



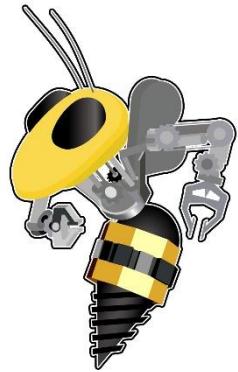
**RoboJackets**

BATTLEBOTS - OUTREACH - IGVC - ROBOCUP - IARRC

# Electrical Training

Week 4: Eagle CAD

# Agenda



- Eagle
- Eagle
- More Eagle

# Install Eagle



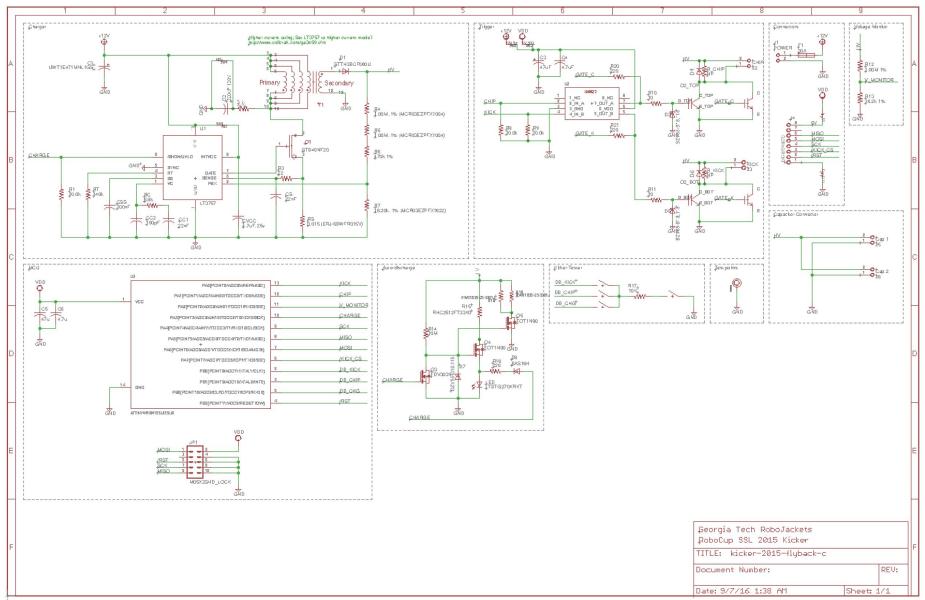
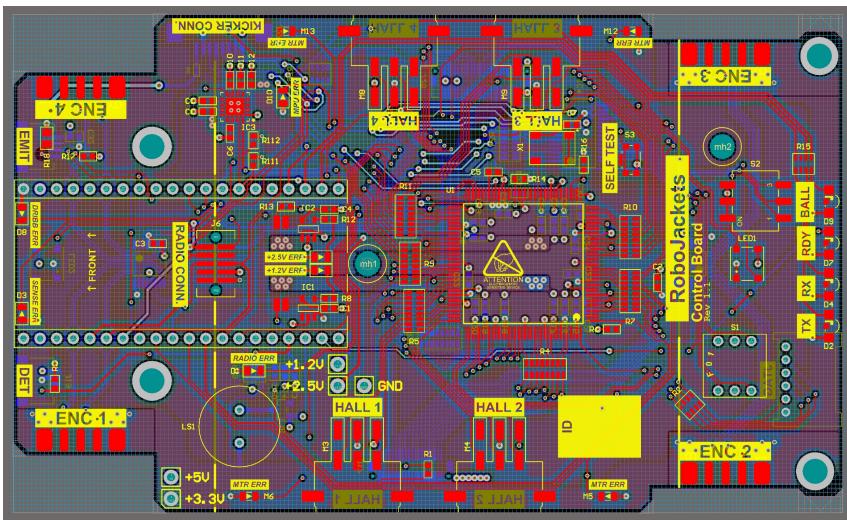
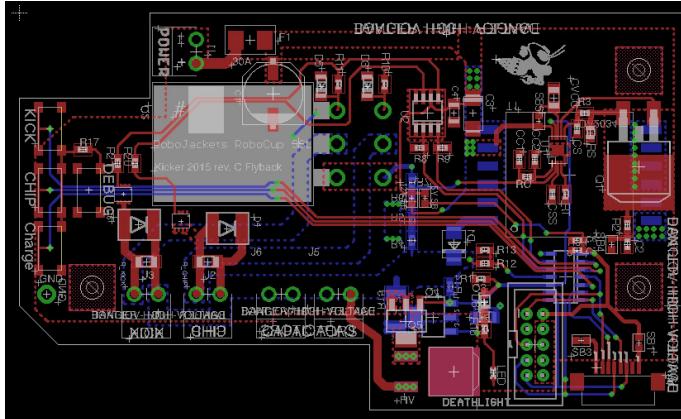
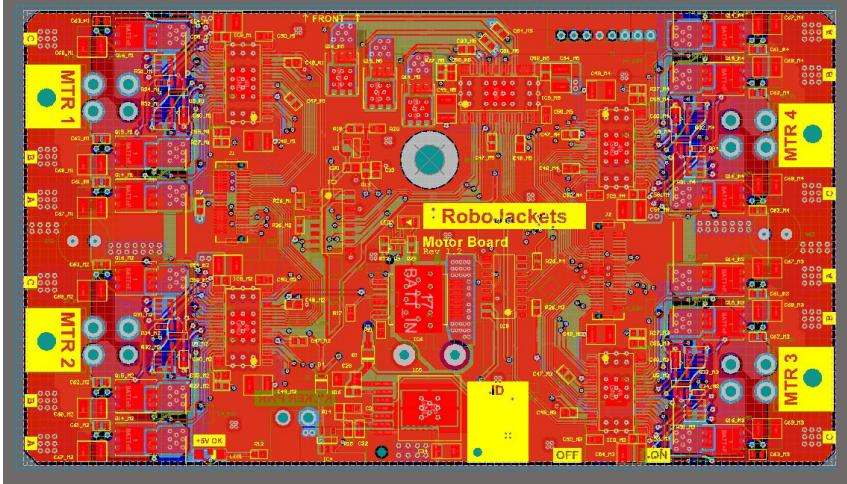
- Download at: <https://cadsoft.io/>
- Eagle License: Run as Freeware
- To request educational Eagle license (FREE)
  - <https://cadsoft.io/pricing/>
  - Less limitations compared to free version
  - Not needed for this training

# Introduction



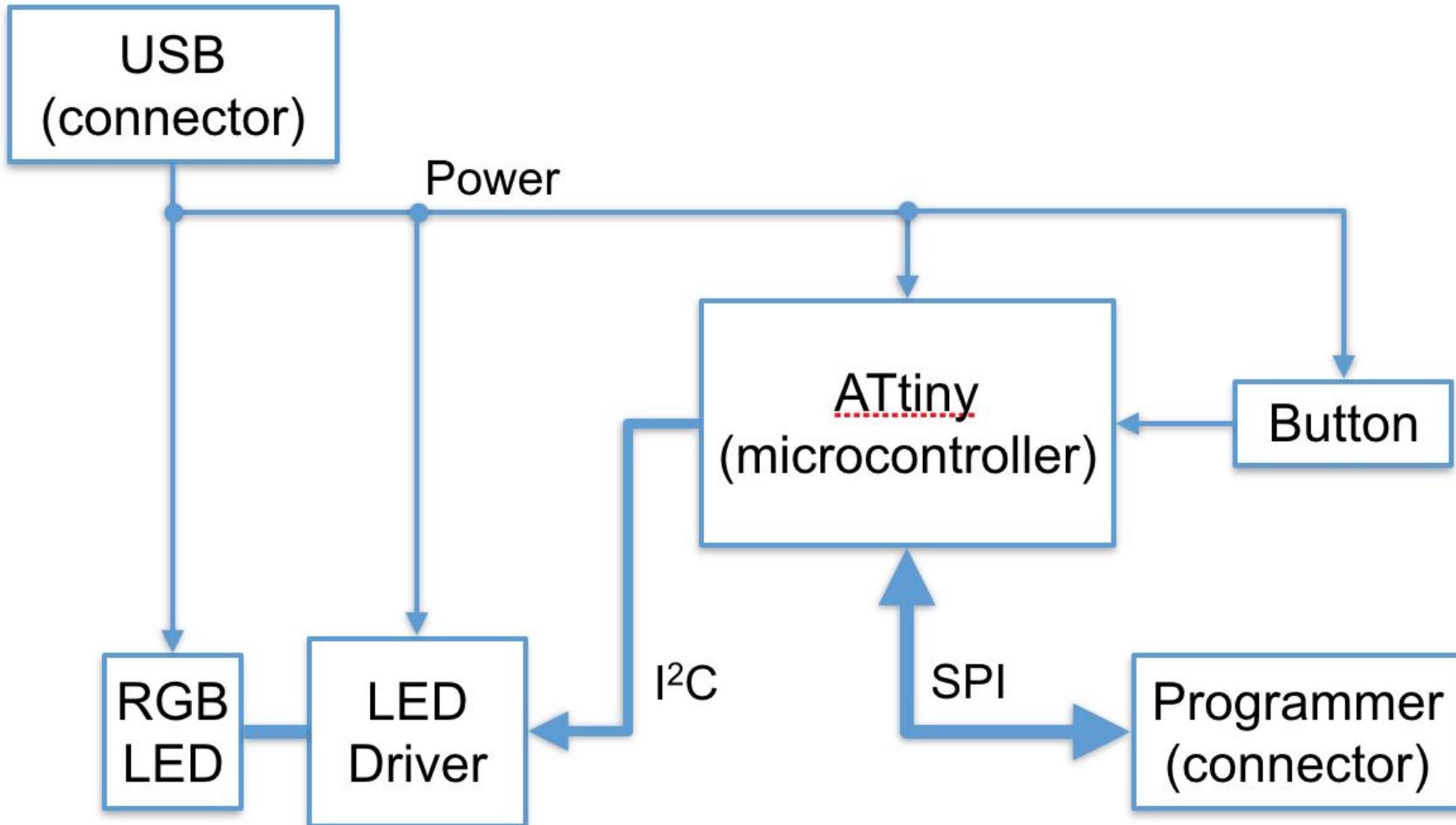
- Part Libraries
  - Contain components to be added to schematic and board
- Eagle Schematic
  - Electrical schematic of all parts and connections that will go on the board
- Eagle Board Layout
  - Laying out parts from the schematic and routing connections for what will become a Printed Circuit Board (PCB)

# Some of our designs

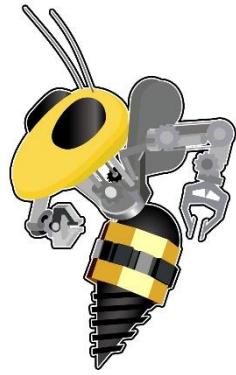


Berkeley Tech RoboJackets  
RoboCup SSL 2015 Kickerv  
TTL\_Ez\_kicker\_2015+flyback.c  
Document Number: REV1  
Date: 5/7/16 1:38 AM  
Sheet 1/1

# What You Are Making



# Libraries

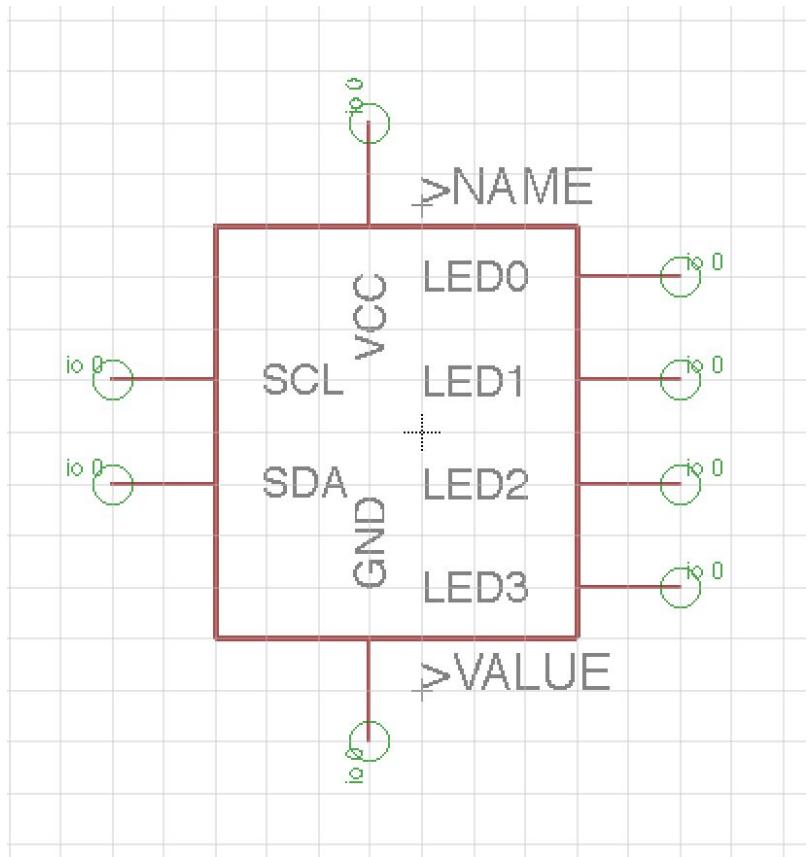


- Store information about parts used in schematic and board layout
- Eagle comes with some but you often need to make your own
- Symbol
  - What is seen in schematic
- Package
  - Footprint used for board layout
- Device
  - Brings symbol(s) and package(s) together

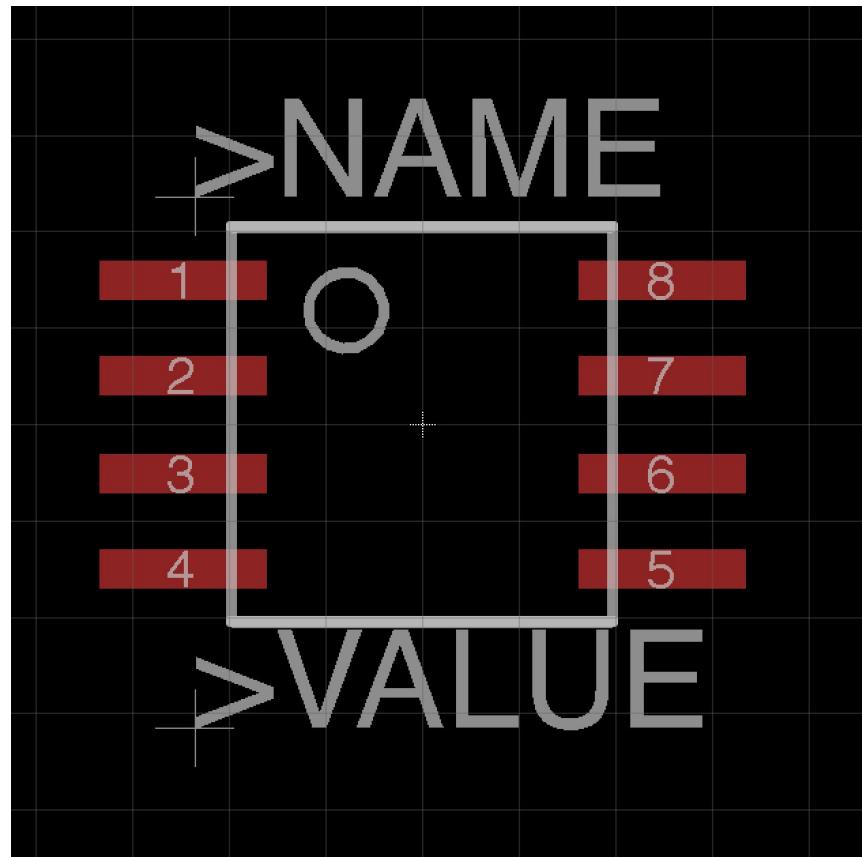
# Libraries



Symbol



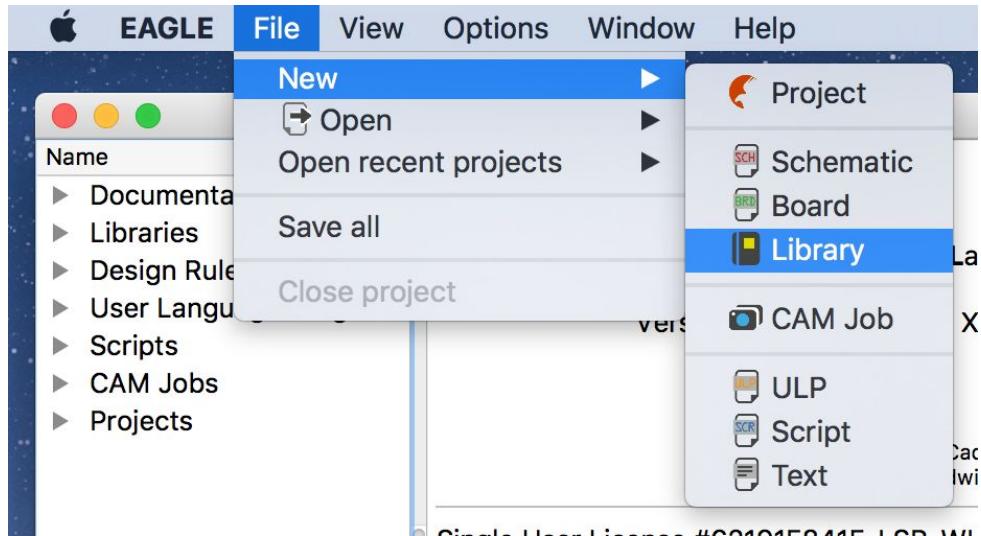
Package



# Create Library



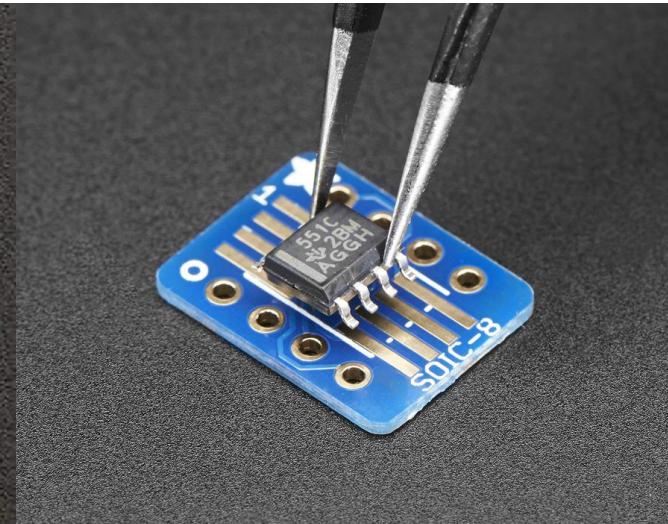
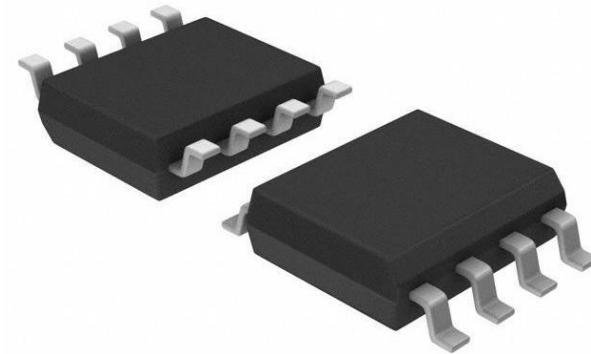
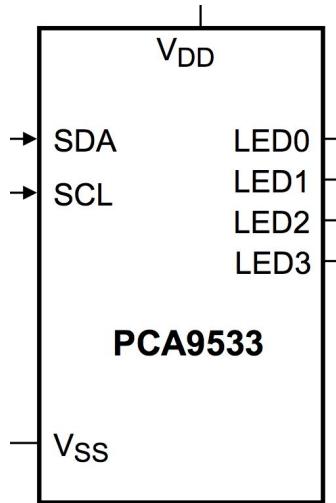
- File>New>Library
- Save to new directory for project
  - Default save location is where Eagle libraries are stored, use new folder in your Documents
- Eagle automatically searches its lbr folder
  - Libraries saved outside of this will need to be added in (we will do this later)



# Library Part

## What You Are Making

- PCA9533
- LED driver
  - Used to power LEDs in the RGB LED
- SOIC-8 Package
- Footprint shown on PCB:

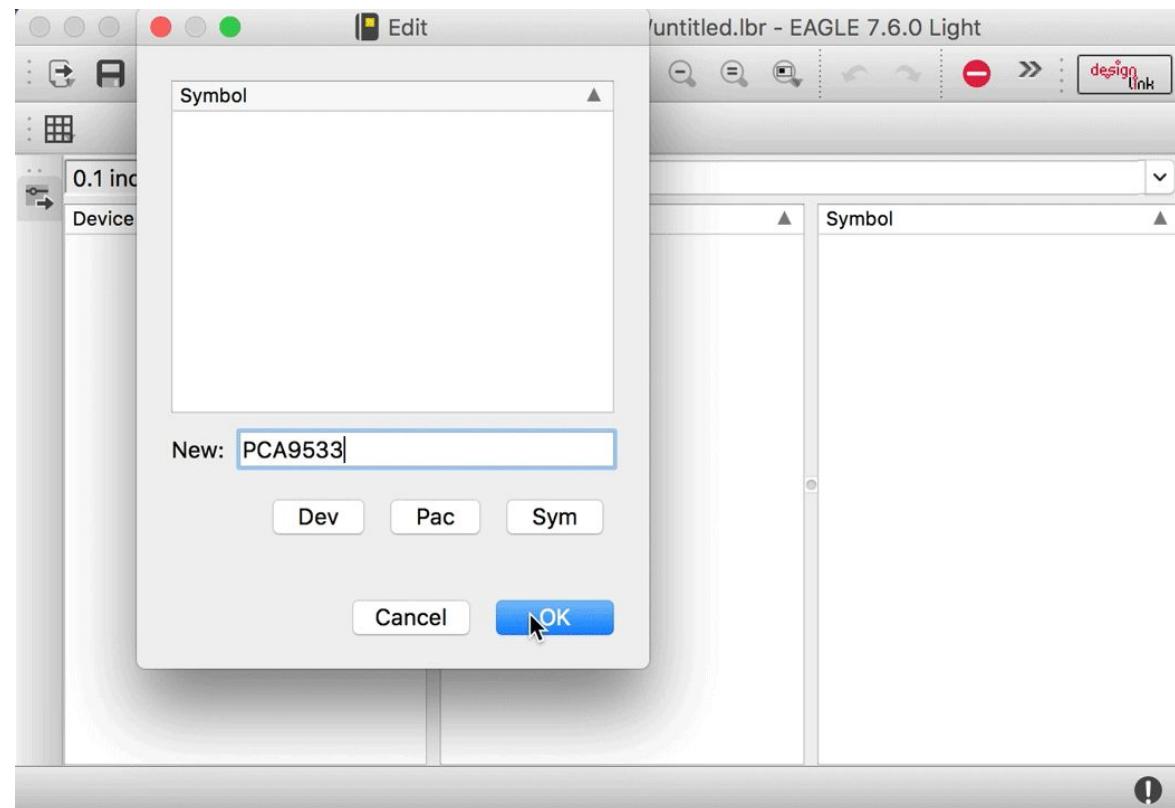


# Library Part: Symbol

## Create Symbol



- Create new symbol: (Library>Symbol)
- Name symbol with the part number
- Will be presented with grid to draw symbol

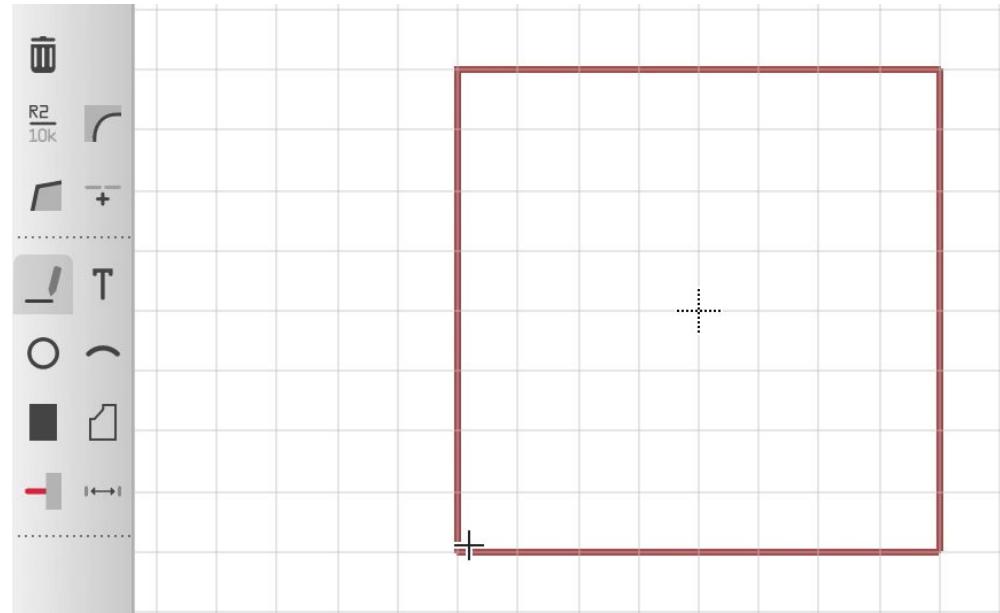


# Library Part: Symbol

Draw Part Outline



- Draw box by using wire button
- This aspect is purely visual so any size/shape is okay

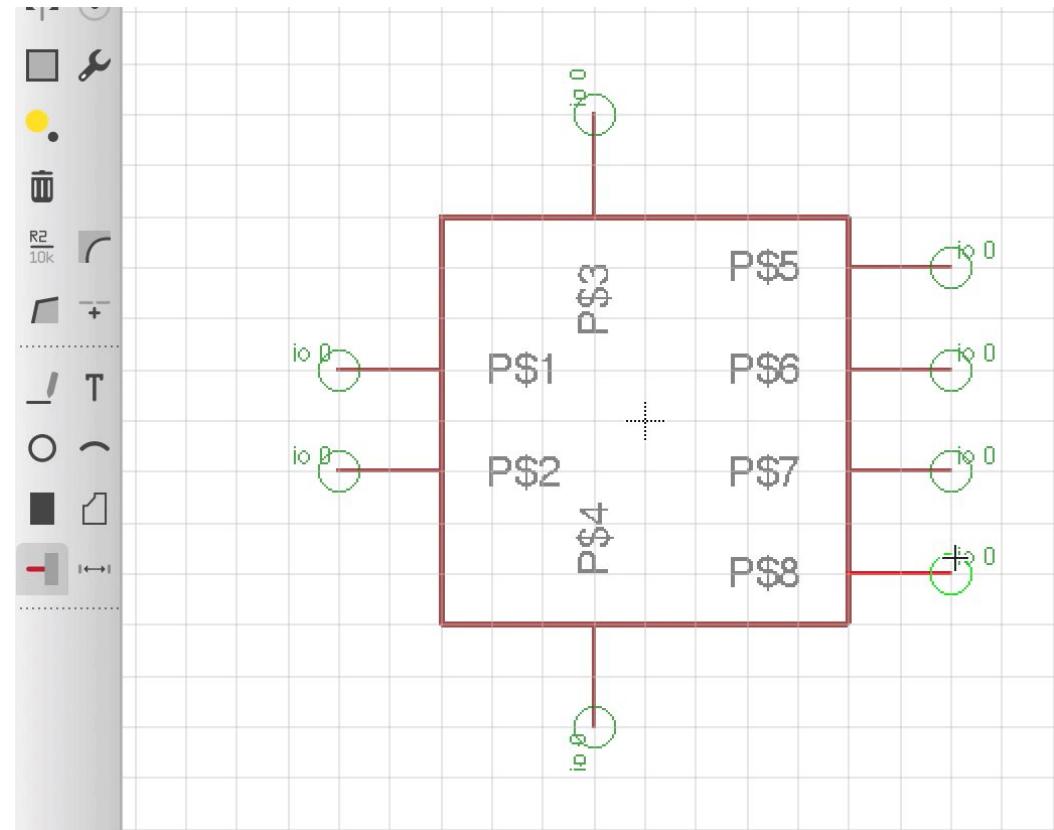


# Library Part: Symbol



## Add Pins

- Add pins to the box (pin button)
- This is what wires are connected to in schematic
- (Tip) Right click while placing to rotate

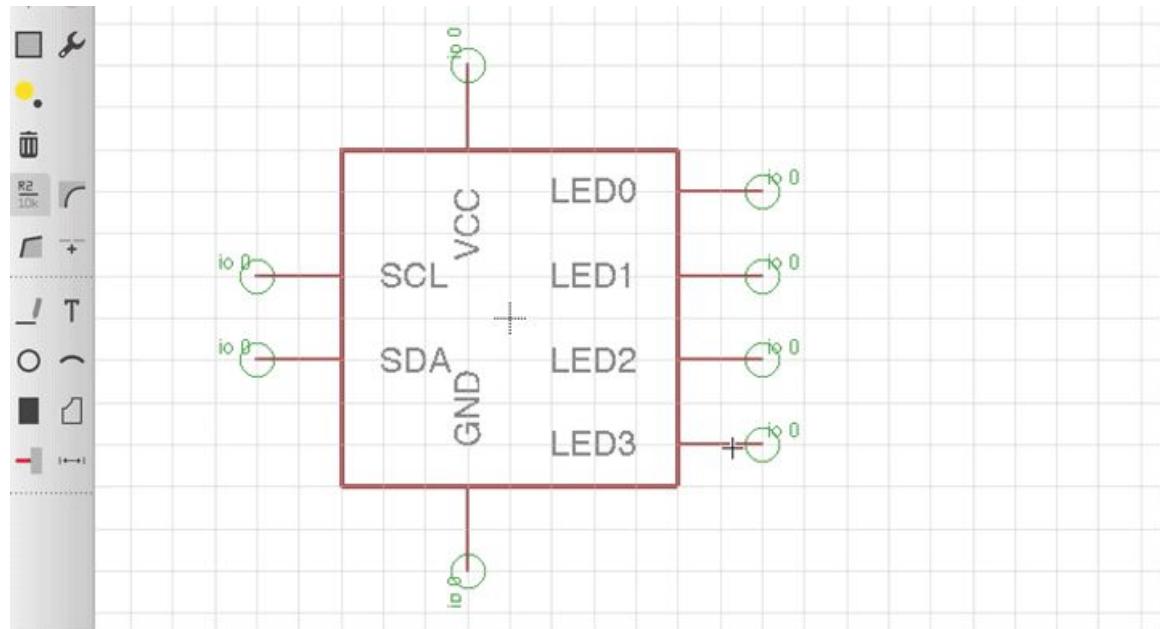


# Library Part: Symbol



## Name Pins

- Name pins corresponding to datasheet
- Name button then click on pin

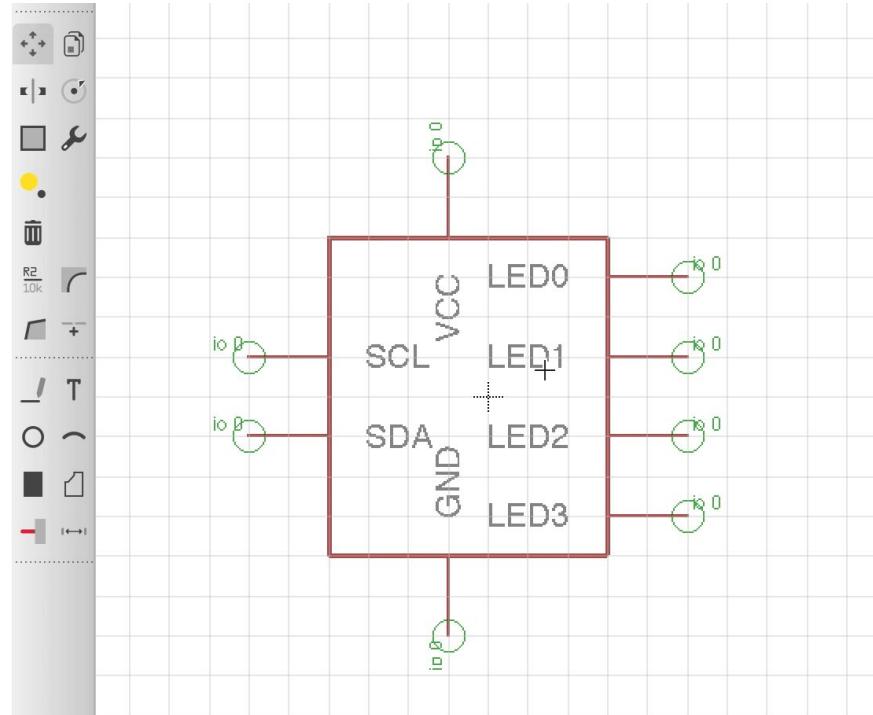


# Library Part: Symbol

Make Pretty



- Adjust part outline (box) so all pins fit and names can be seen



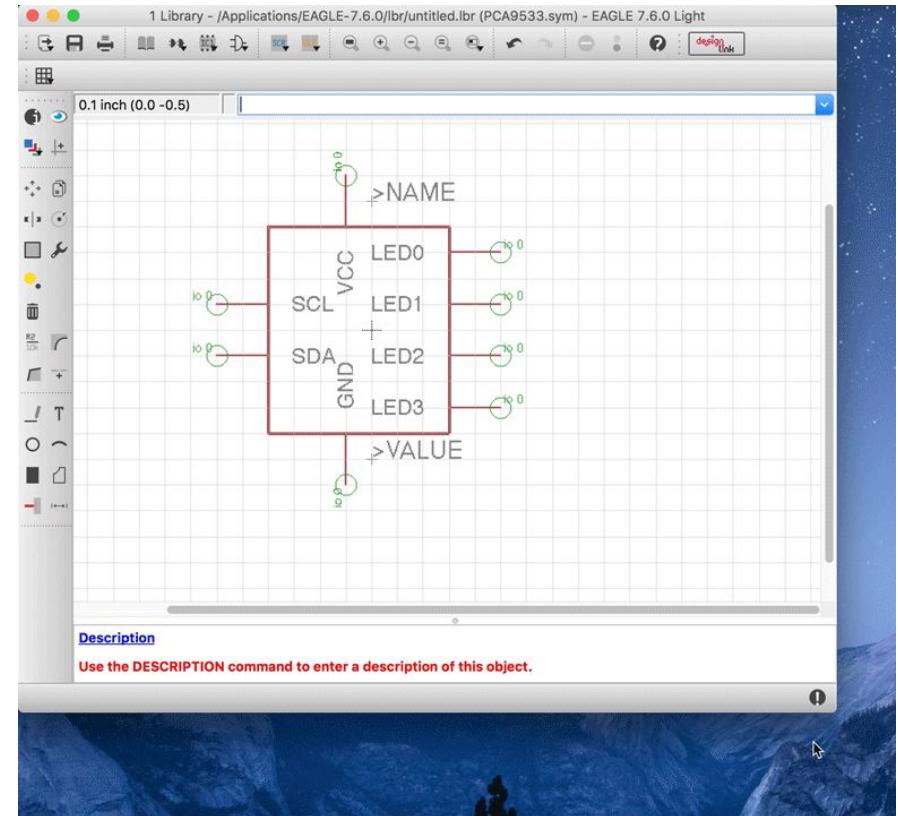
- (Tip) To move a group: Hit group button, select what you want to move, right click>Move:Group

# Library Part: Symbol

Add Name and Value



- Hit text button
  - >VALUE and >NAME are keywords that will be automatically filled out
- Change layer to Names and Values respectively
  - Once placed, right click>Properties>Layer



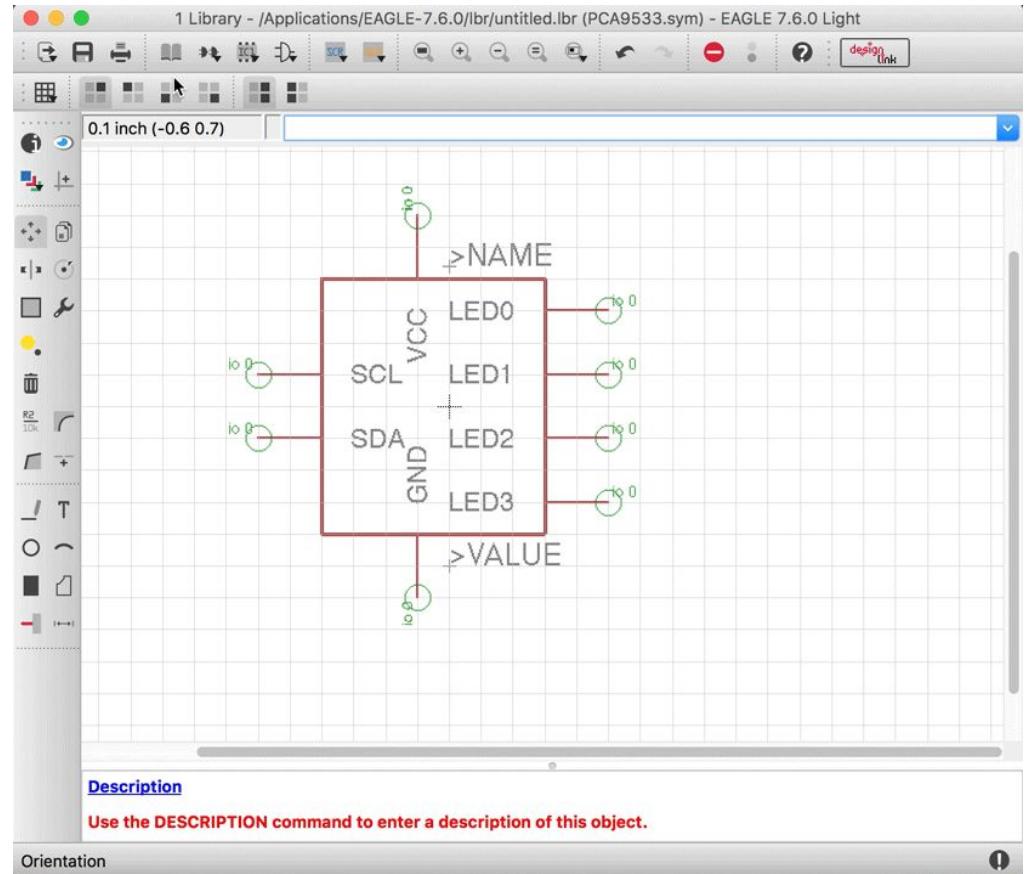
- (Tip) Objects are clickable and selectable at grey “+”

# Library Part: Symbol

## Complete



- Should look similar to this:
- Once done click on Table of Contents button (at top)

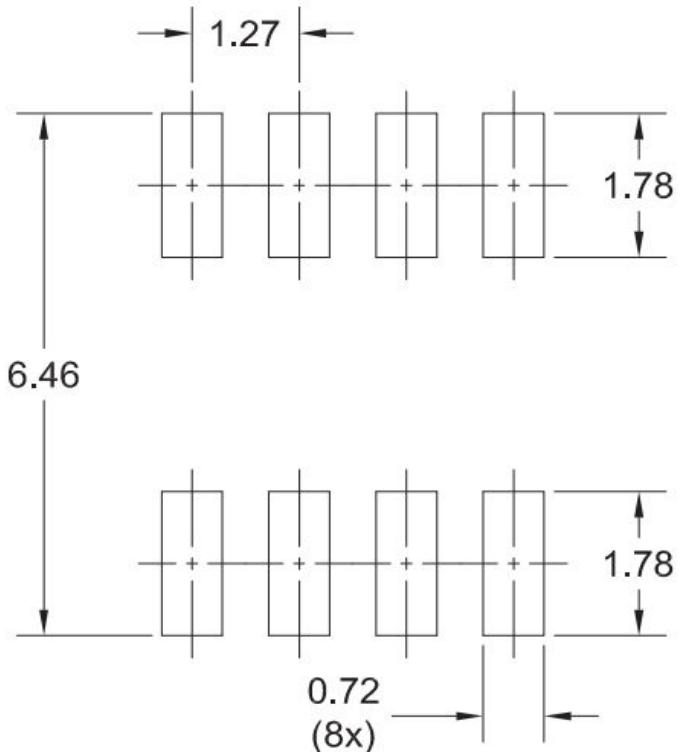


# Library Part: Package

## Starting Information



- Typically footprint for package found in datasheet
  - Contains all measurements needed to create the package
- This package will go on the PCB and is what the component is soldered to

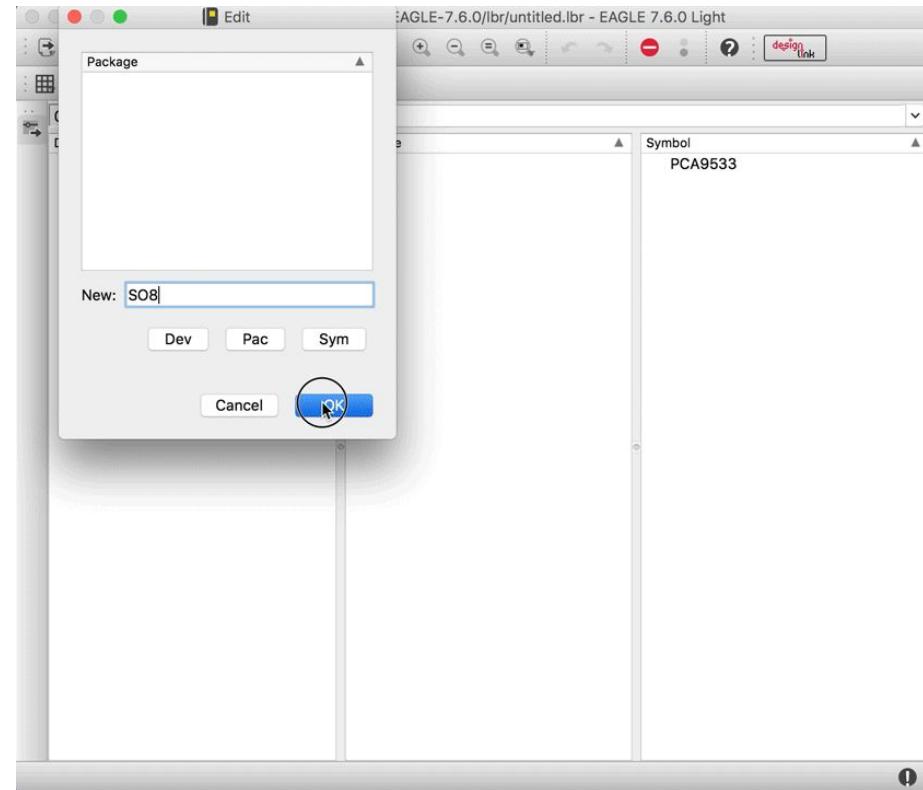


# Library Part: Package



## Create Package

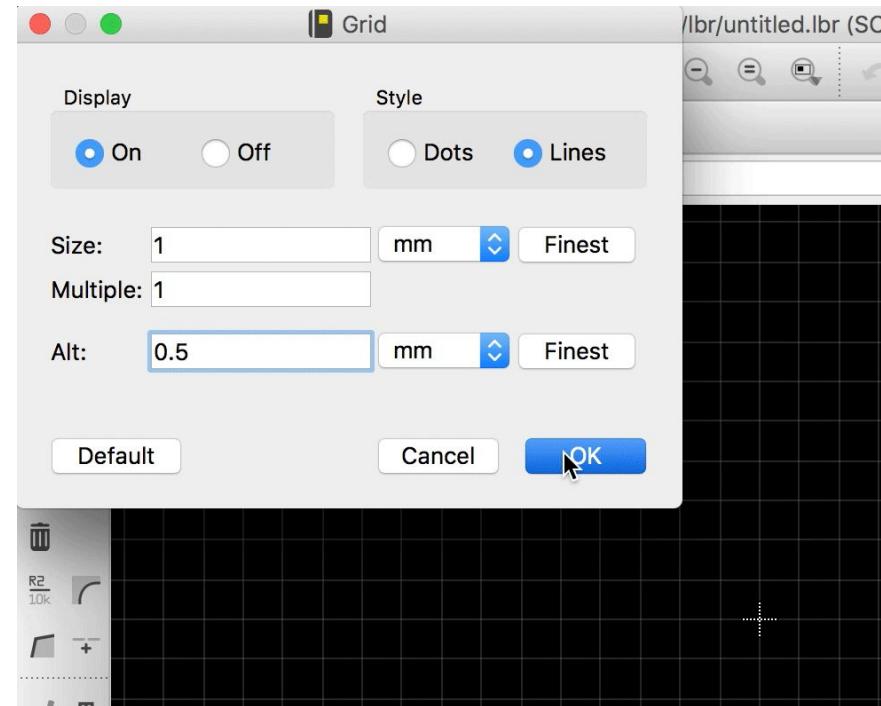
- Create new package: (Library>Package) or click: 
- Name package with the package name
- Common for multiple components to have the same package



# Grid Info



- Grid button for settings
- Change grid size
  - Size can be whatever is helpful to you
- Units: use same units given in datasheet footprint
  - Values for size and position used later in these units



# Library Part: Package

## Place Pads



- Place first pad
  - Pad button for through hole
  - SMD button for surface mount (what we are using)
- Resize pad
  - Right click, properties then Smd Size
  - Size given from datasheet

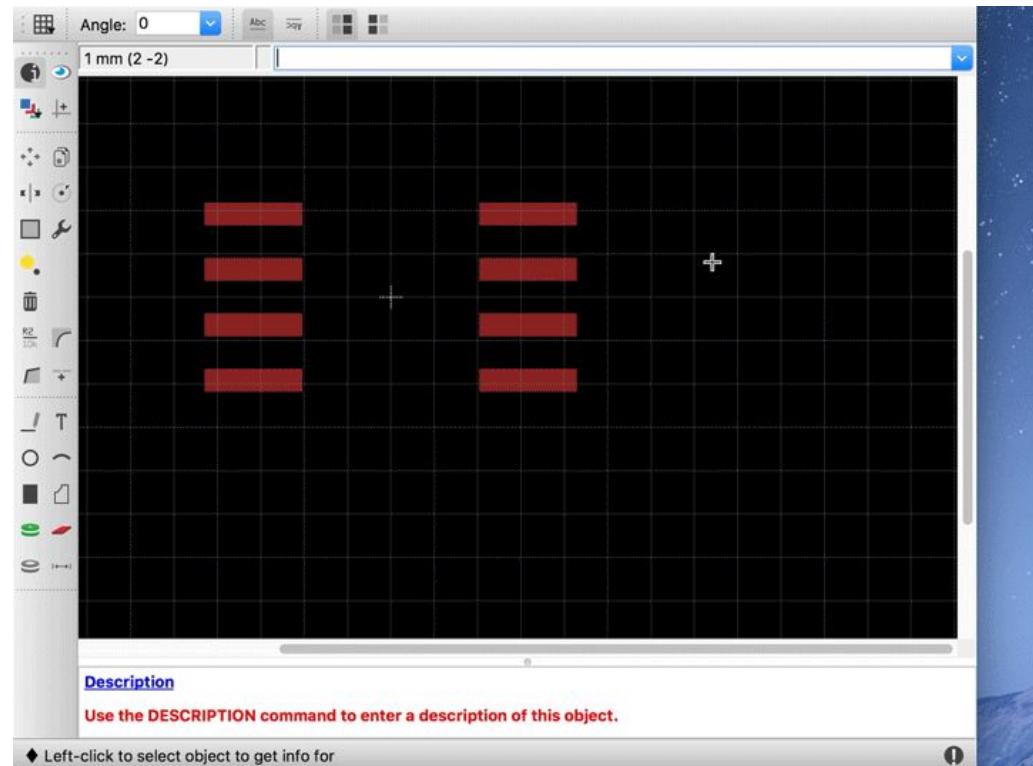


# Library Part: Package

## Move and Resize Pads



- Copy tool to place pads in approximate position
- Use values given in datasheet footprint to calculate position (to the center) of each pad
- Use positions so (0,0) is in center of the component

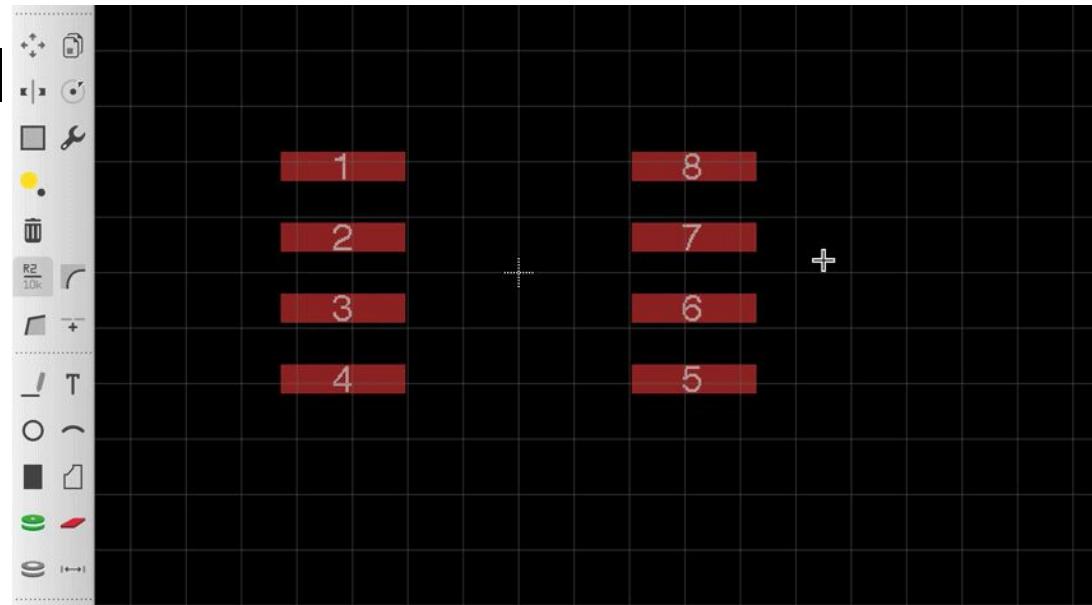


# Library Part: Package

## Rename Pads



- Use rename tool to name pads by number
- Pad numbers will be given by datasheet



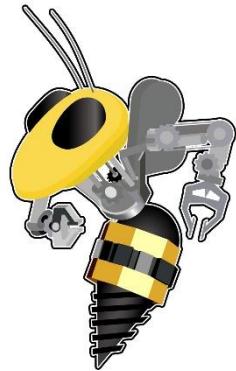
# Layers



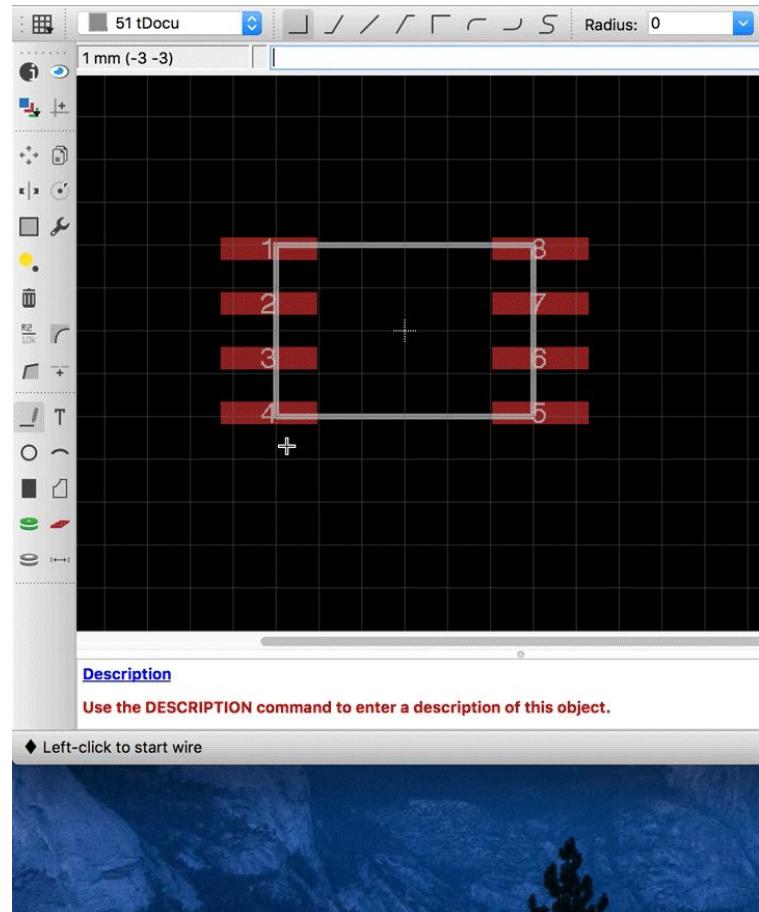
Color	Layer Name	Layer Num	Layer Purpose
Red	Top	1	Top layer of copper
Blue	Bottom	16	Bottom layer of copper
Green	Pads	17	Through-hole pads (copper on top and bottom)
Green	Vias	18	Vias to route signal between layers (copper on top and bottom)
Grey	Dimension	20	Outline of the board
	tPlace	21	Silkscreen for top
Yellow	bPlace	22	Silkscreen for bottom
Gold	tDocu	51	Top documentation layer (just for reference)

# Library Part: Package

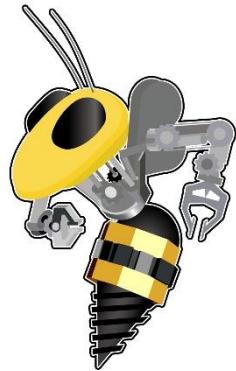
## Add Part Outline



- Wire button, select layer to 51 tDocu, then draw basic part outline
- The tDocu layer is just for reference (to ensure no overlapping parts) so will not appear on printed board

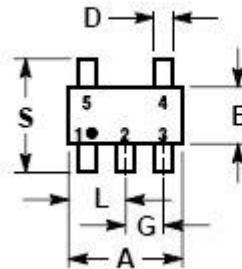


# Library Part: Package

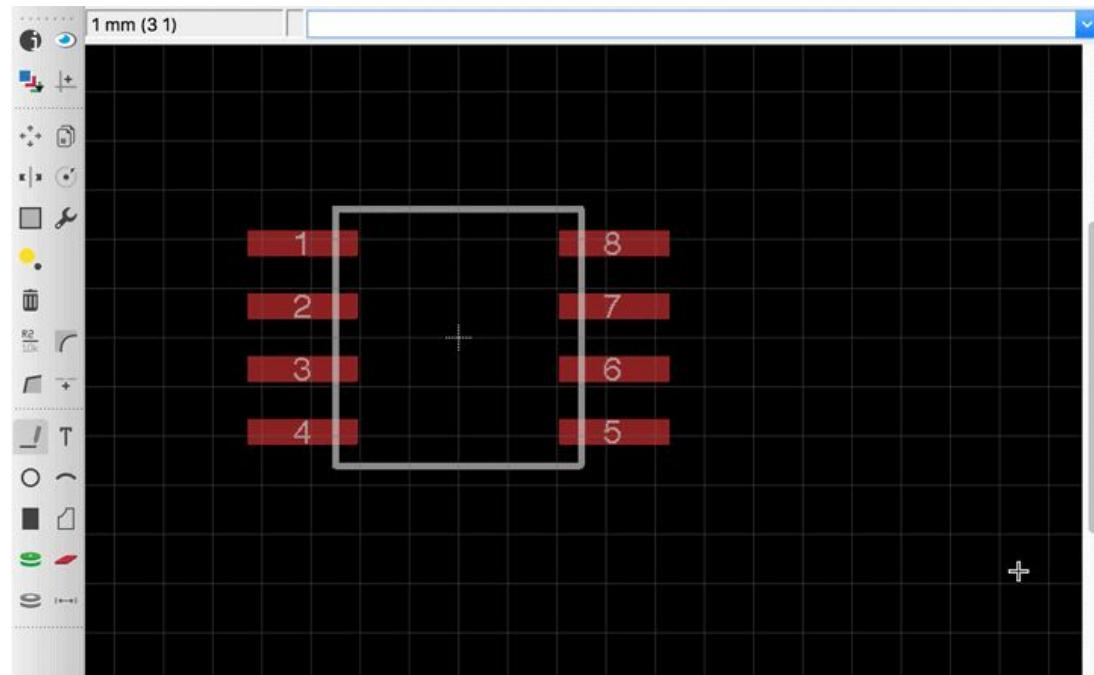


## Adjust Part Outline

- Use info in datasheet to find specific dimensions for outline
- Lines can be manually adjusted by right click>properties

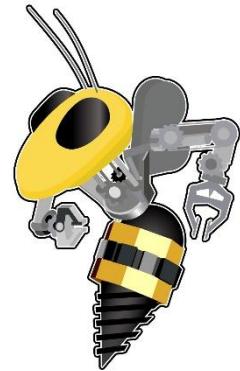


	MILLIMETERS	
DIM	MIN	MAX
A	2.90	3.10
B	1.30	1.70
C	0.90	1.10
D	0.25	0.50
G	0.85	1.05
H	0.013	0.100
J	0.10	0.26
K	0.20	0.80
L	1.25	1.55
M	0	10
S	2.50 <sup>+</sup>	3.00

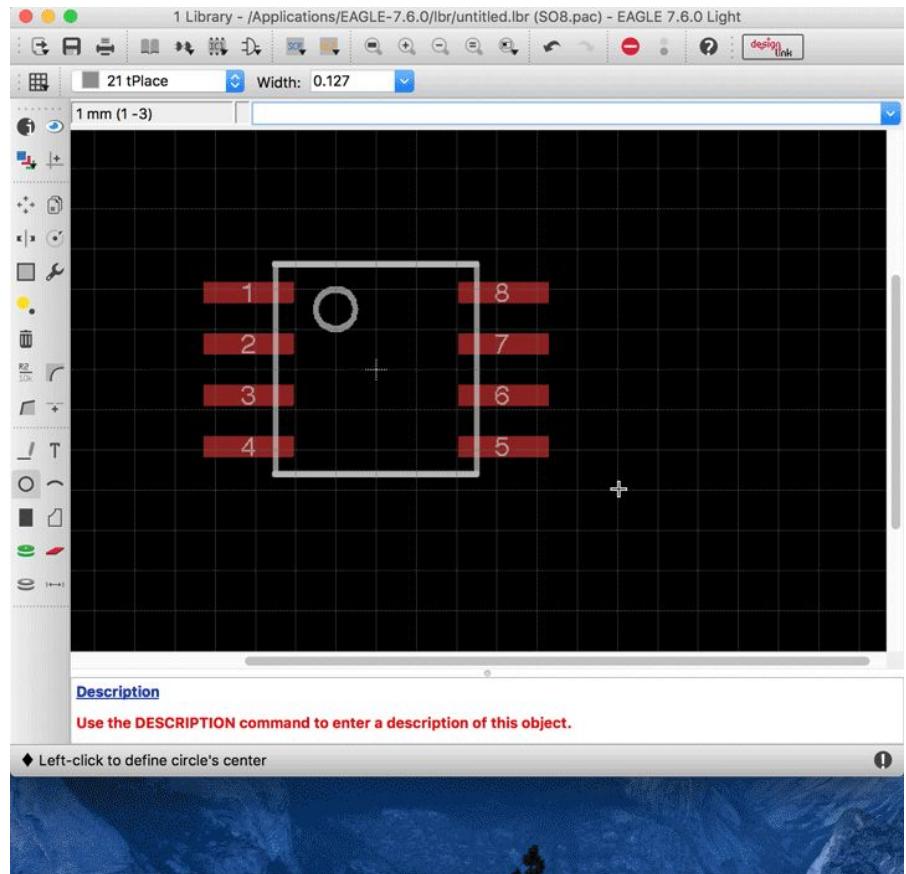


# Library Part: Package

## Add Silkscreen



- Silkscreen gets printed on board and is used to help place components
- Draw lines on layer tPlace as outline that does not overlap with pads
- (Tip) Add dot on pin 1 corner to identify when soldering component

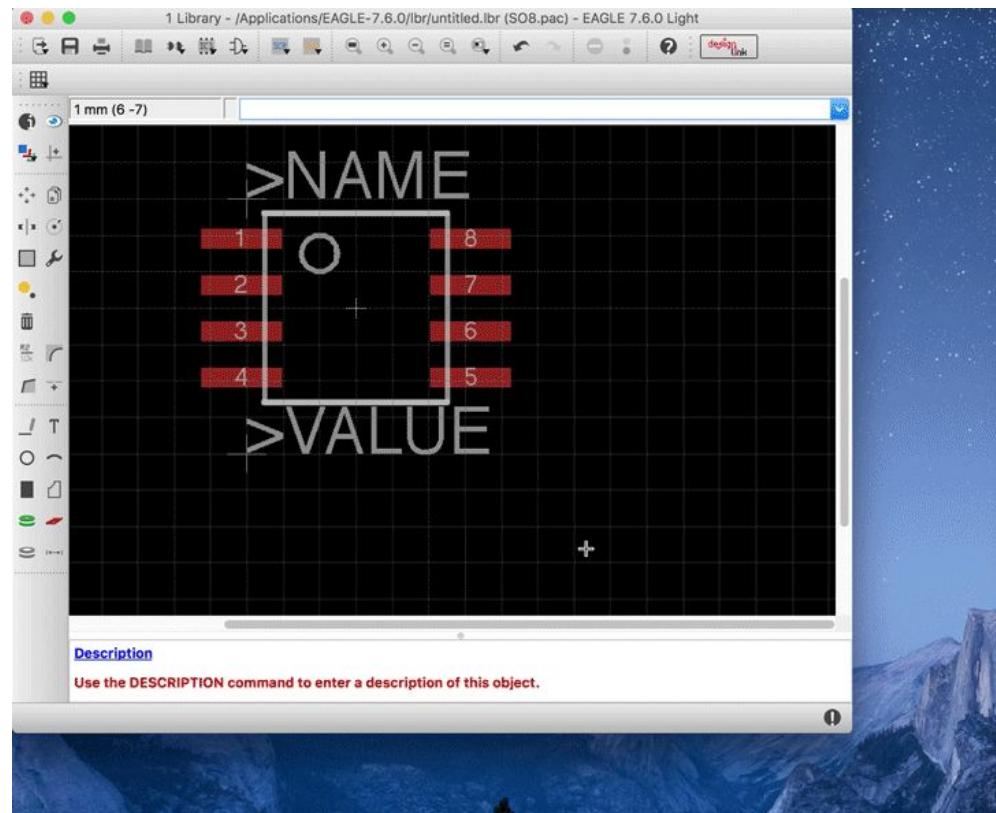


# Library Part: Package

Add Name and Value



- Add >NAME and >VALUE text similar to in symbol
- Adjust layer so they are on tNames and tValues respectively

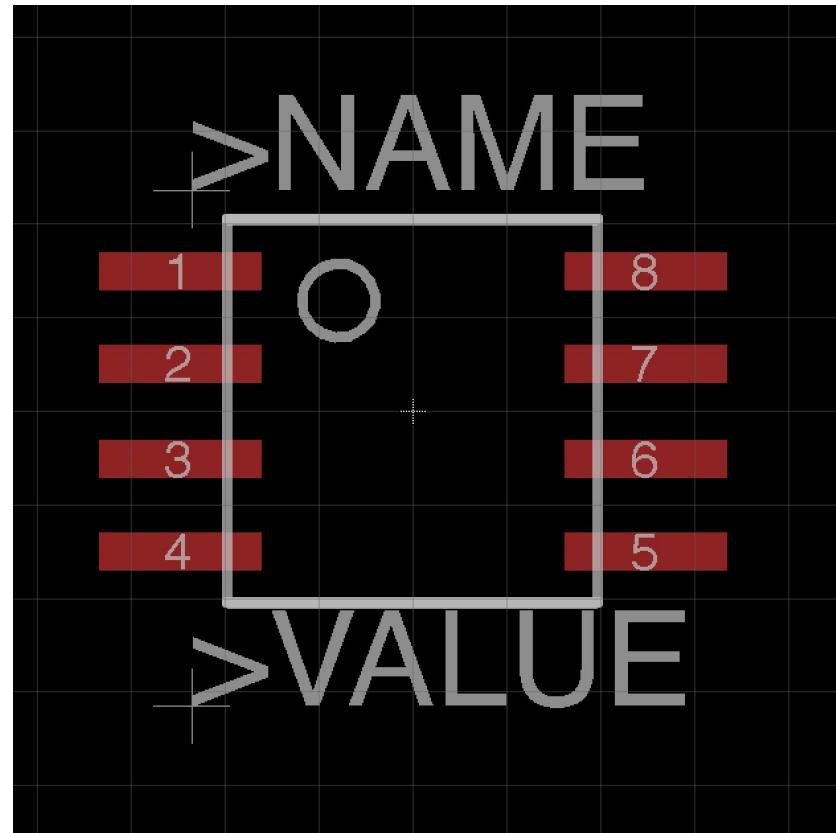


# Library Part: Package

Complete



- Should look something like this:
- Once done click on Table of Contents button



# Library Part: Device



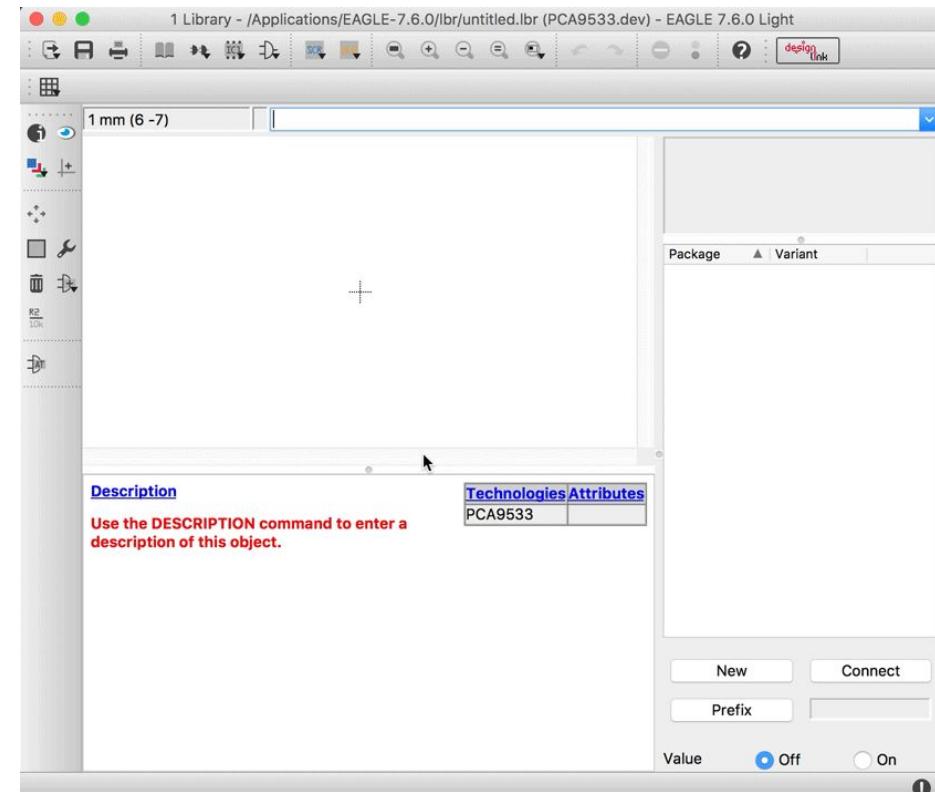
- This is where the symbol and the package come together
- This device is what will be added to schematic

# Library Part: Device

## Create Device



- Create new device: (Library>Device) or click: 
- Name device what general part number is
  - Not package specific because multiple packages can be added to one device

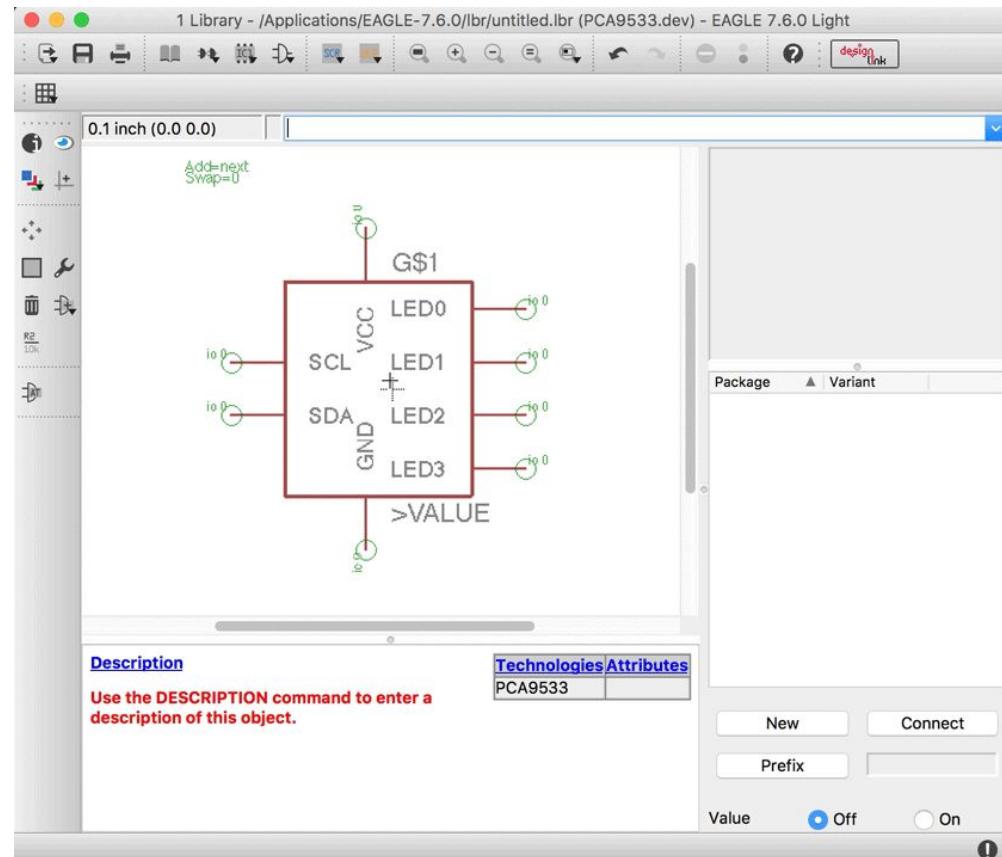


# Library Part: Device

## Add Symbol



- Hit add button then select symbol you previously created
- Place symbol then hit esc twice to get rid of part window

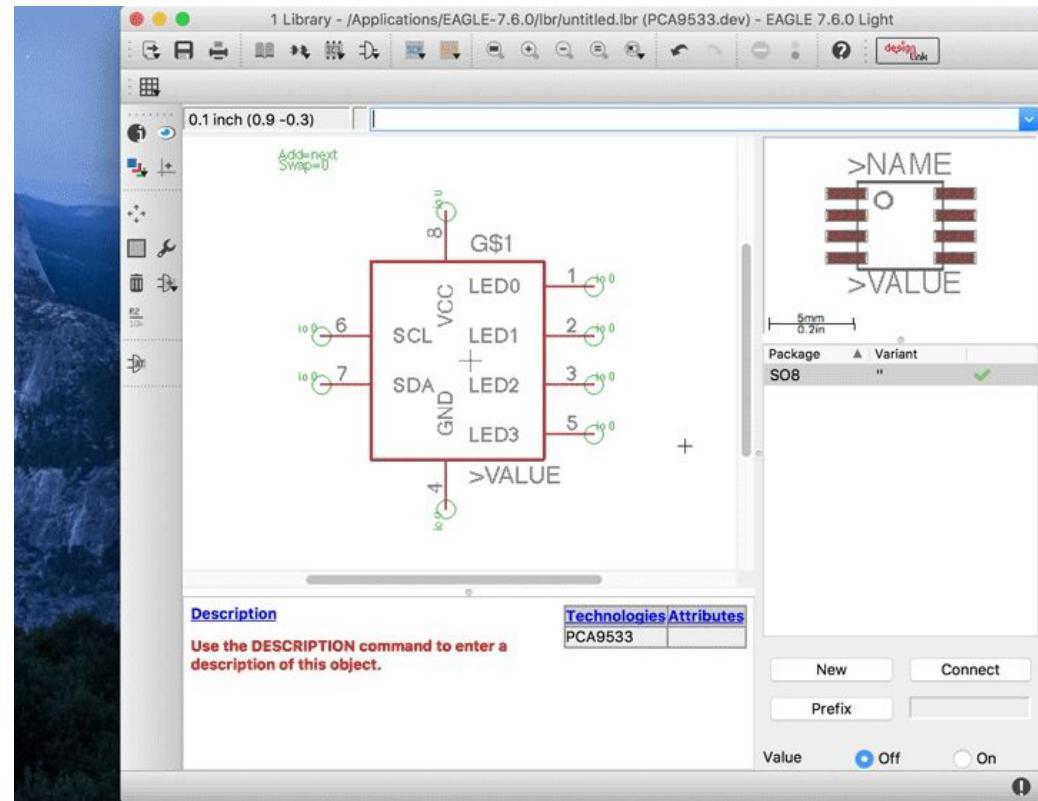


# Library Part: Device

## Add Package



- Hit new button in package area then select package previously created
- (Optional) Variant names can be added when adding multiple packages to choose from for one device

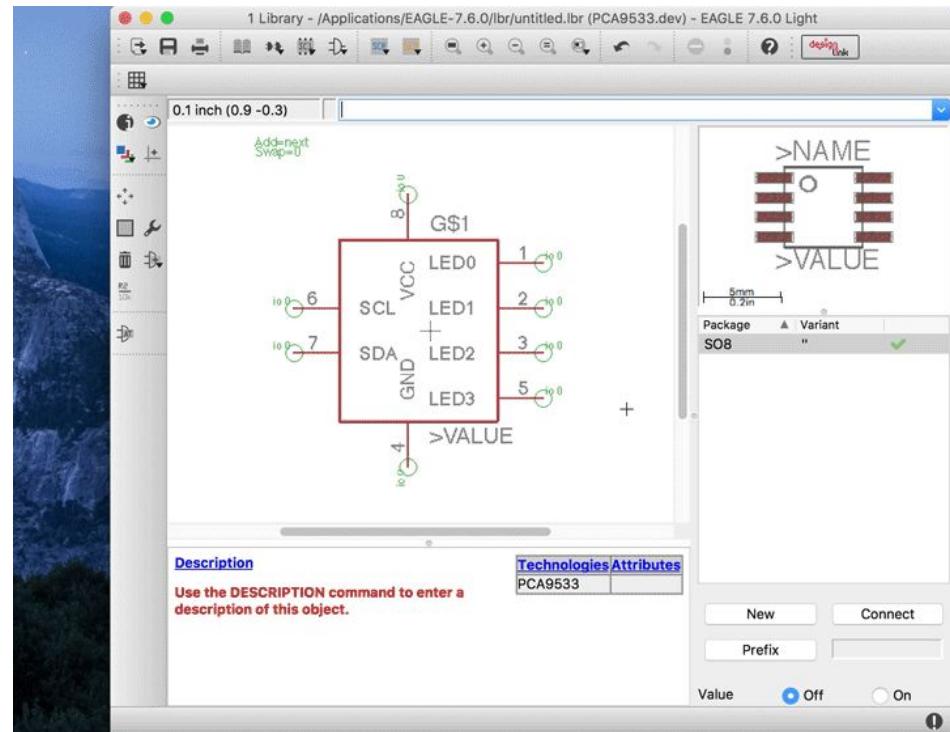


# Library Part: Device

## Connect Pins



- Hit the connect button in the package area
- Use datasheet to see what pin names connect to what pad numbers
- Select pin name and corresponding pad number then hit connect, repeat for each one



# Library Part



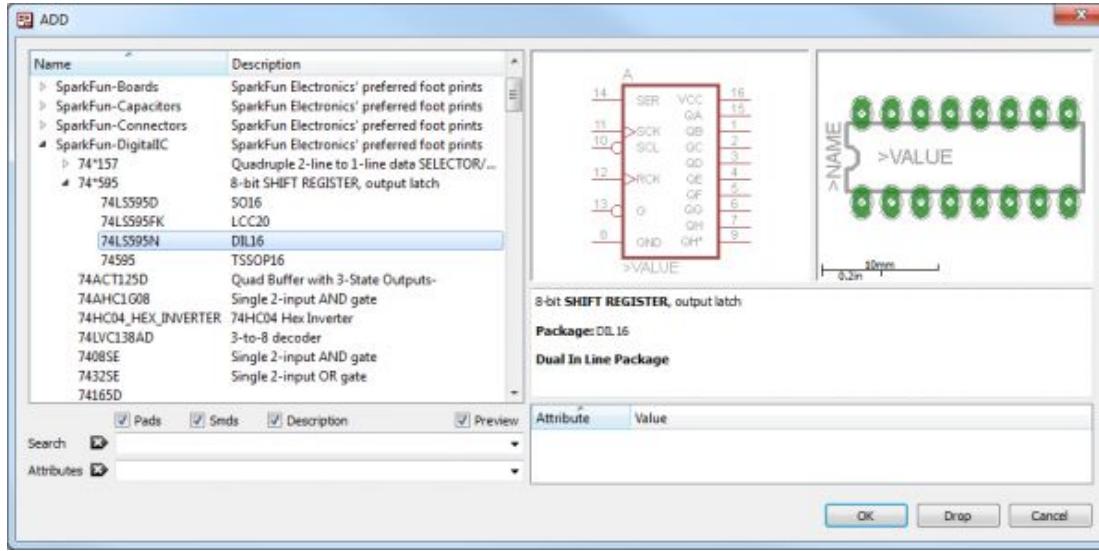
- Done with library part
- Ready to be used in schematic
- More parts can be added to this library
- Parts that share packages only need package created once, check if it exists first

# Schematics



- Schematics are how we organize the board to see the parts used and the connections between them
- Create new schematic with File>New>Schematic

# Schematic: Adding Parts



- Components can be added from libraries
  - To use the library we just created Library>Use then select library
    - Repeat for library provided containing training parts
  - Add components by hitting add button to show all libraries then select component

# Schematic: Adding Parts



## What to Add

- Add a frame to keep schematic organized
- Add components needed for schematic
  - In our case: part from library you just created and one of each device in training library provided
- Power and ground parts will be added later when wiring

# Schematic: Adding Parts

## Adding Resistor/Capacitors



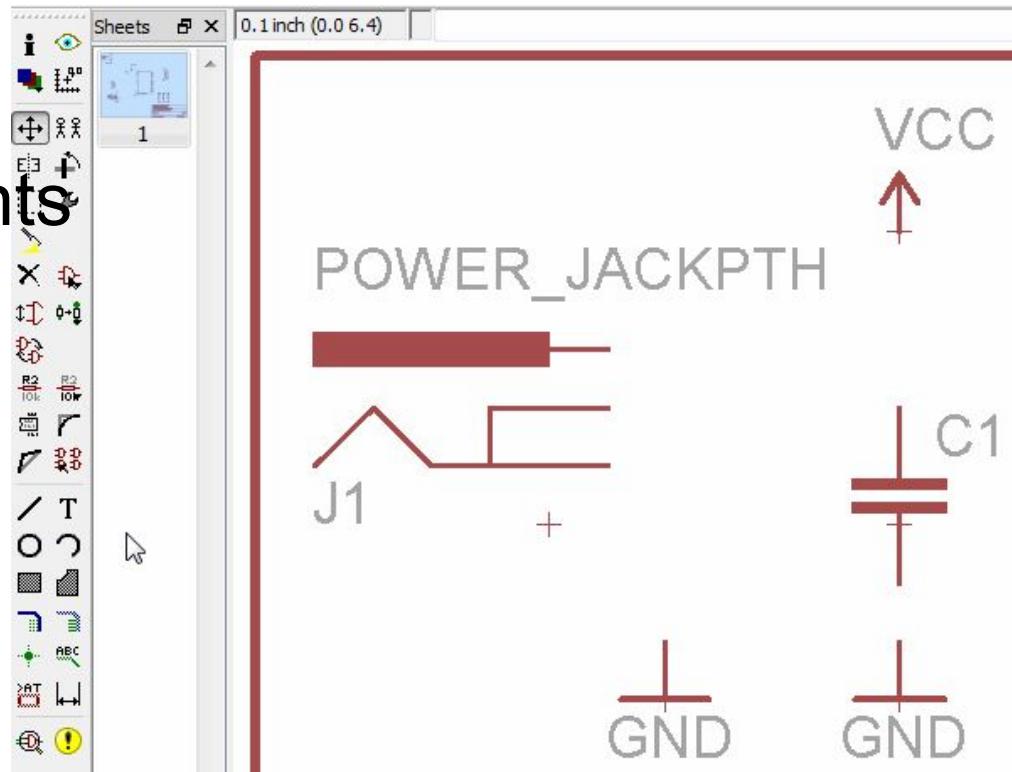
- Resistors and Capacitors are found in default Eagle libraries (rcl.lib)
  - Search for resistor and go to the resistor library
  - Go to the R-US section
  - We are using package size 0603 for this so find surface mount resistor with package size 0603 (labeled R-US\_R0603)
- Capacitors in resistor library, C-US section
  - Most we are using are C0603
  - One capacitor with size C1206
- Change value using value tool (no effect on board, only used as information)

# Schematic: Wiring



- Use the net tool to draw connections between components

This →  **NET**  
Not This →  **WIRE**



# Schematic: Wiring Organization



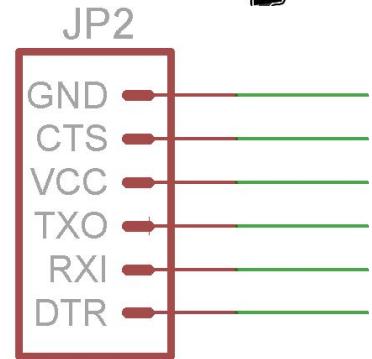
- Components only need to be placed in a way to be able to read easily
- Sometimes wiring can make the schematic messy and difficult to read
  - We can separate these wires and as long as they are named the same, they are considered connected

# Schematic: Wiring

## Connections Without Wiring



- For example: add short nets connected to only one pin
- Use the name tool to click on a net and name it something corresponding to the pin
- Repeat on the pin you want to connect it to (using the same name)
- Now that the nets have the same name, they are considered connected

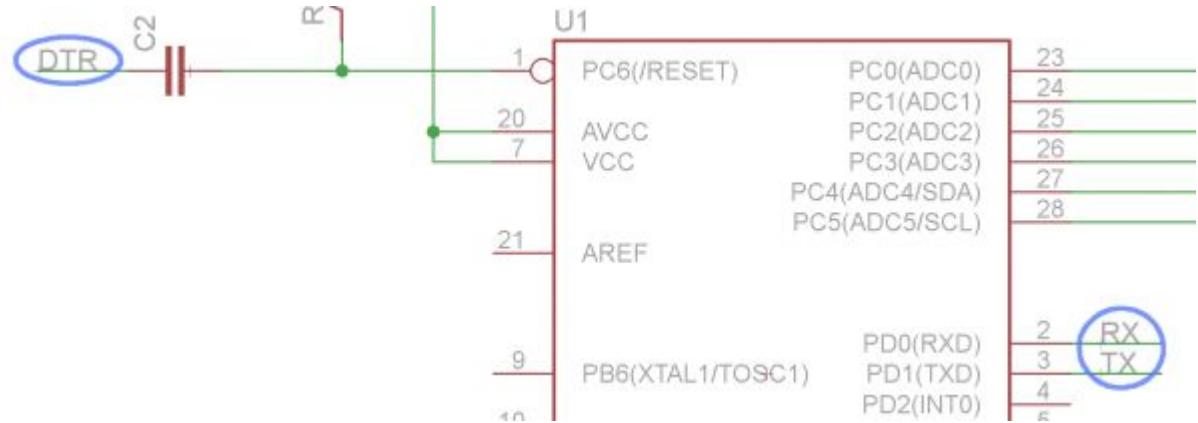
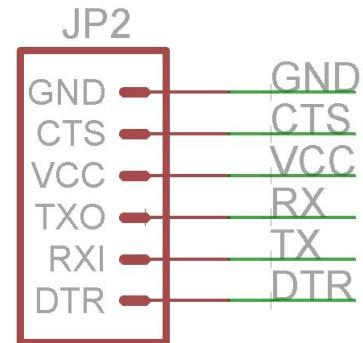


# Schematic: Wiring

## Connections Without Wiring



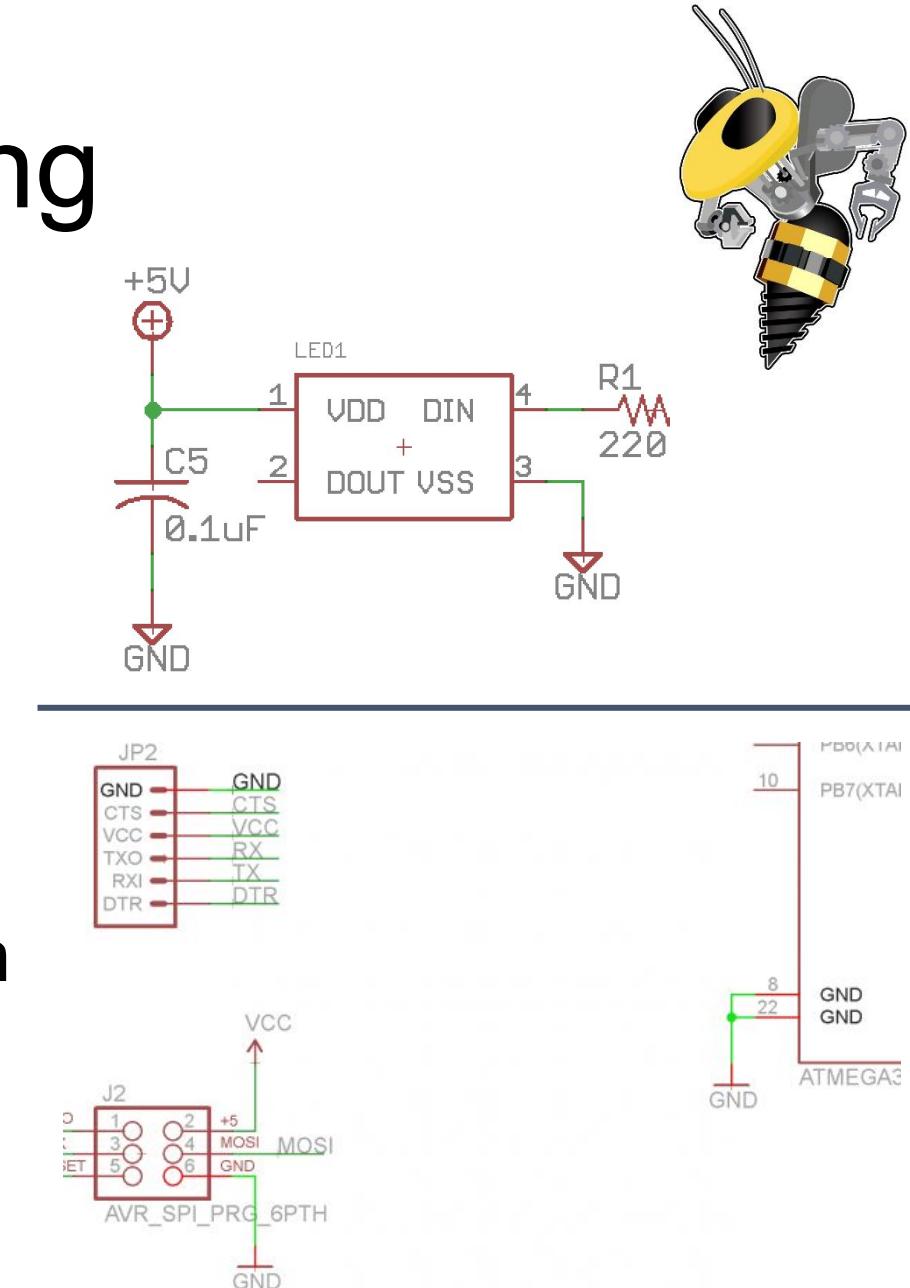
- To make it easier to see nets that are wired like this use the label tool to show the net name



# Schematic: Wiring

## Other Organization

- Power/ground can be connected together by adding power/ground parts from the supply library
- Connections can be seen and checked with the show tool



# Schematic: Checking



- Use the Electrical Rule Check (ERC) to check for any electrical errors and warnings
- Note: this will often not tell you about connections you missed
- Many warnings it gives can be ignored because often they have to do with net names, etc

# Board Layout

## Create Board File

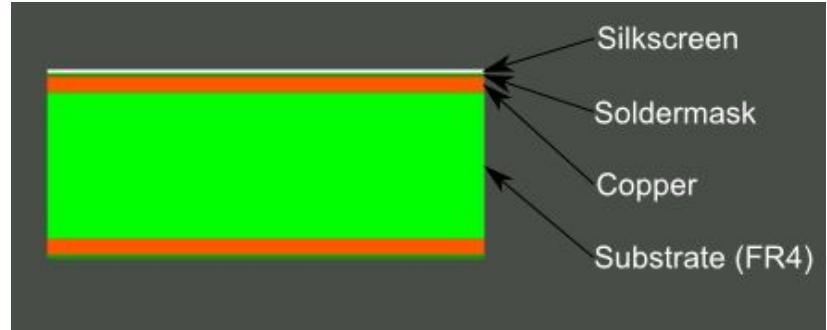


- When done with schematic click switch to board: 
- If no board file is found (file with same name as schematic), it will generate a new one
- You will be presented with components randomly placed with lines showing connections that need to be made

# Board Layout: Layers



- What is actually printed:
- Click on Layer Settings to adjust layers currently viewed
- (Tip) Right click on layer settings and create new group to quickly switch views



# Layers



Color	Layer Name	Layer Num	Layer Purpose
Red	Top	1	Top layer of copper
Blue	Bottom	16	Bottom layer of copper
Green	Pads	17	Through-hole pads (copper on top and bottom)
Green	Vias	18	Vias to route signal between layers (copper on top and bottom)
Grey	Dimension	20	Outline of the board
	tPlace	21	Silkscreen for top
	bPlace	22	Silkscreen for bottom
Yellow	tDocu	51	Top documentation layer (just for reference)

# Board Layout: Arranging Moving Parts



- Use the move tool (right click to rotate)
- How you arrange your parts has large impact on routing difficulty in next step
- First consider requirements of board
- Then consider what is the easiest placement for when you route the board

# Board Layout: Arranging

## Moving Parts: Requirements



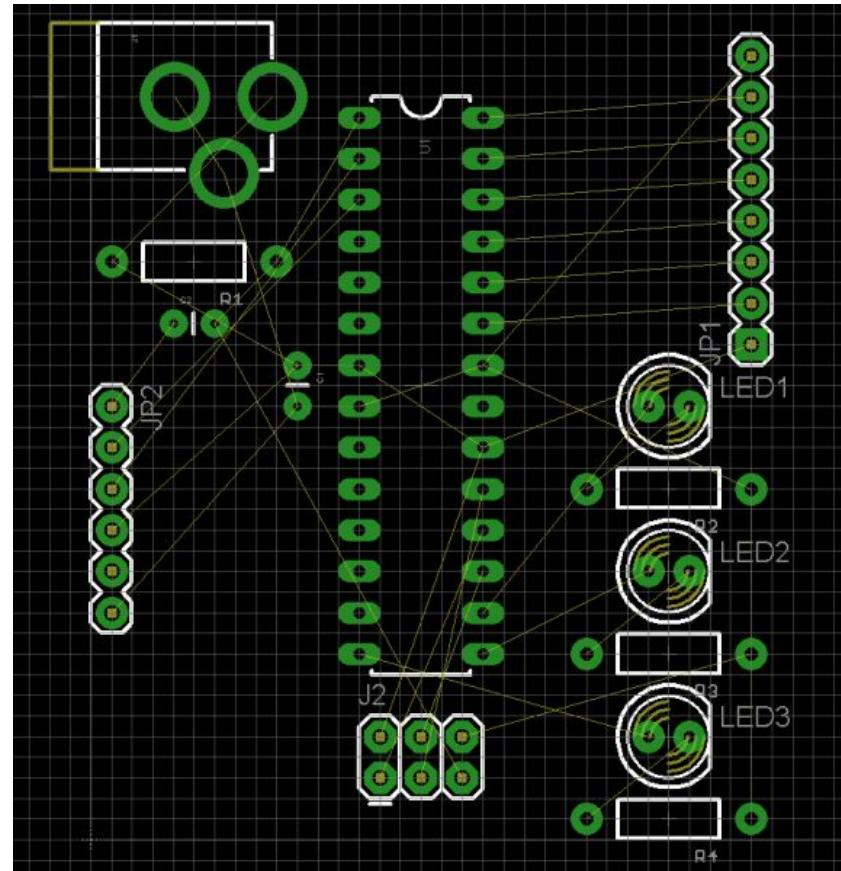
- Maximum size your board can be
- Location of specific parts
  - Specific location for connectors
  - Decoupling capacitors very close to IC
  - Maximum distance parts can be that communicate
- Location of other parts on the board
  - Mounting holes
  - Clearance with other boards or object around it

# Board Layout: Arranging

## Moving Parts: Suggestions



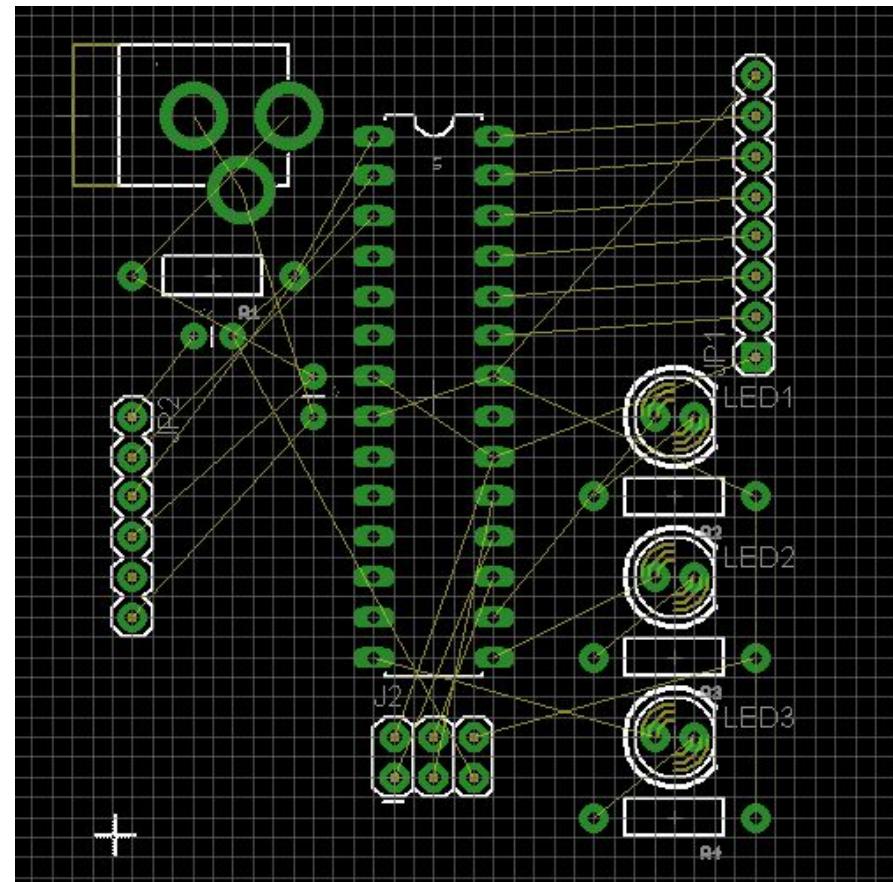
- Leave space between parts
  - Leave space for routing
  - Make sure components don't collide
- Minimize intersecting airwires
  - Much more difficult to route when airwires cross



# Board Layout: Arranging Dimension



- Use delete tool to erase automatically placed dimension lines
- Redraw box so all of your parts are within
  - Wire tool, set layer to 20 Dimension, then draw



# Board Layout: Routing



- Making all the connections shown by the airwires without overlapping anything
- Use the Route tool (not the wire)
- This is a two layer board so you can route on the top or the bottom
  - Vias are used to connect the top and bottom layers
- Don't worry about ground connections yet

# Board Layout: Routing

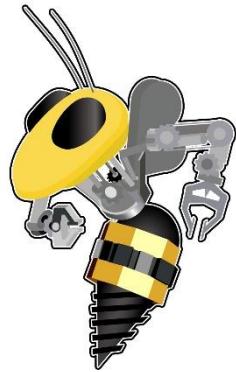
## Route Tool



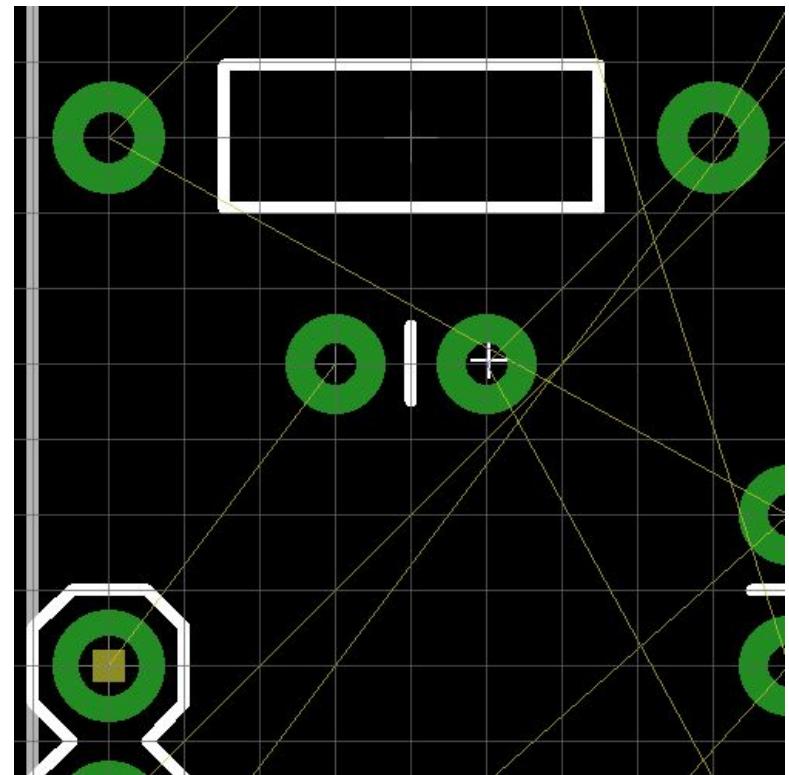
- Layer: select layer to route on
- Bend Style: angle of wires (good practice to always use 45° angles)
- Width: How wide copper trace is (default of 16 mil is fine for this)
  - Trace width is sometimes important (Ex: power)
- Via Options: Shape and size of vias (use round vias)

# Board Layout: Routing

## Routing



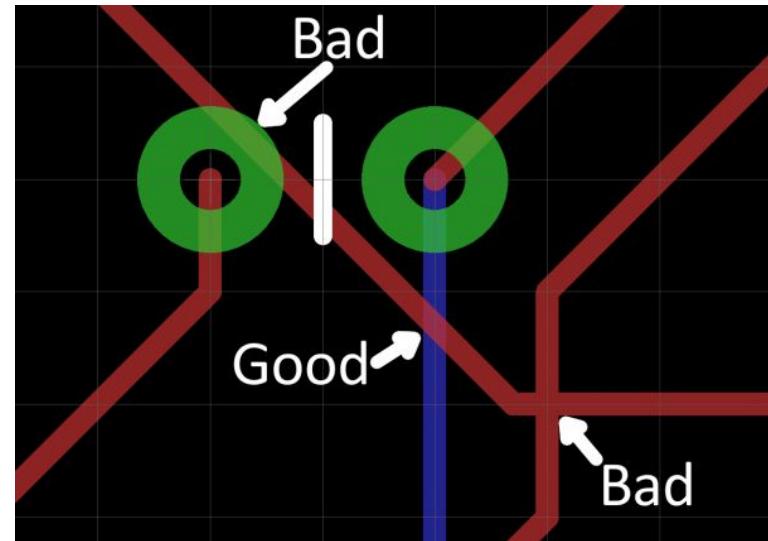
- Must start from end of an airwire and route to other end of that airwire
- Left click when routing to place segment and continue routing



# Board Layout: Routing Overlap



- For pads, traces, etc on the same layer there must be zero overlap
- Traces on separate layers can overlap
- Vias and through hole components (colored in green) are on both layers

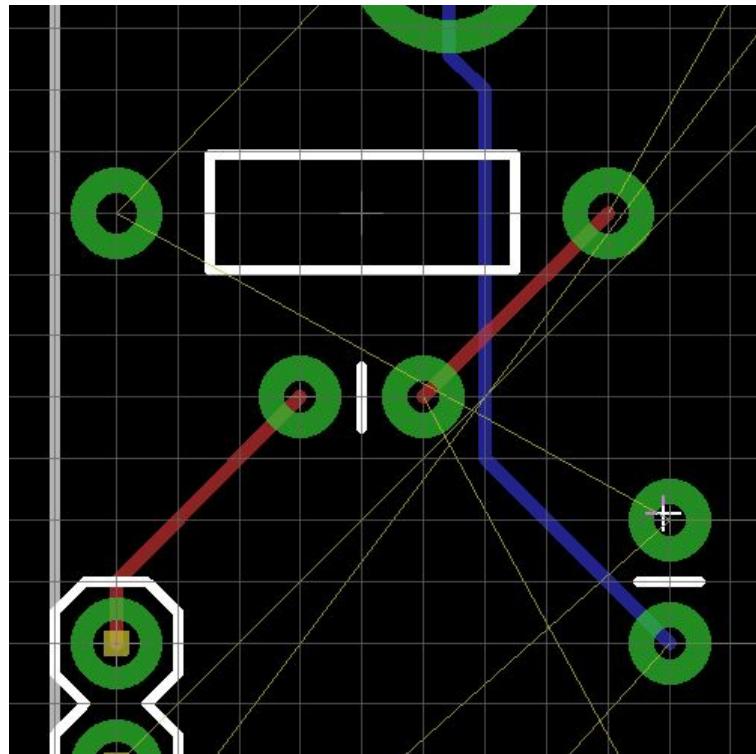


# Board Layout: Routing

## Vias



- Middle click when routing to switch layers and add via
- Vias can be added manually with the via tool
  - Net is not set automatically so must use name tool to change name of via to the same of what you are starting from

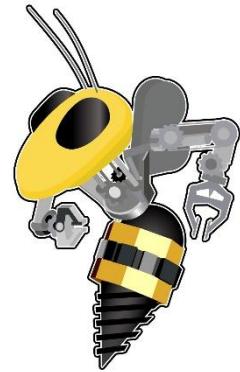


# Board Layout: Routing

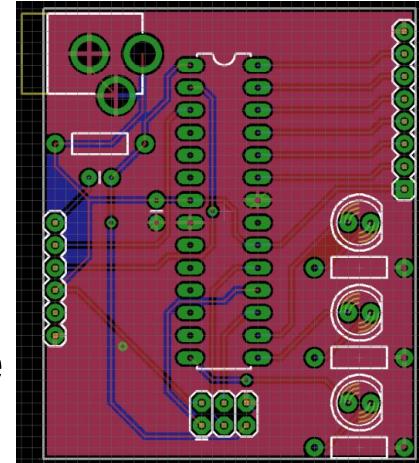


- Different PCB manufacturers have specifications for the minimum distance they can produce between traces
  - You must ensure your traces are at least that far apart
- Traces cannot be removed with the delete tool, the ripup tool must be used

# Board Layout: Polygons



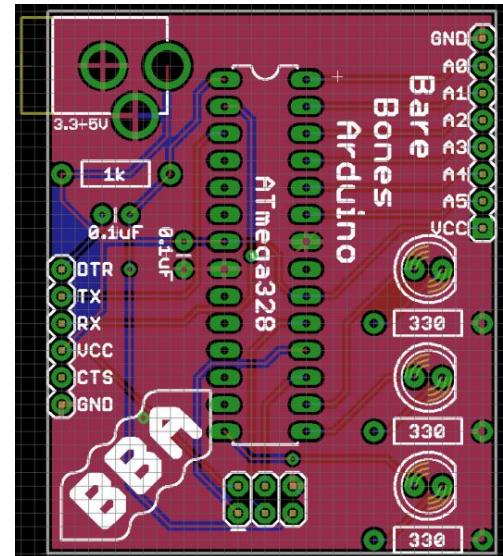
- Polygons are how large sections of copper (like ground planes) are done
- Click polygon tool and make a shape around what you want to fill
  - We want a ground plane to fill the whole board so draw the polygon along your dimension lines
- Use the name tool to connect this polygon to the net (in this case GND)
- Hit ratsnest tool to fill in the polygons



# Board Layout: Silkscreen



- Silkscreen is printed on top of the board and has no effect on the function of the board
- Most of the silkscreen comes from that done in the individual parts
- Often these labels from parts are misaligned
  - Use the smash tool to allow you to move these labels

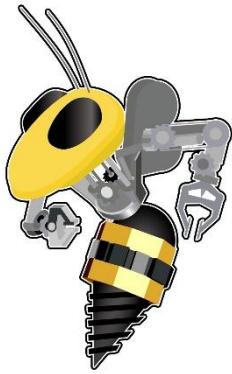


# Board Layout: Checking For Errors



- Click on the ratsnest tool then the notification at the bottom left for how much needs routing
  - Everything is routed if “Nothing to do”
- Use the Design Rule Check (DRC) to check for clearance, overlap, etc
  - Many settings can be changed for requirements of where your board is being printed
- When “Ratsnest: Nothing to do” and “DRC: No errors” you are done

# Board: Exporting



- Export board into Gerber files which are sent to the manufacturer to print
- Click on the CAM processor tool then File>Open>Job then select gerb274.cam
  - Settings for specific layers to export can be changed
- There will be a file for each layer (Silkscreen, copper, soldermask, and drill file)