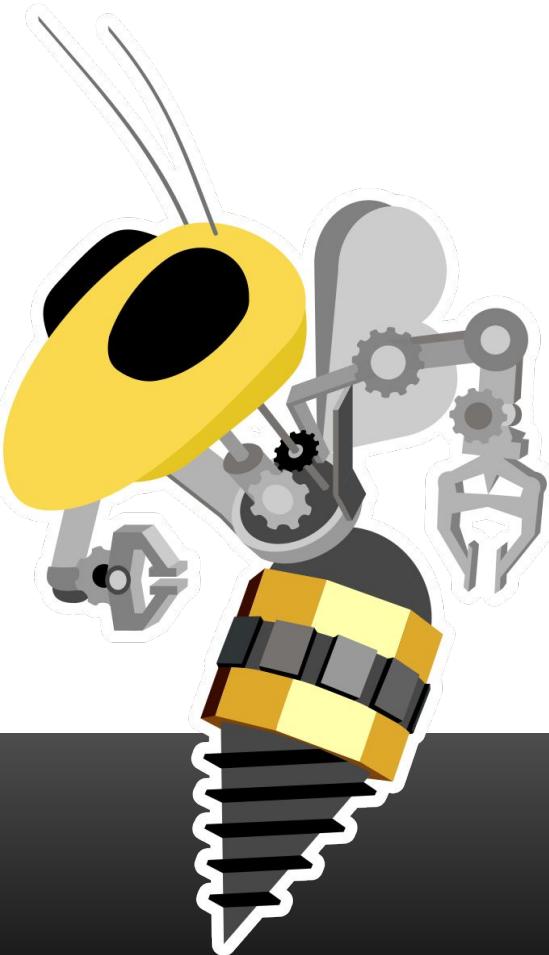


Welcome!

Electrical Training
Week 2

ROBOJACKETS
COMPETITIVE ROBOTICS AT GEORGIA TECH

www.robojackets.org

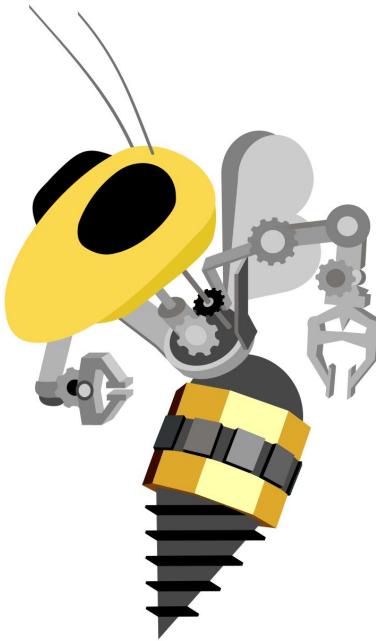


Last Week!

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

This Week!

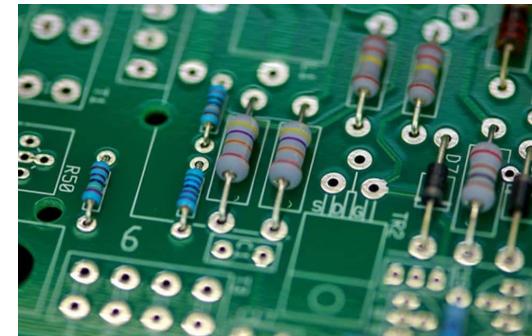
- Introduction to PCBs
- Introduction to EAGLE CAD
- Parts and Libraries in EAGLE
- Configuring EAGLE Setup
- Making a Part in EAGLE



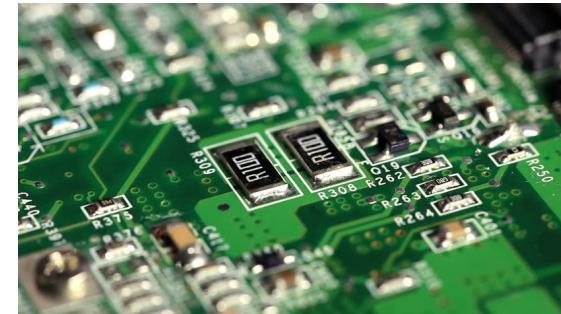
What are PCBs?

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



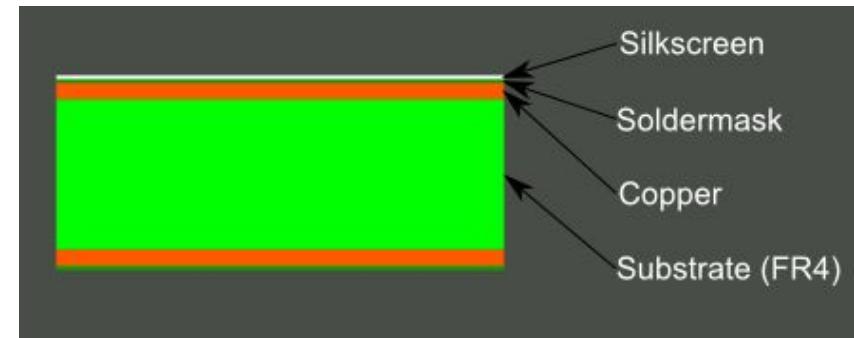
Through Hole



Surface Mount

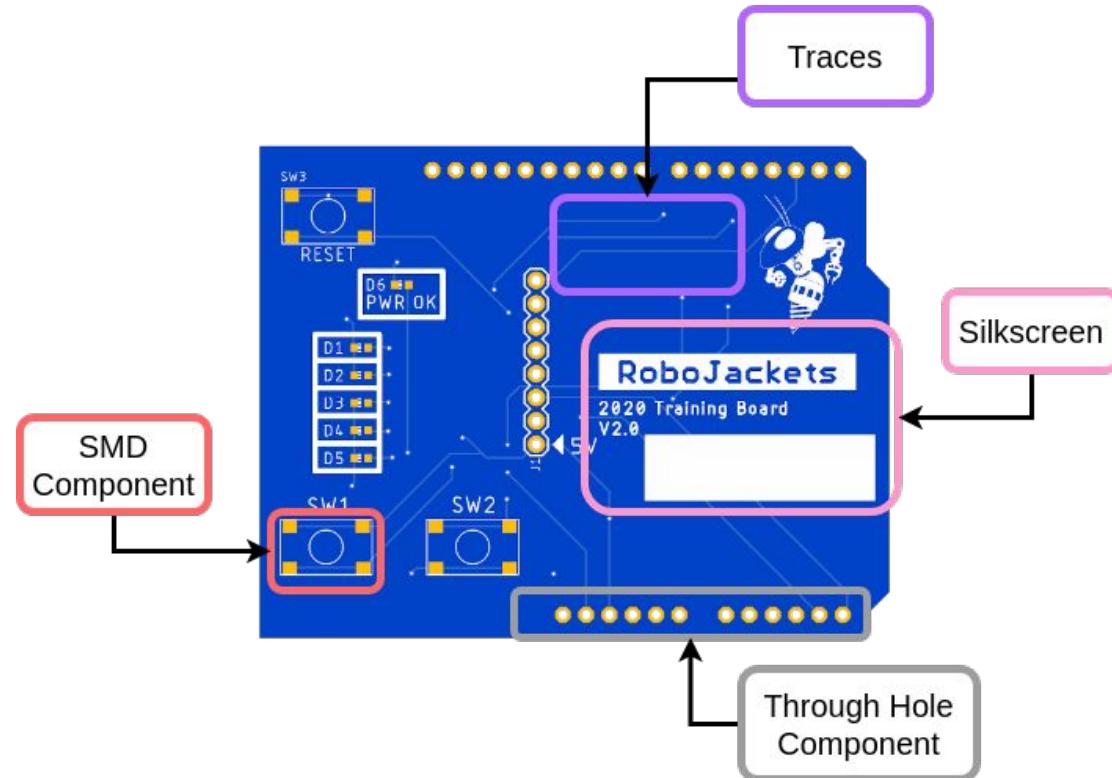
Layers

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



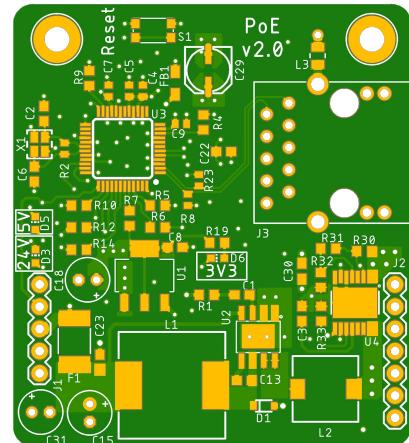
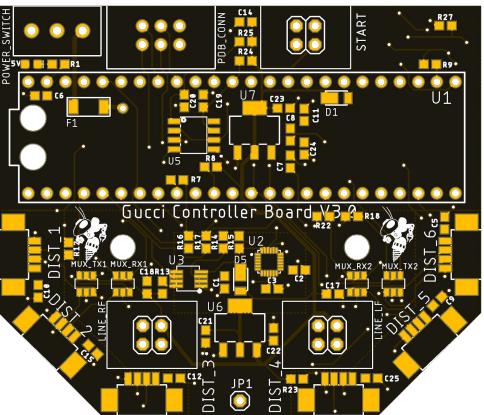
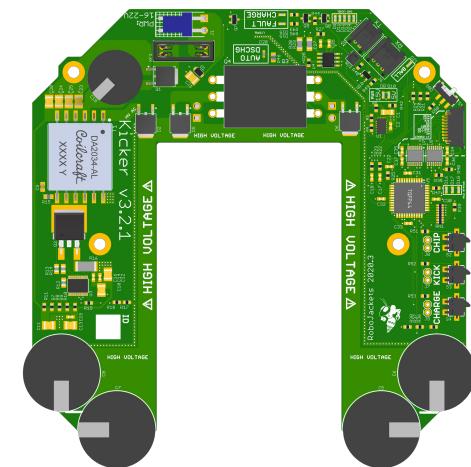
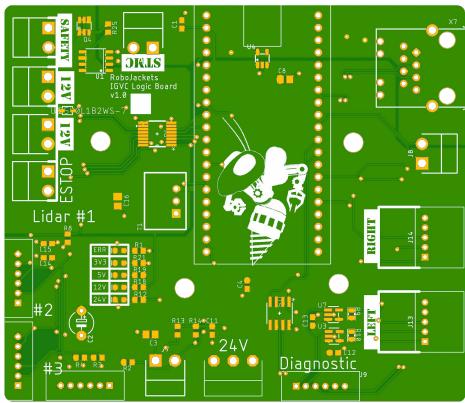
PCB Features

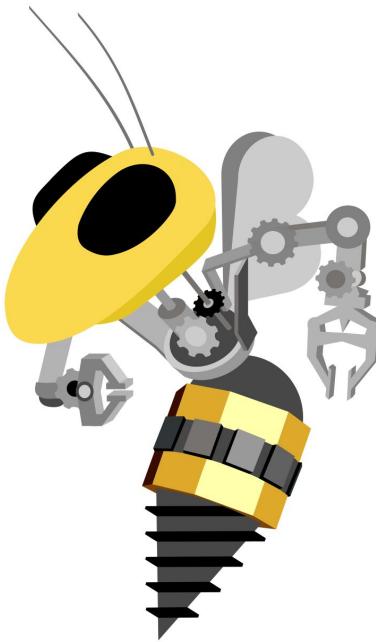
Training Board



Team Examples

*We use PCBs
for a wide
range of
problems*

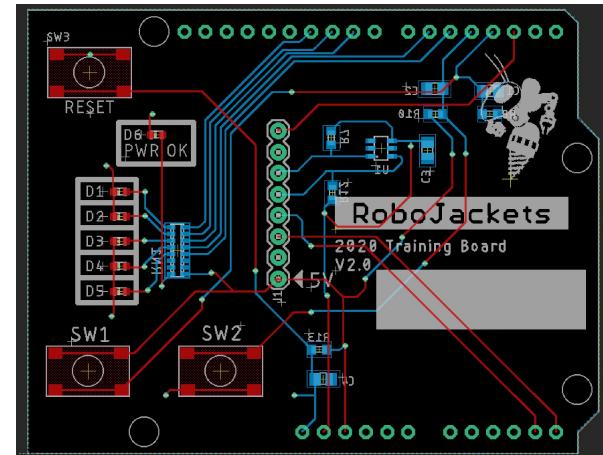
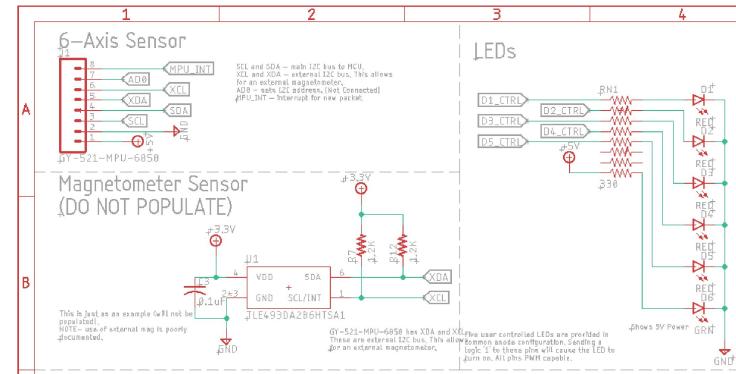
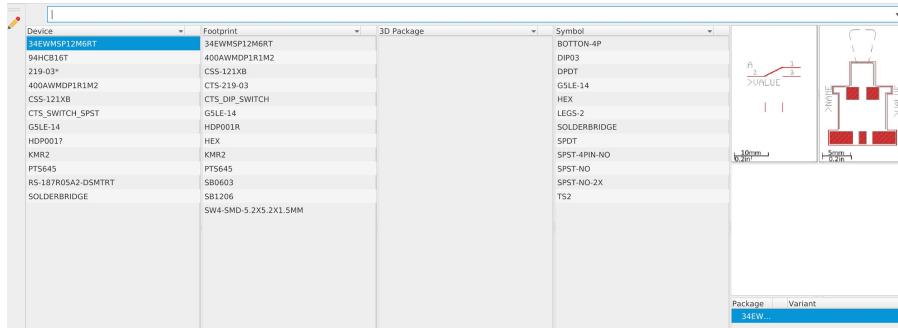




What is EAGLE?

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.

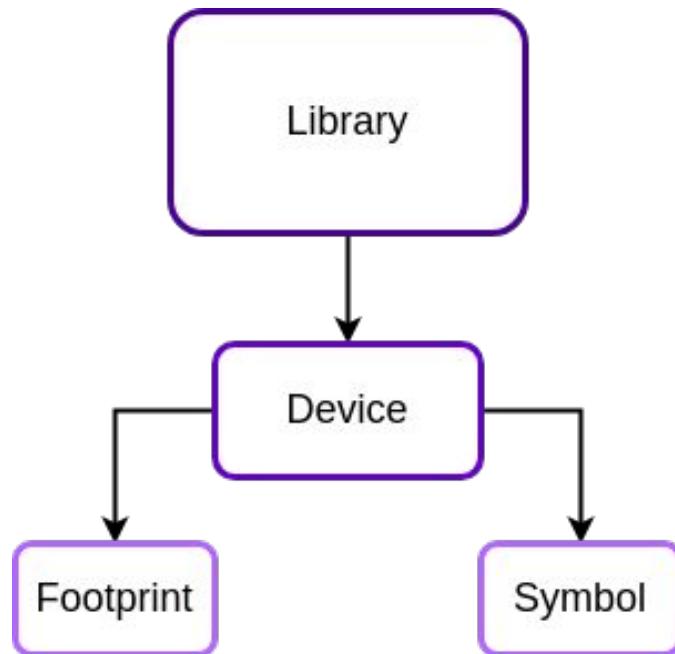
See previous webinars

Quick Tips & Tricks

QUICK TIP

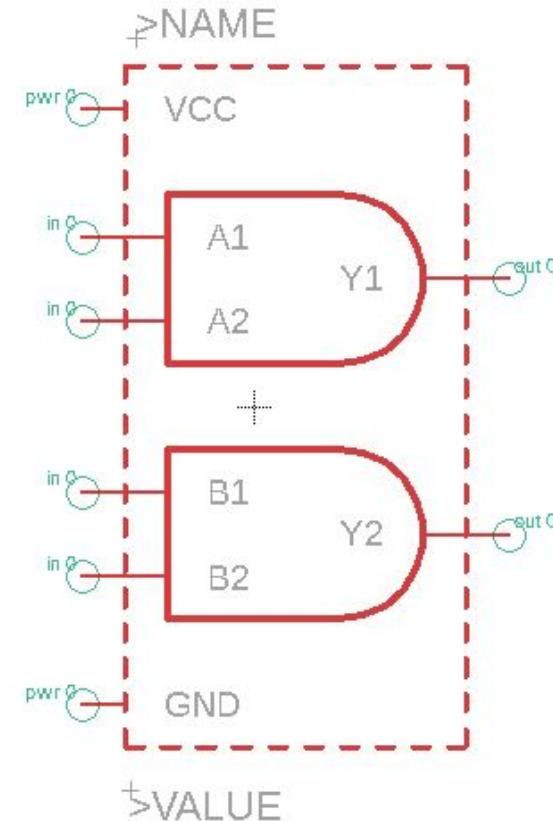
Libraries

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



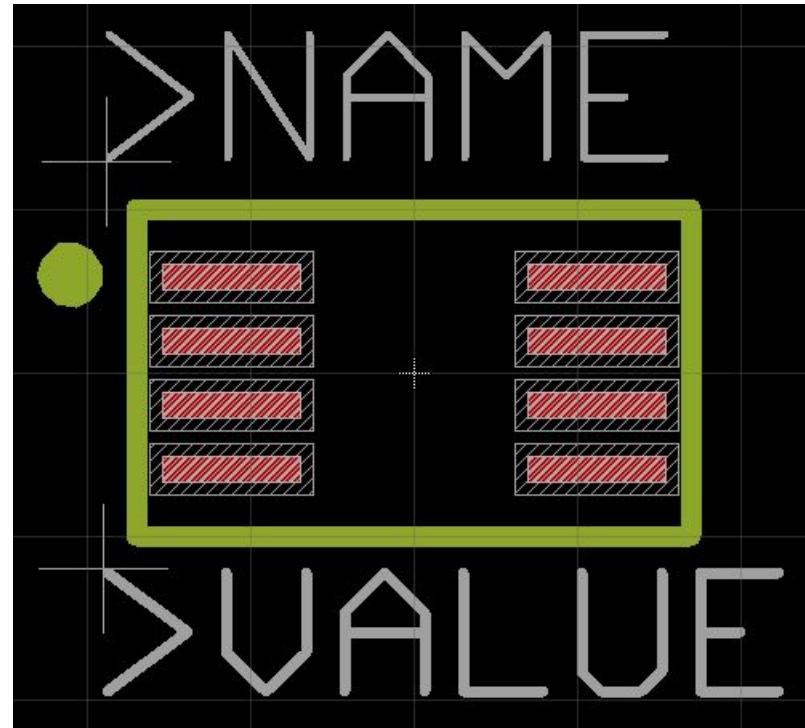
Symbol

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



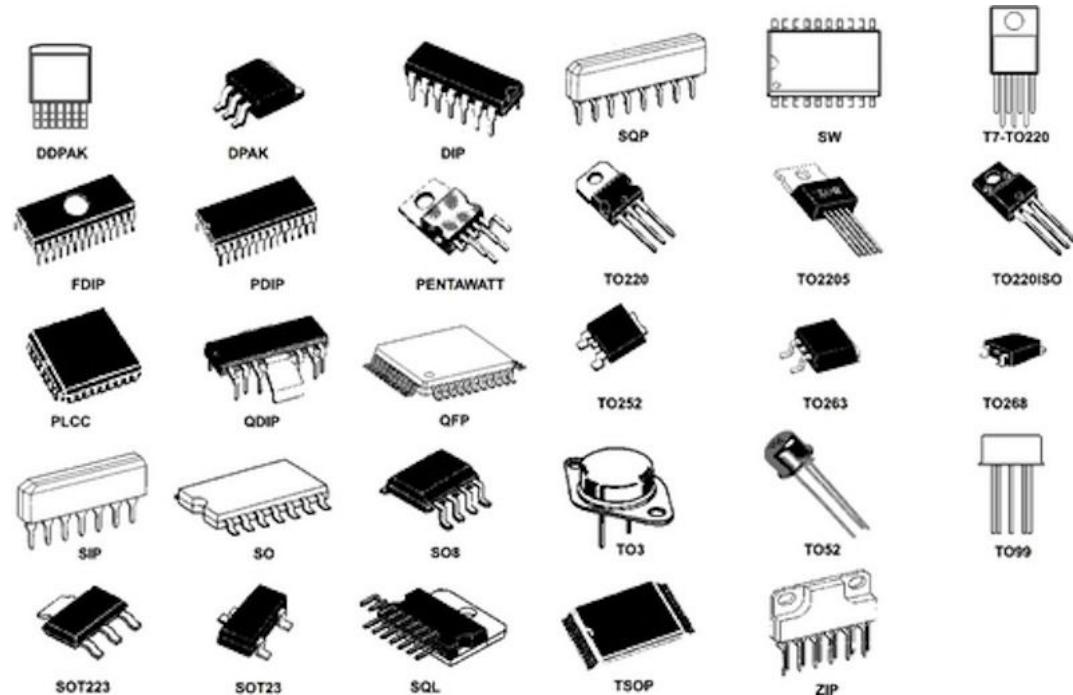
Footprint

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



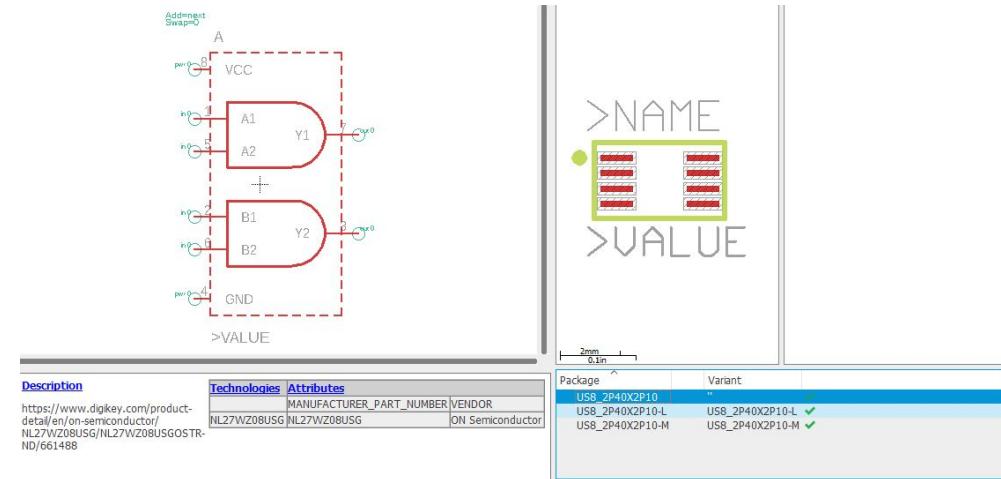
Common Packages

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

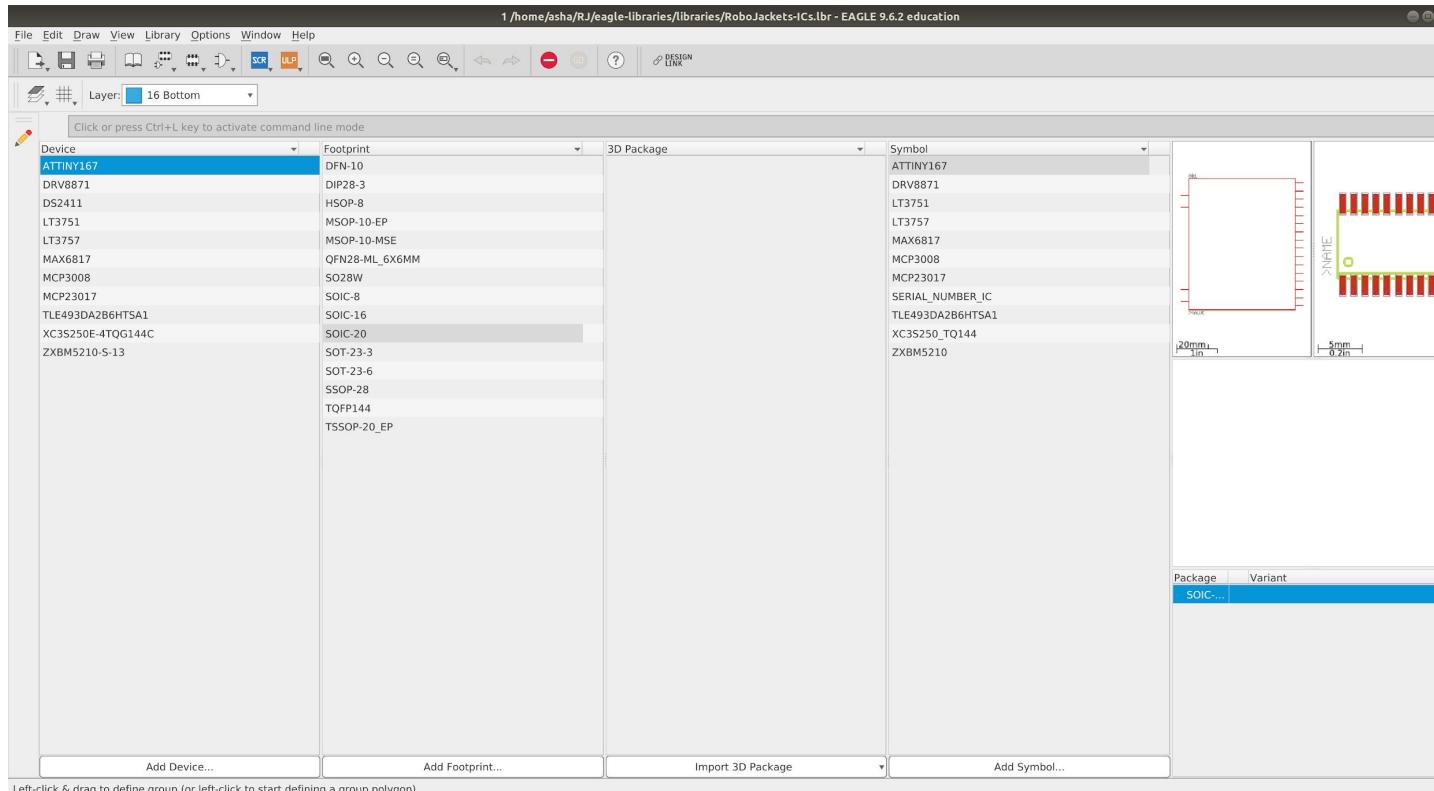


Device

- 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)

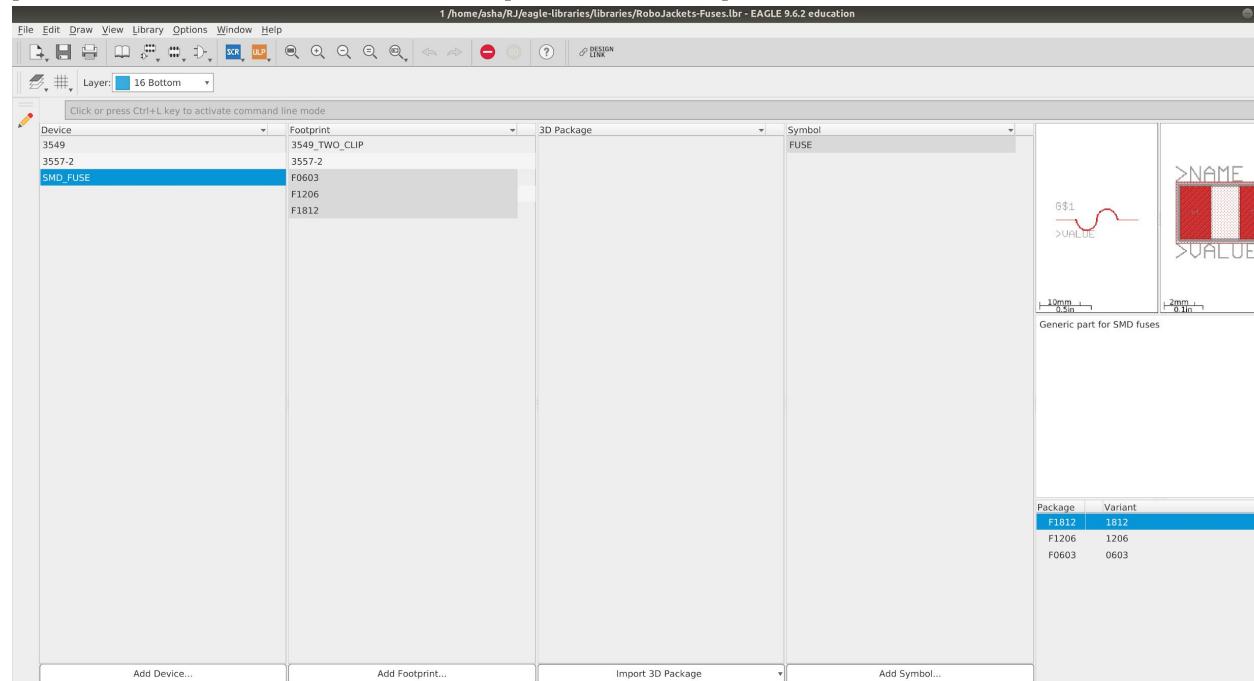


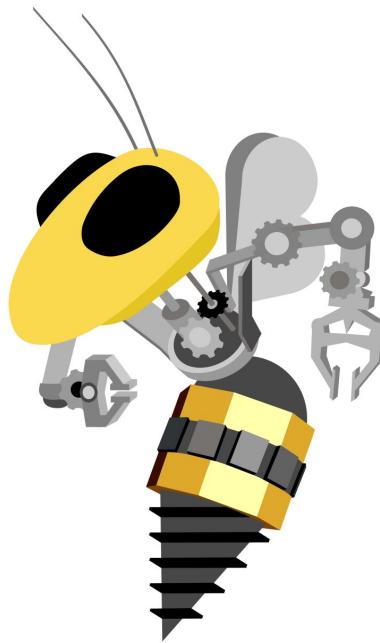
Example Library



Example Library

A component with multiple footprints





Lab!

Setup + Making a Part in
EAGLE

Installing Software

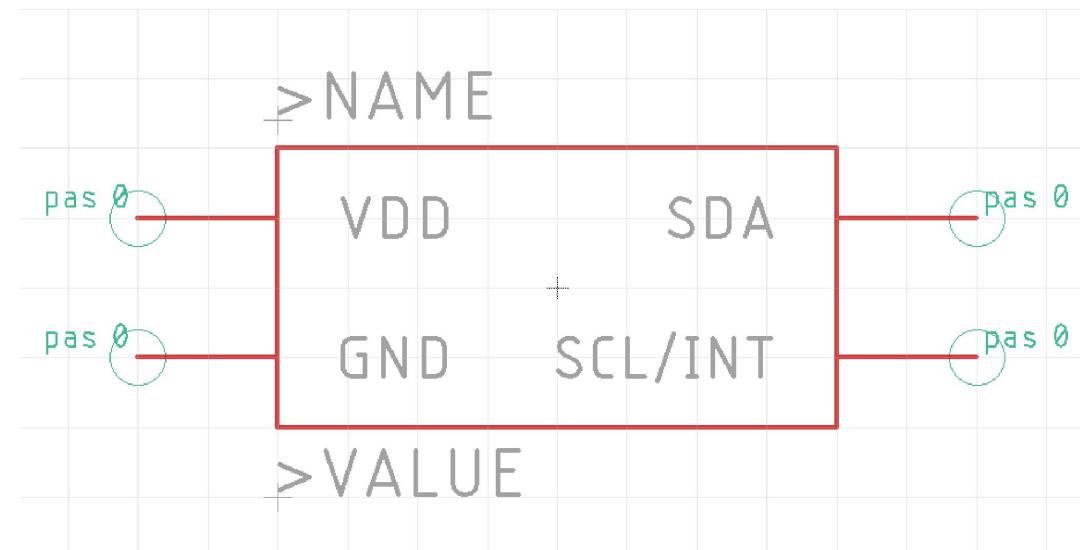
- EAGLE
 - <https://www.autodesk.com/products/eagle/free-download>
 - Requires Autodesk account registered with GA Tech email
 - <https://www.autodesk.com/education/edu-software/overview?sorting=featured&page=1>

Adding Repositories

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

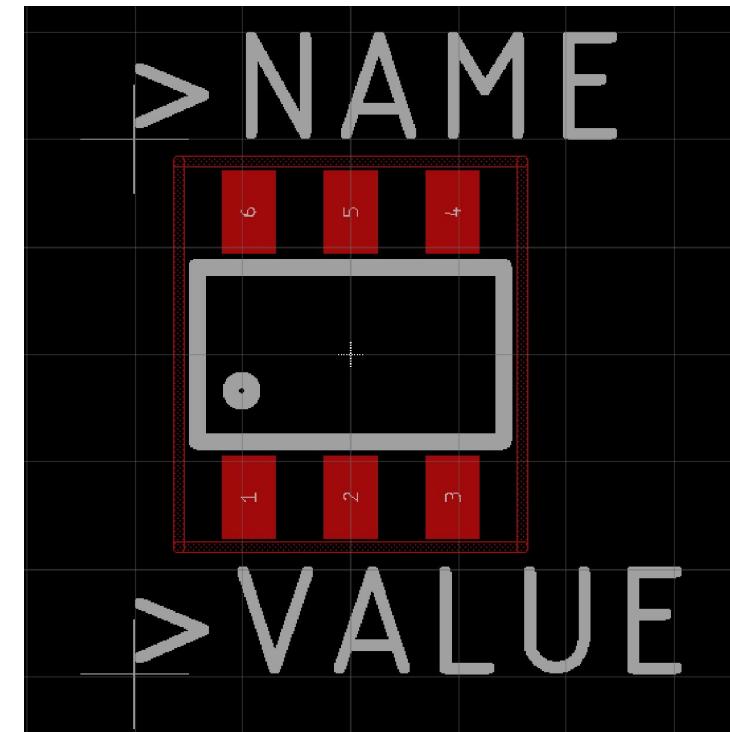
Making a Part - Symbol

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



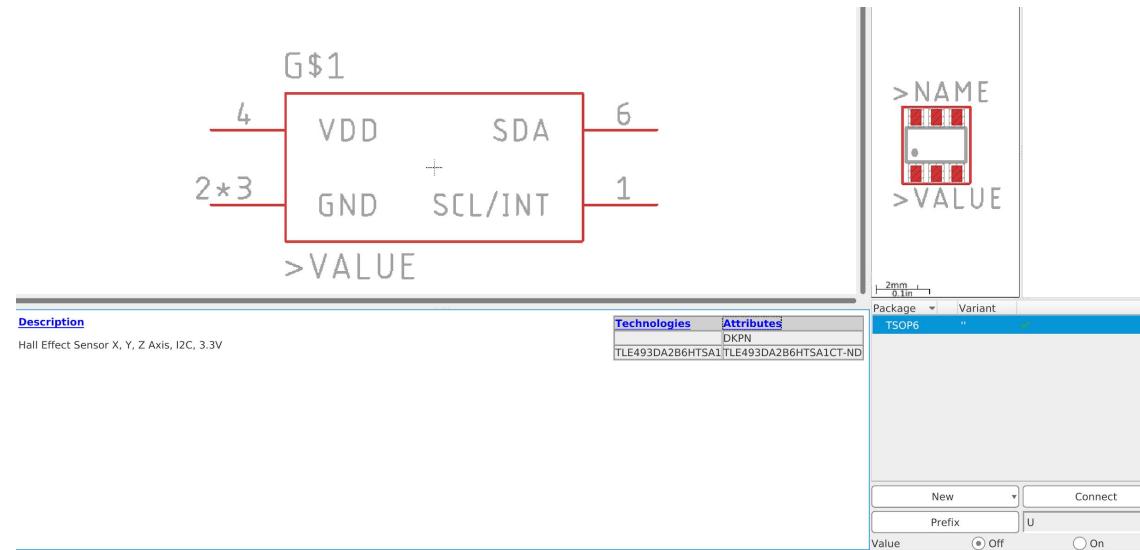
Making a Part - Footprint

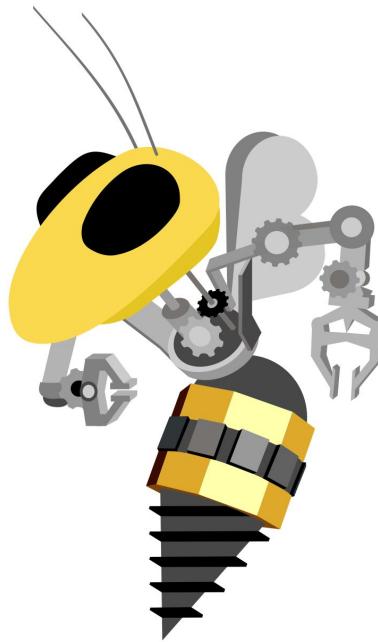
- Create a physical representation of the part by reading the package outline details in the datasheet and then placing correctly sized pads/drills at the appropriate location



Making a Part - Device

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





Thank You!

Any Questions?