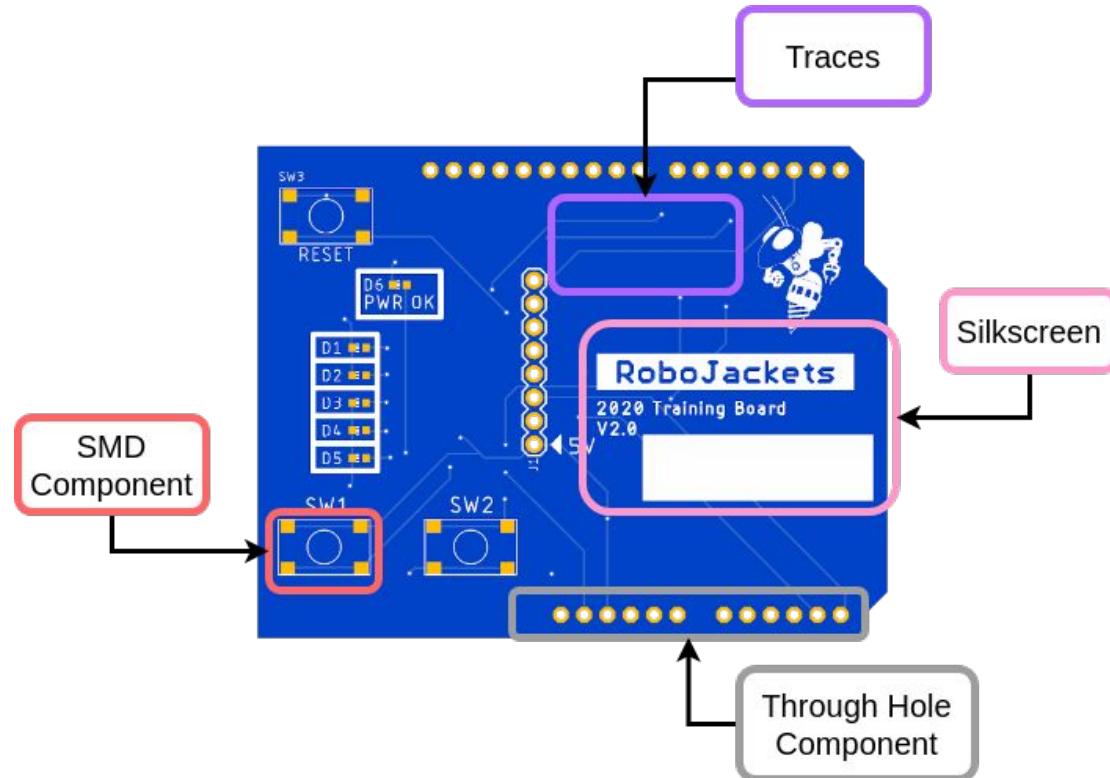
# Layers

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



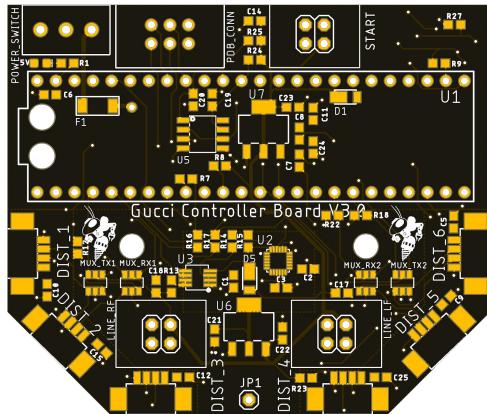
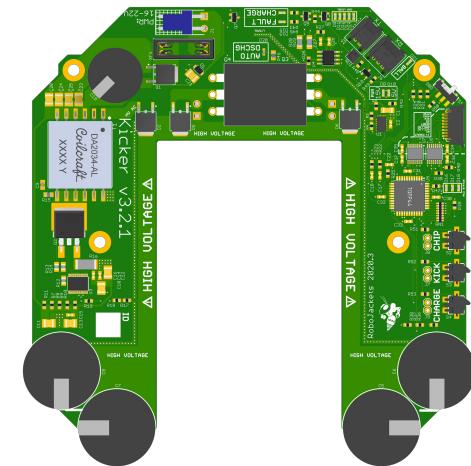
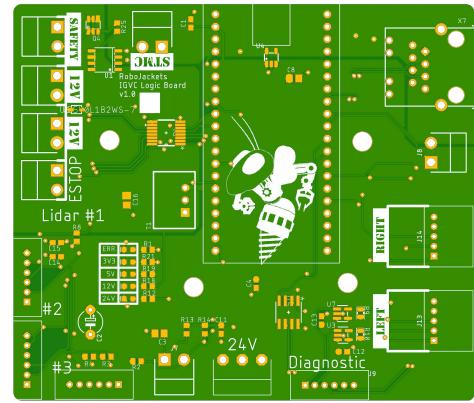
# PCB Features

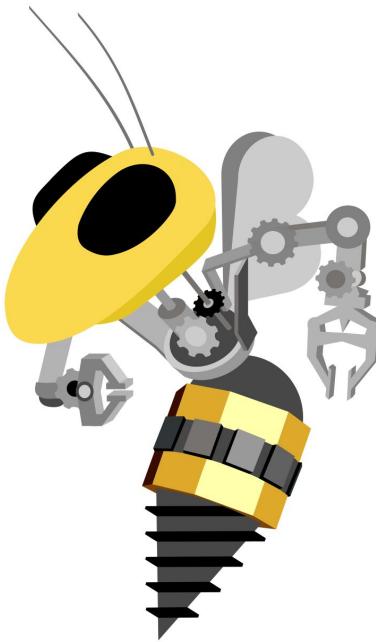
*Training Board*



# Team Examples

*We use PCBs  
for a wide  
range of  
problems*

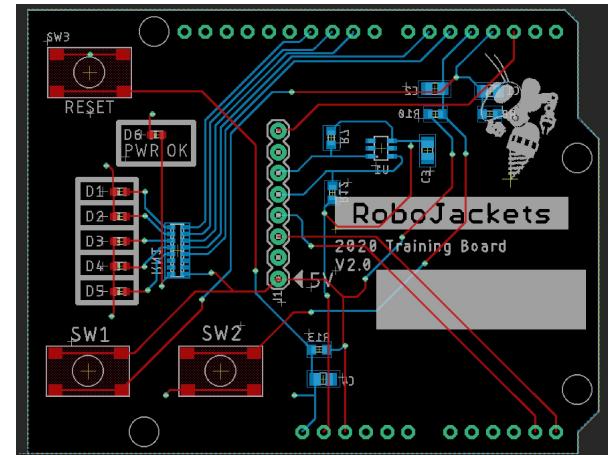
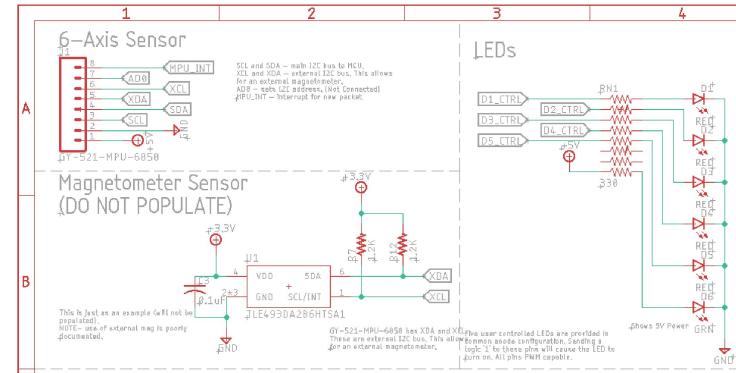
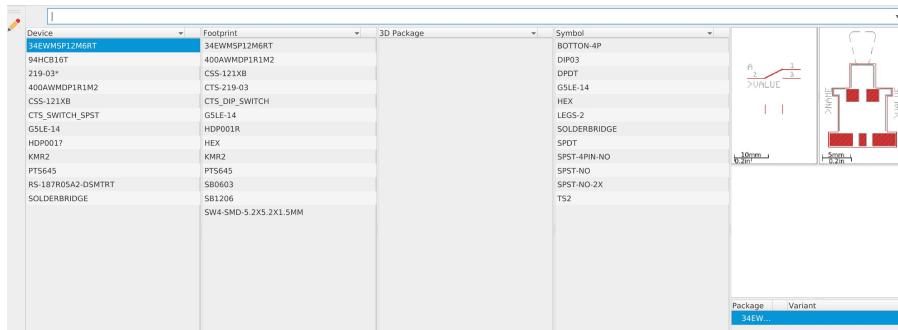




# What is EAGLE?

# EAGLE

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



# Control Panel

Control Panel - /home/asha/EAGLE/projects/New\_Project - EAGLE 9.6.2 education

File View Options Window Help

Name Description Last Modified

- Libraries
  - Asha Bhandarkar
  - Managed Libraries
  - RJ
    - Adafruit-DC-Stepper-Motor-HA...
    - Adafruit-E-Paper-Display-Brea...
    - Adafruit-RFM-LoRa-Radio-Bre...
  - catkin\_ws
  - eagle-libraries
    - cam
    - design blocks
    - design rules
  - libraries
    - Robojackets-Aesthetics.lbr
    - Robojackets-Boards.lbr
    - Robojackets-Capacitors.lbr
    - Robojackets-Connectors.lbr
    - Robojackets-Diodes.lbr
    - Robojackets-Discrete.lbr
    - Robojackets-Frames.lbr
    - Robojackets-FreqGen.lbr
    - Robojackets-Fuses.lbr
    - Robojackets-ICs.lbr
    - Robojackets-Inductors.lbr
    - Robojackets-LEDs.lbr
    - Robojackets-PowerCs.lbr
    - Robojackets-RF.lbr
    - Robojackets-Resistors.lbr
    - Robojackets-Sensors.lbr
    - Robojackets-Supplies.lbr
    - Robojackets-Switches.lbr
  - projects
  - scripts
  - spice
  - ulp
  - electrical-training
  - igvc-electrical

Home Preview

EAGLE users - the full power of Fusion 360 is in your hands!

Beginning January 2020, your EAGLE account entitles you to Fusion 360!

Student, startup, hobbyist, or professional - get started using the first truly end to end, whole-product design & manufacturing platform.

Electronics Design, eCooling / Thermal Analysis, 3D Modeling, Industrial Design, CNC Machining, 3D Printing, Sheet Metal - one platform, one massive step up!

Autodesk EAGLE now included with Fusion 360.

Recent Files

- logic.brd
- logic.sch
- estop\_tx.brd
- estop\_tx.sch

Recently Generated 3D Files

Your recent generated 3d files will be visible here.

What's New in Eagle 9.6.2

SPLINES  
WHAT'S NEW

Upcoming Webinars

No upcoming webinars scheduled. Check out our previous webinars below.

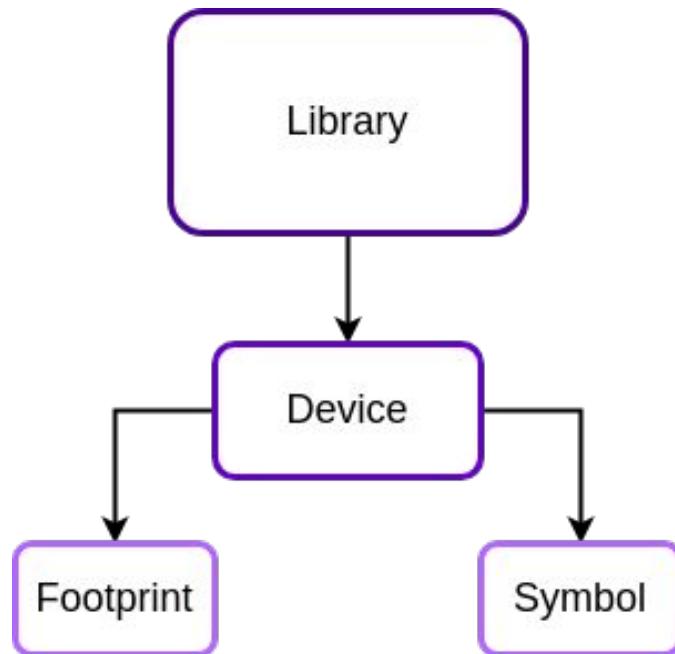
See previous webinars

Quick Tips & Tricks

QUICK TIP

# Libraries

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



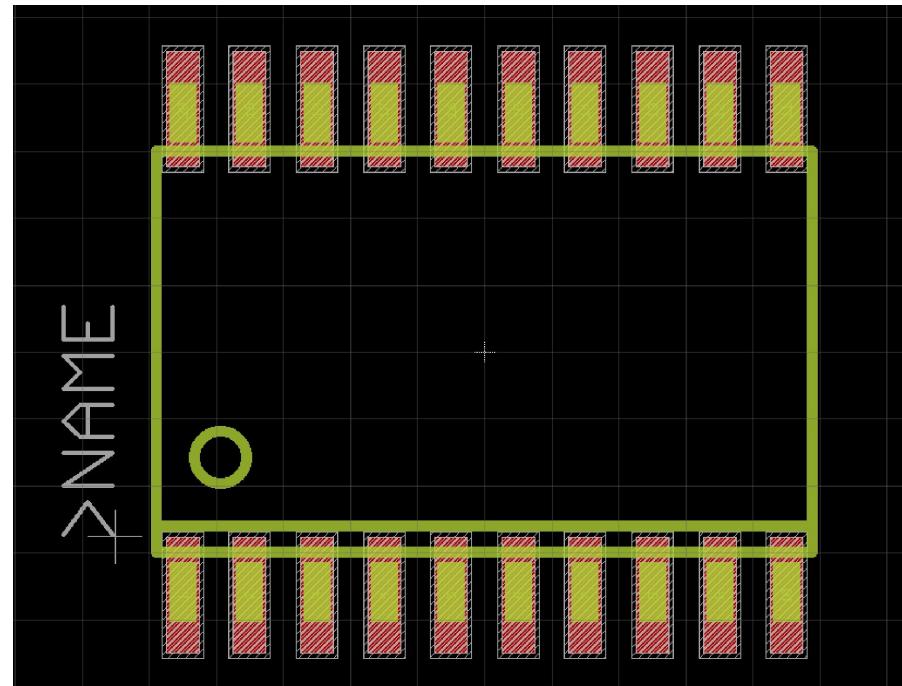
# Symbol

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

>NAME	>VALUE
io 0	VCC
io 1	AVCC
io 2	PA0(RXLIN/RXD/ADC0/PCINT0)
io 3	PA1(TXLIN/TXD/ADC1/PCINT1)
io 4	PA2(MISO/D0/OC0A/ADC2/PCINT2)
io 5	PA3(INT1/ISRC/ADC3/PCINT3)
io 6	PA4(MOSI/SDA/DI/ICP1/ADC4/PCINT4)
io 7	PA5(SCK/SCL/USCK/T1/ADC5/PCINT5)
io 8	PA6(SS/AIN0/ADC6/PCINT6)
io 9	PA7(AREF/XREF/AIN1/ADC7/PCINT7)
io 10	PB0(PCINT8/OC1AU/DI/SDA)
io 11	PB1(PCINT9/OC1BU/DD)
io 12	PB2(PCINT10/OC1AV/USCK/SCL)
io 13	PB3(PCINT11/OC1BV)
io 14	PB4(PCINT12/OC1AW/XTAL1/CLKI)
io 15	PB5(PCINT13/ADC8/OC1BW/XTAL2/CLKO)
io 16	PB6(PCINT14/ADC9/OC1AX/INT0)
io 17	PB7(PCINT15/ADC10/OC1BX/RESET/DW)
io 18	AGND
io 19	GND

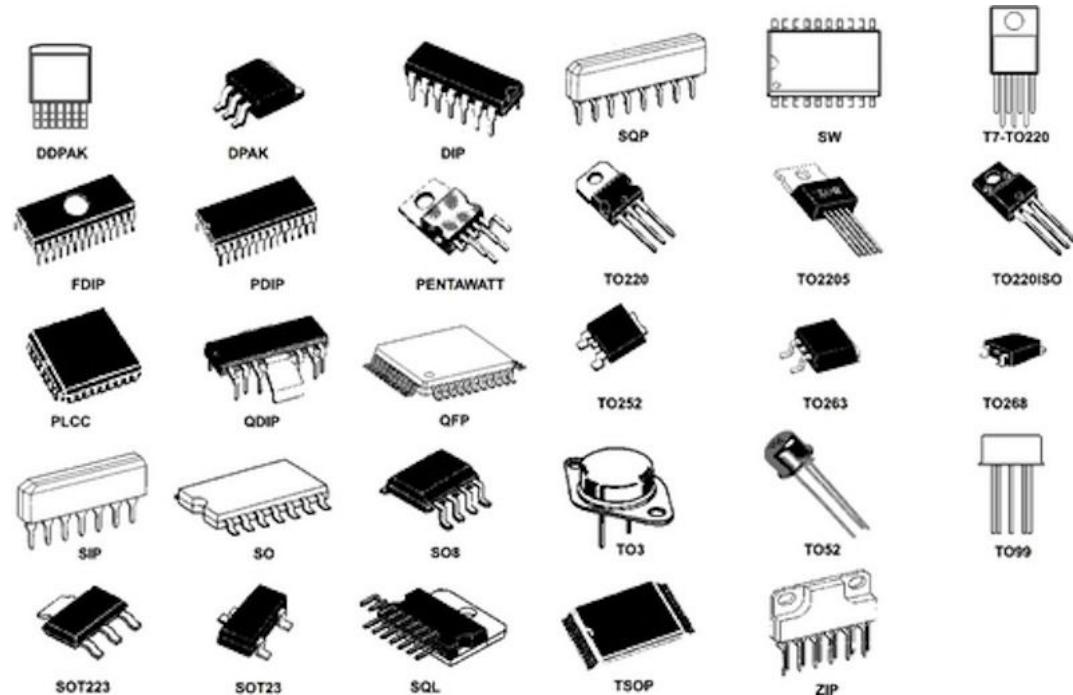
# Footprint

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



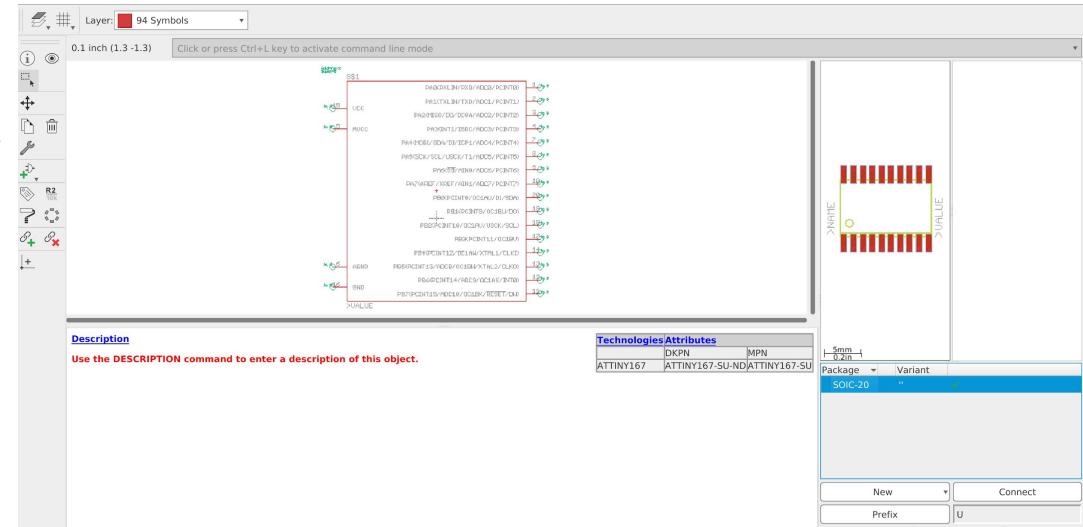
# Common Packages

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

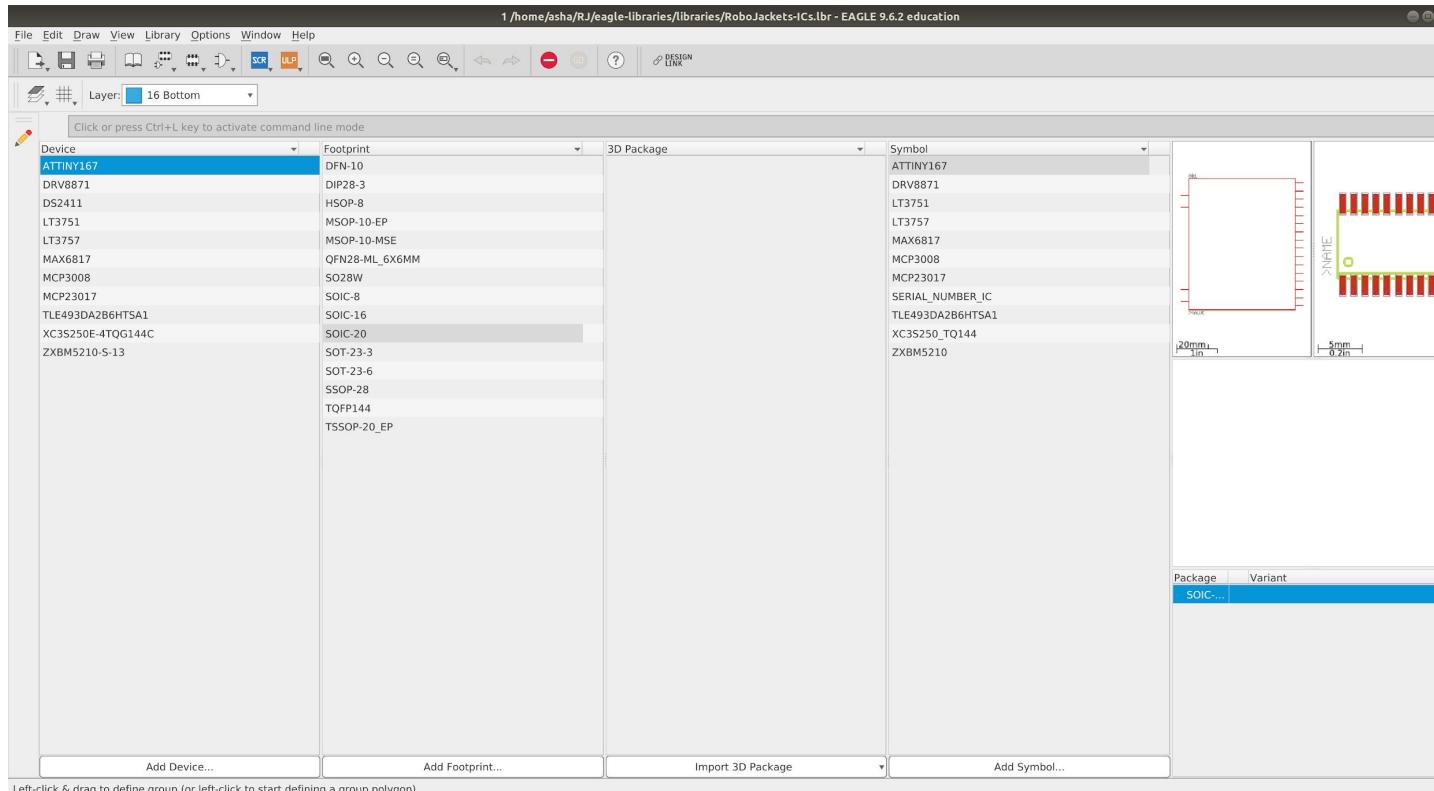


# Device

- 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)

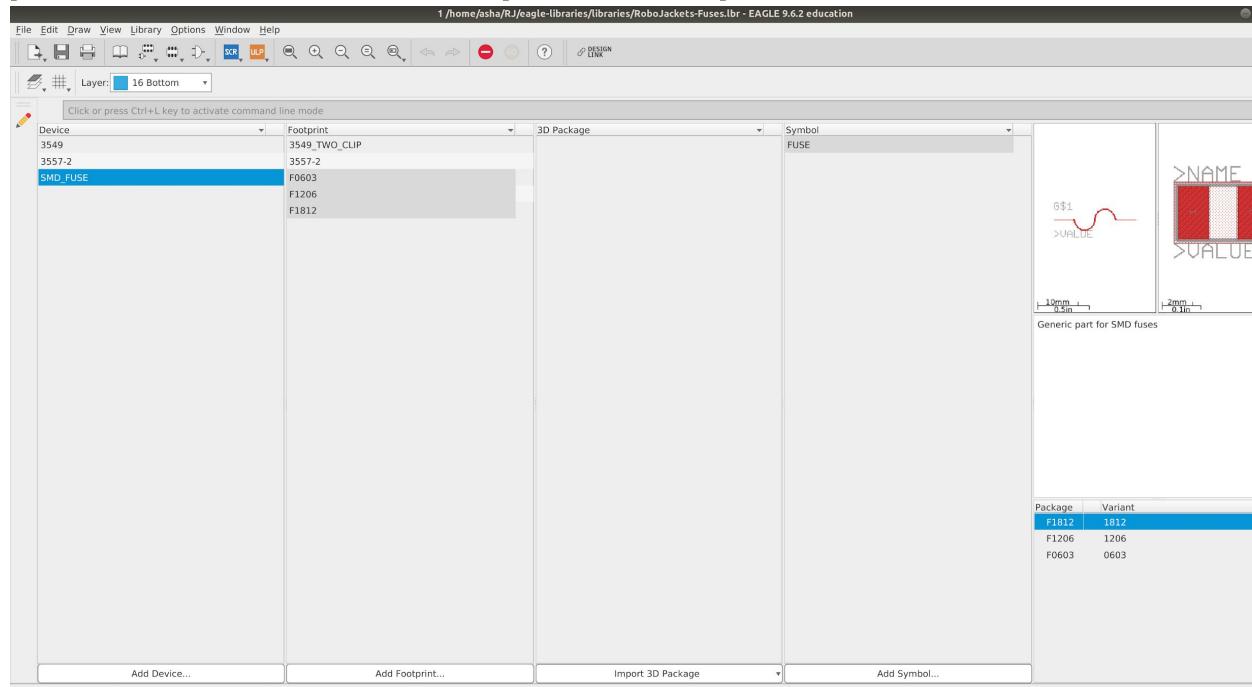


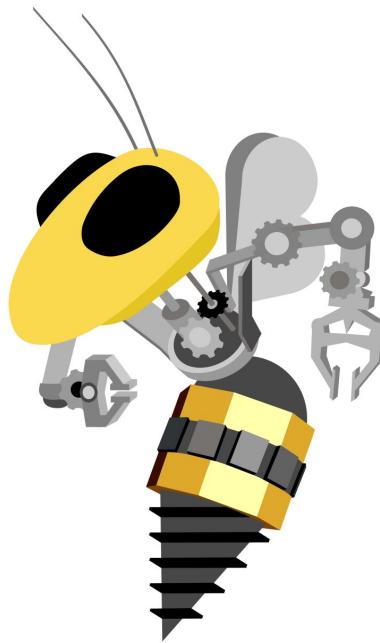
# Example Library



# Example Library

A component with multiple footprints





# Lab!

Setup + Making a Part in  
EAGLE

# Installing Software

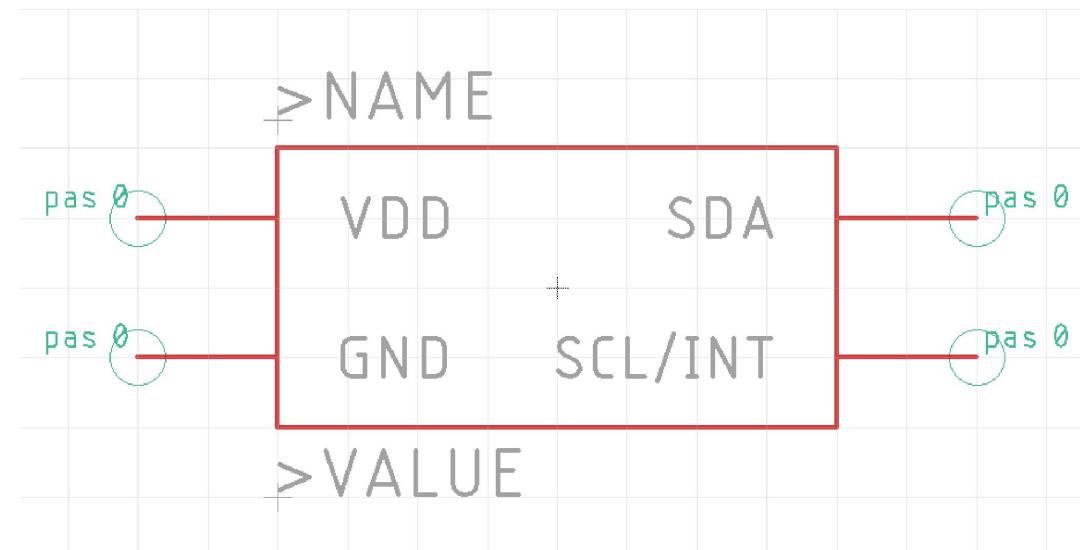
- Should have EAGLE installed and either GitHub Desktop or terminal Git
- If not, check the lab document and earlier sent email about this

# Adding Repositories

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

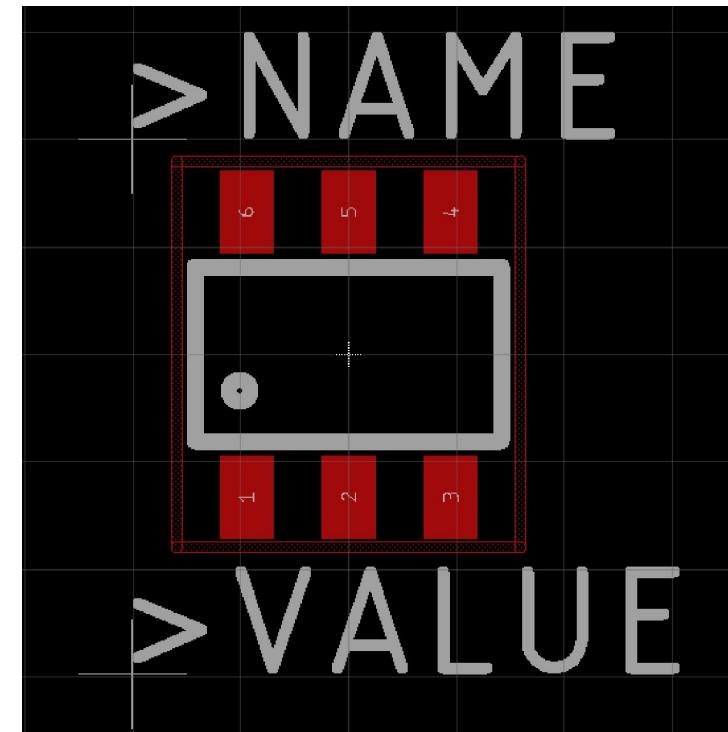
# Making a Part - Symbol

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



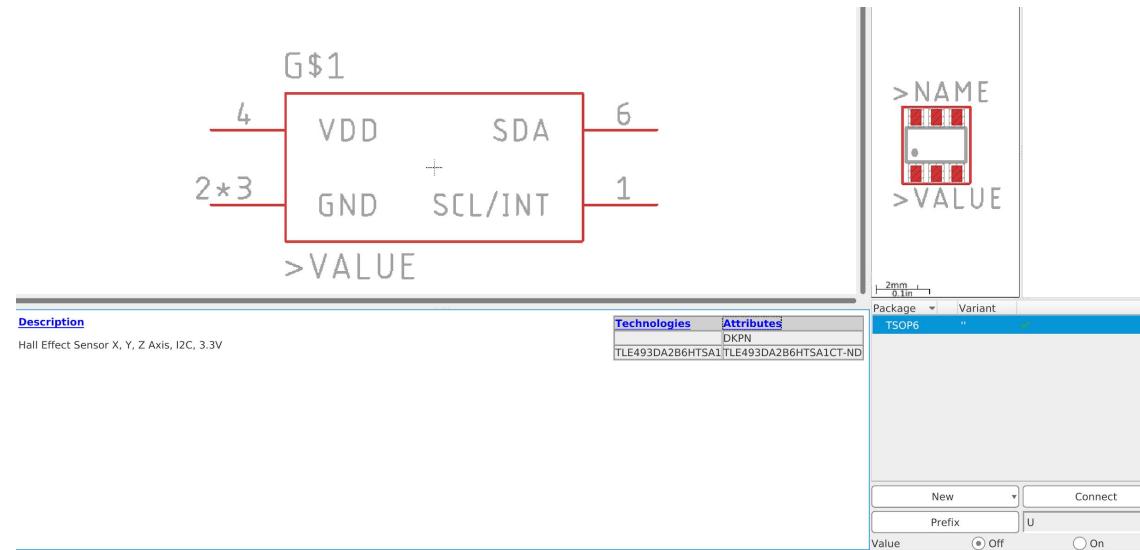
# Making a Part - Footprint

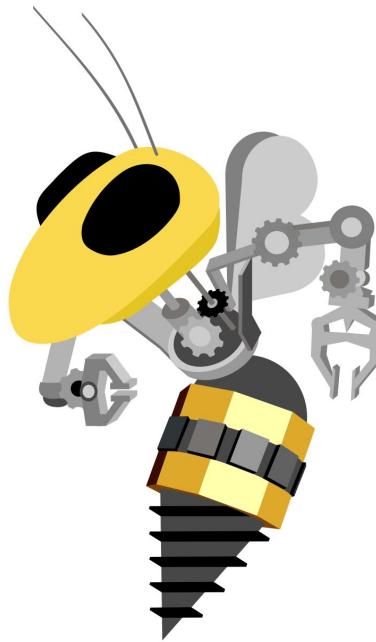
- Create a physical representation of the part by reading the package outline details in the datasheet and then placing correctly sized pads/drills at the appropriate location



# Making a Part - Device

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





# Thank You!

Any Questions?