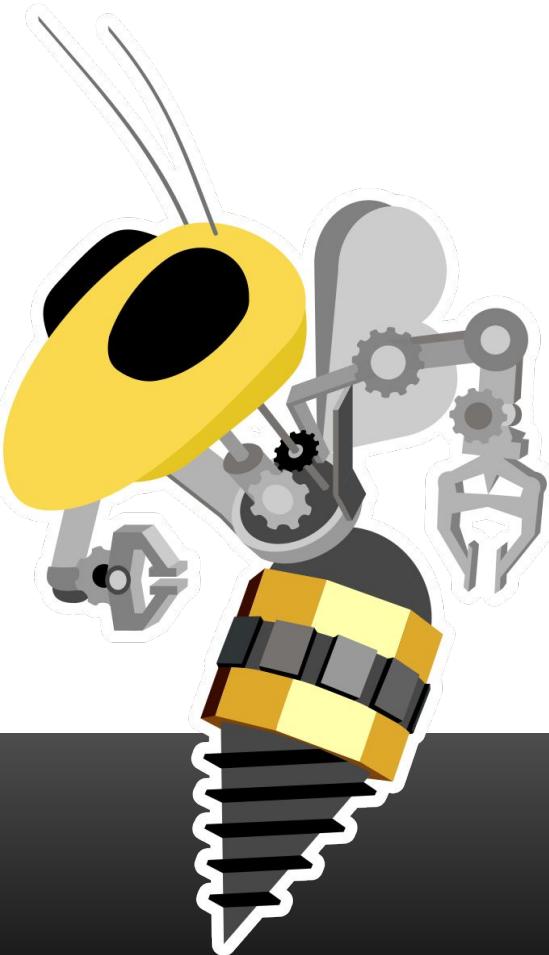


Welcome!

Electrical Training
Week 3

ROBOJACKETS
COMPETITIVE ROBOTICS AT GEORGIA TECH

www.robojackets.org

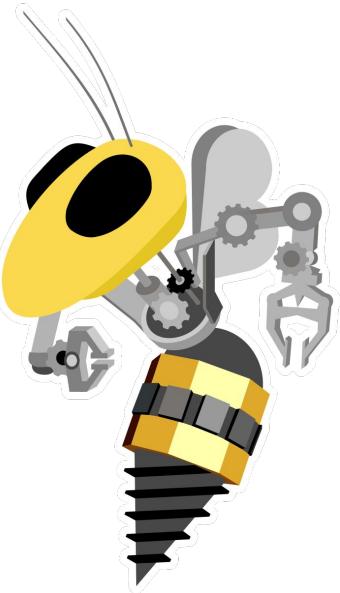


Last Week!

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

This Week!

- Communication Systems
- EAGLE Schematics

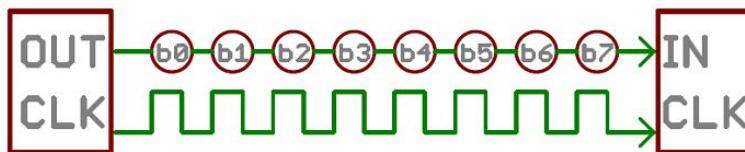


Communication Systems

How many are there again...?

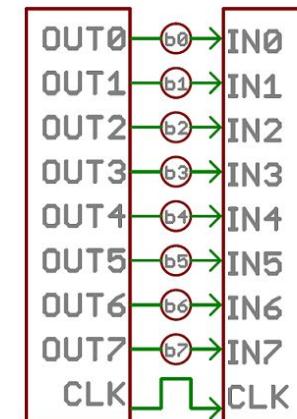
Serial vs Parallel Communication

Serial



- Stream data one bit at a time
- Example: USB, SPI

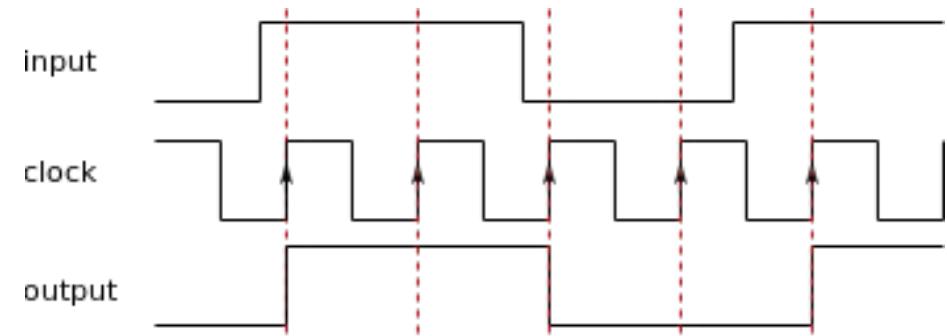
Parallel



- Many bits of data sent at the same time through different wires.
- Example: PCI and DIMM (on computer motherboards)

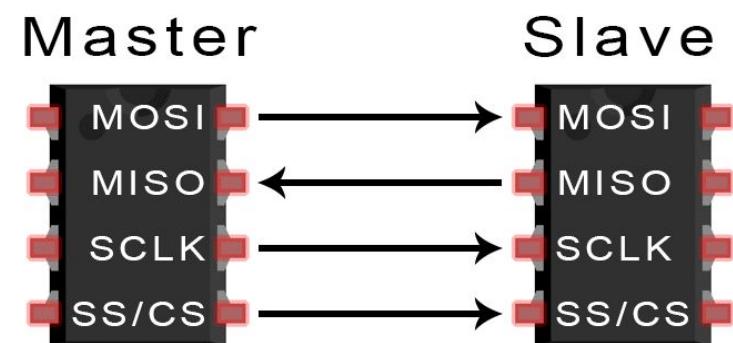
Clock Signals

- Square waves of known frequency (baud rate)
- Edge used to synchronize data reading across communicating devices



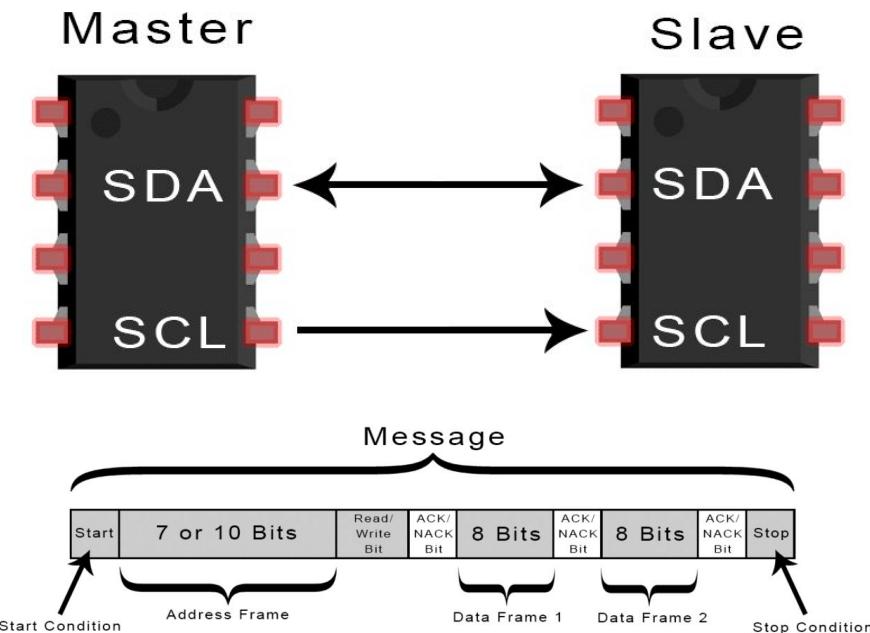
SPI (Serial Peripheral Interface)

- Continuous bidirectional transfer
- All devices share 3 Lines
 - Unique Slave Select line per device
- Master controls CLK



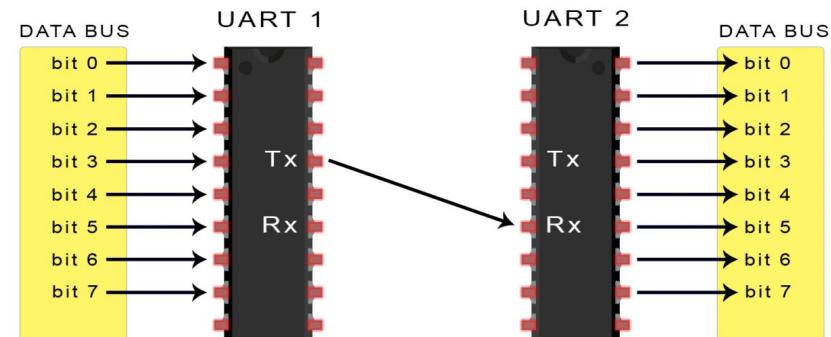
I2C (Inter-Integrated Circuit)

- Synchronous
- Uses only two wires :
 - SCL: Clock signal
 - SDA: Data signal
- Sends data in ‘frames’
- Any device can claim master by controlling SCL



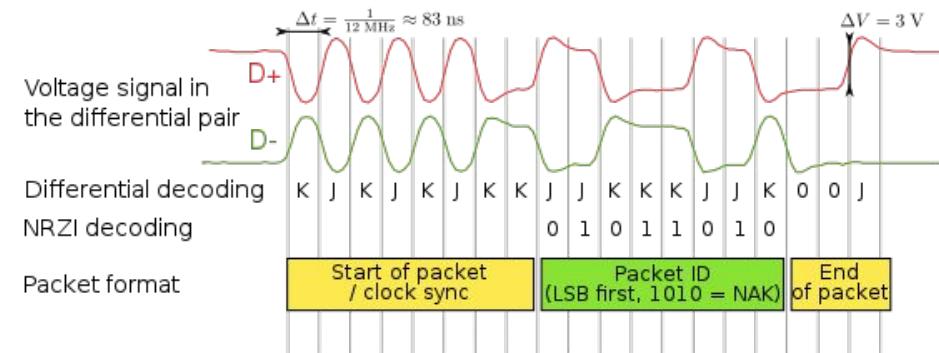
UART (Universal Asynchronous Receiver/Transmitter)

- Asynchronous (no clock needed)
- Uses 2 wires
- Need same baud rate



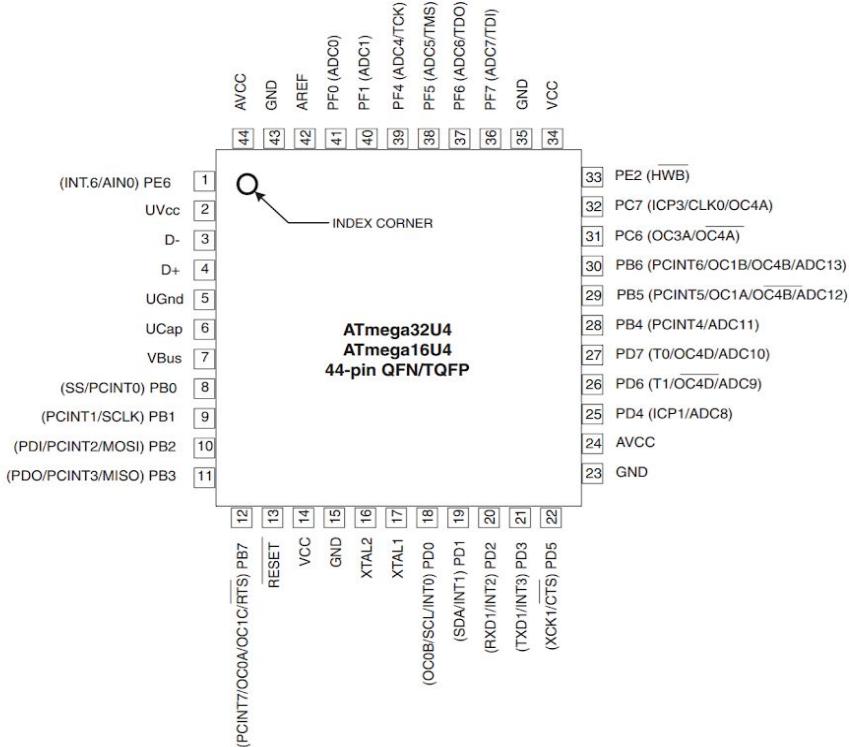
USB (Universal Serial Bus)

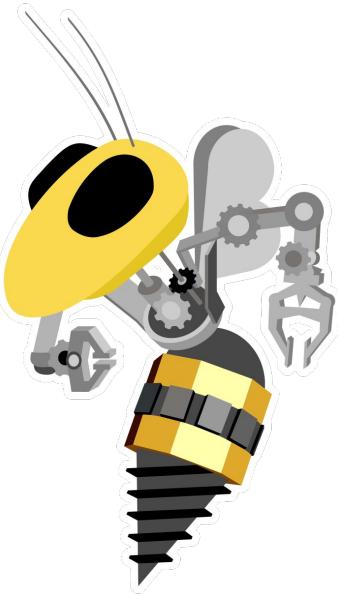
- Differential Pair signal
- Defines rate in “clock sync” phase
- Useful for computer-device communication



ATmega32u4 (MCU) Pinouts

- Shows pin configurations and functions on the training board microcontroller
- Datasheet





What is EAGLE?

ROBOJACKETS
COMPETITIVE ROBOTICS AT GEORGIA TECH

Installation

- Free for students
 - Requires an Autodesk account and GT Email
- <http://www.autodesk.com/education/free-software/engine>

Control Panel

File View Options Window Help

Name Description Last Modified

- ▶ Libraries
- ▶ Design Blocks
- ▶ Design Rules
- ▶ User Language Programs
- ▶ Scripts
- ▶ CAM Jobs
- ▶ SPICE Models
- ▶ Projects

Home Preview

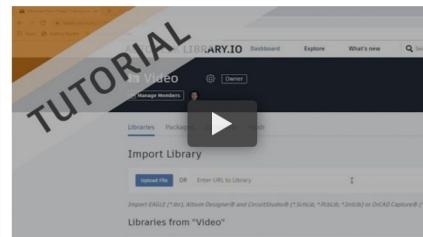
Recent Files

- Control.brd
- Control.sch
- Control.sch
- Control.brd

Recently Generated 3D Files

Your recent generated 3d files will be visible here.

What's New in Eagle 9.5.1



TUTORIAL

Signal Quality Series Part 3: Transmission Lines Principles
Tue Oct 22 2019 02:01 PM [Register](#)

Mapping Simulation Models
Thu Oct 31 2019 02:00 PM [Register](#)

See previous webinars

Shared Managed Libraries

Sharing Managed Libraries. Invite members to collaborate, share and edit content on your Managed Folder.

QUICK TIP

Buy Now! >

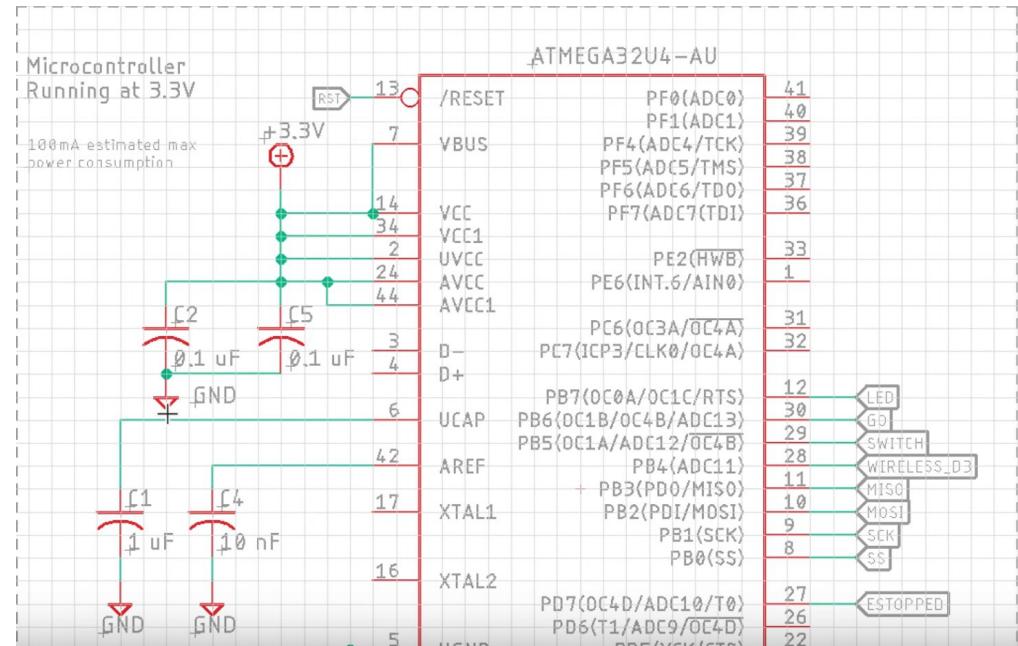
Unlock more layers, sheets, and a larger board area with an EAGLE Subscription.

Learn

Learn how to quickly start designing with our product tutorials and resources in our Learning Center. Or if you're having a specific issue, ask our experts on the forum!

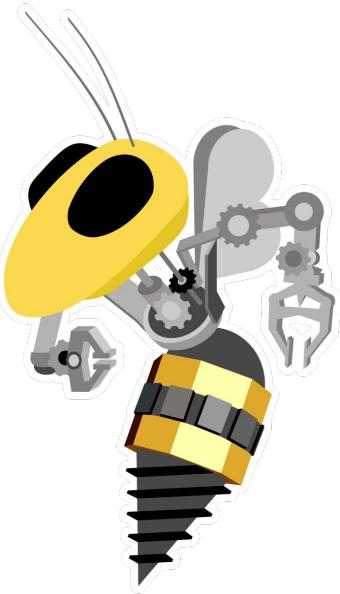
Learning Center >

Goals by End of Today (Schematics)



Goals by End of Today (Schematics)

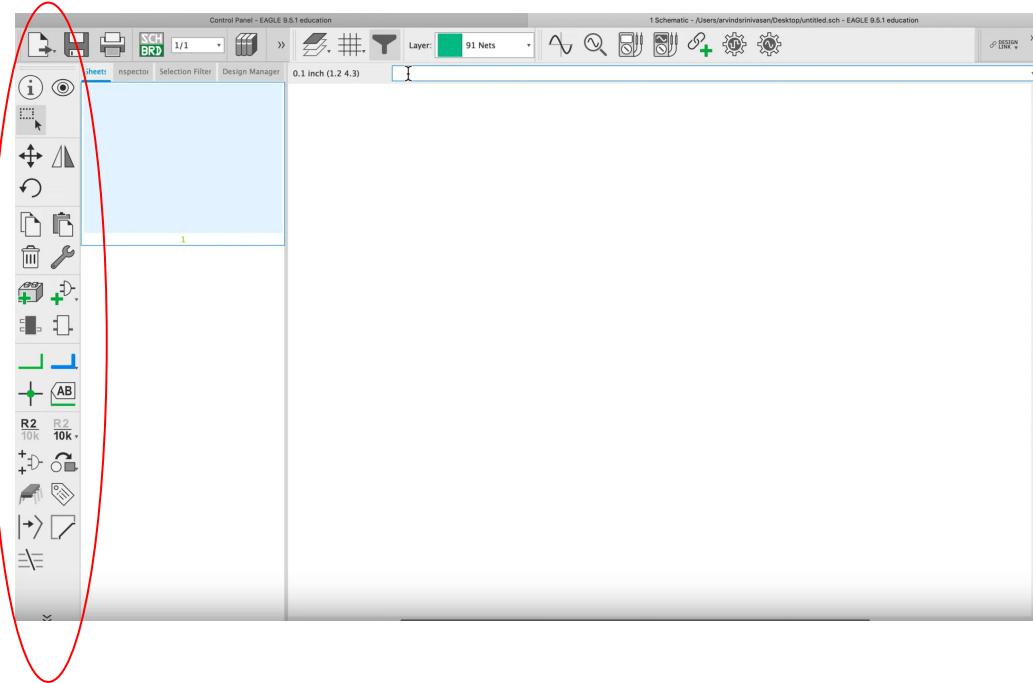
- We'll learn how to make something like what you just saw on the previous slide
- If what you just saw seemed intimidating, no worries! By the end of training today, you'll understand what the symbols mean and how to create your own schematic :)



Some Useful Features to Know

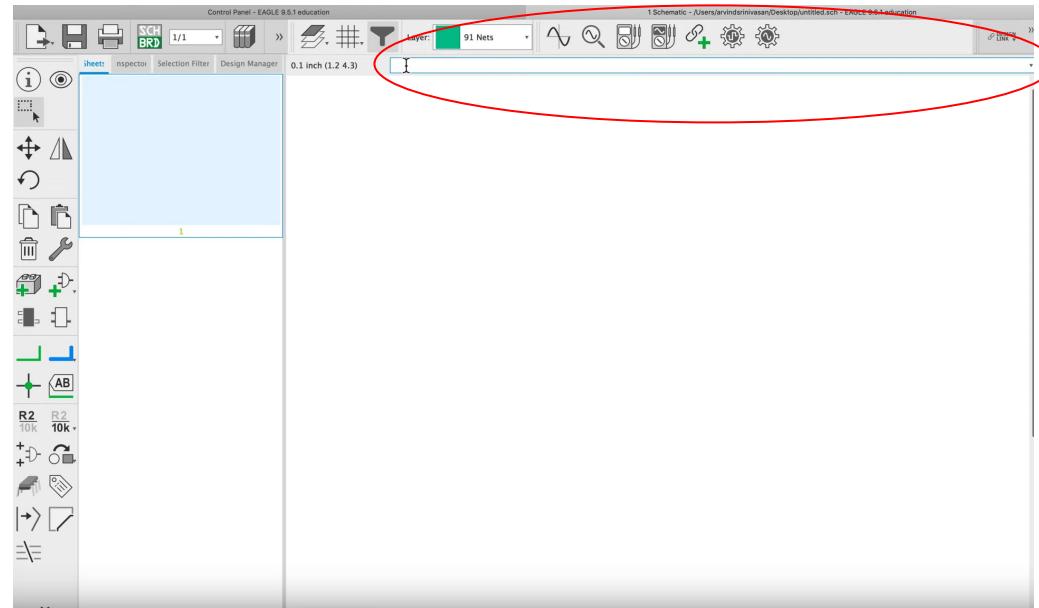
SideBar Tools

Most actions and tools are in the bar on the left (ex. move, add ERC)



Command Line

You can also perform actions by typing them in the command line (ex. Moving parts, ERC)



Searching for Parts

*Use wildcard
character if
you don't have
a perfect
match*

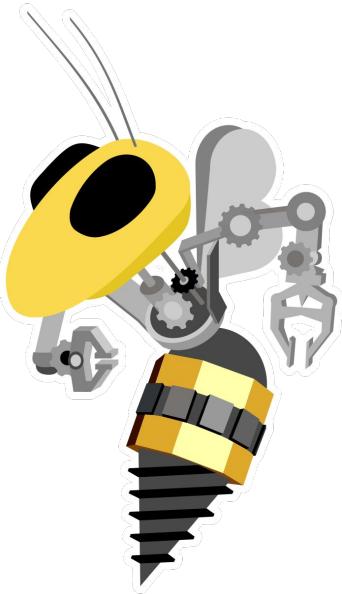
The screenshot shows the EAGLE software's library search interface. On the left, a tree view of libraries is shown, with 'RoboJackets-Resistors' expanded to show 'R0603W'. In the main pane, the search results for 'R0603' are displayed, with 'R-EU_R0603' selected. The right side of the interface shows the component details: a schematic symbol (G\$1), a footprint diagram (NAME > VALUE, 5mm / 0.2in), and a 3D model of the resistor. Below the footprint diagram, the component is identified as a 'RESISTOR, European symbol' with a 'Footprint: R0603 (Version 1)'. The search bar at the bottom contains the query '*R0603*', with the asterisk highlighted by a red oval.

Searching for Parts

- Placing asterisk symbols before and after part name you typed will search for parts that include those characters if you can't get perfect match (called wildcard)
- We'll go more in depth about searching for parts but for now, PLEASE DON'T FORGET the wildcard feature

EAGLE Resources

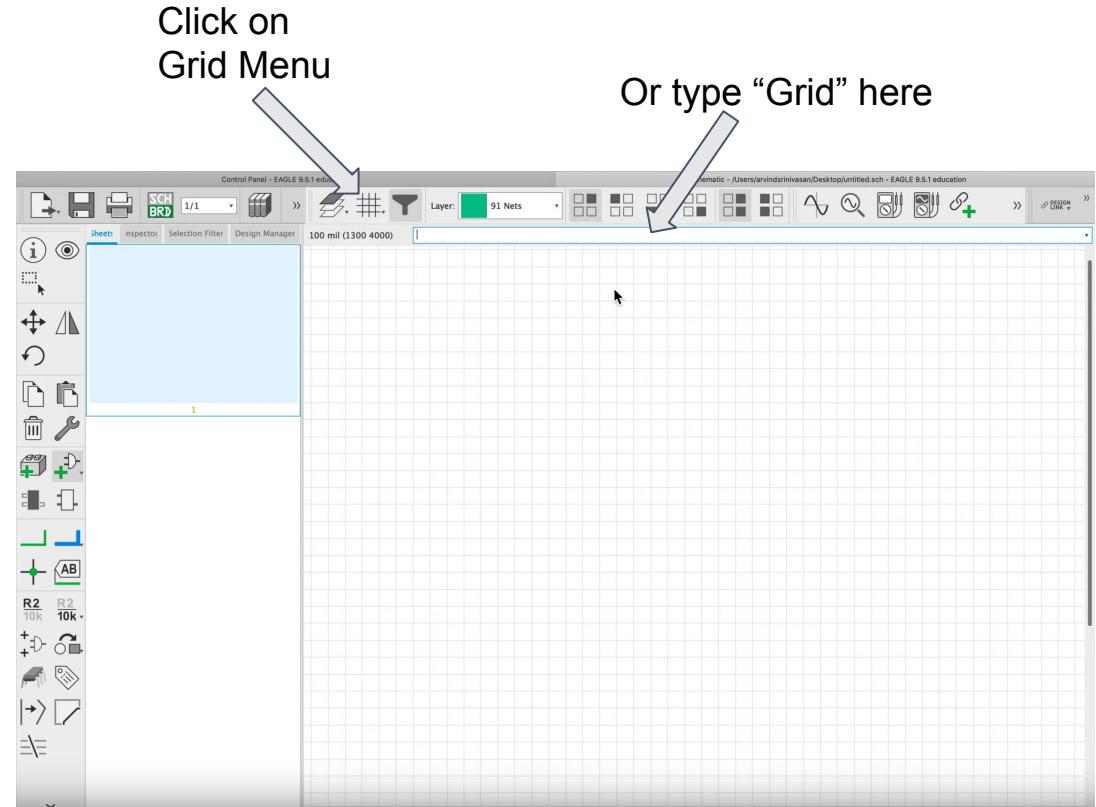
- Detailed walkthrough of all EAGLE features
 - [/references/eagle_training_guide/eagle_guide.pdf](#)
- Quick “cheat sheet” to look at for command names
 - [/references/eagle_training_guide/eagle_cheat_sheet.pdf](#)
- Videos
 - [YouTube -> RoboJackets Training -> Playlists -> EAGLE Training](#)



How to Build your Own Schematic

Step 1: Grid Lines

*Your screen
should look
like this.*



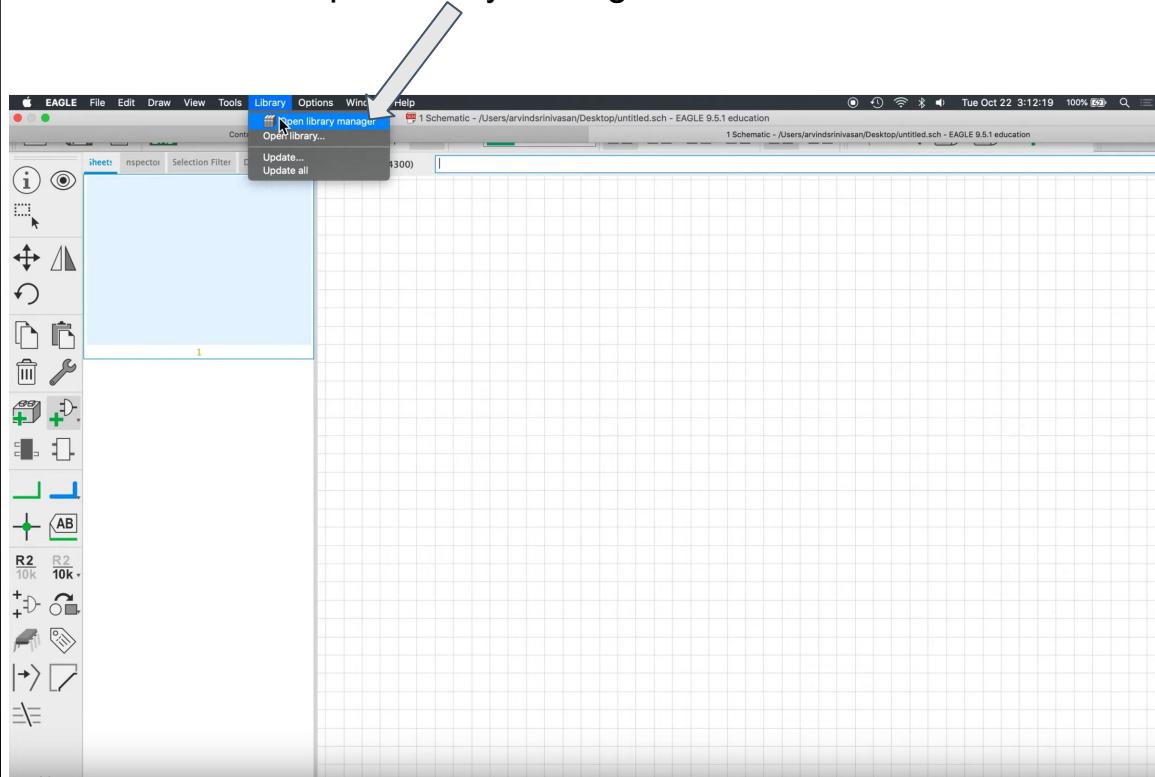
Step 1: Grid Lines

- Open up the grid menu by either clicking on the grid menu at the top or typing “grid” into command bar and make sure:
 - Display is on
 - Style is line
 - Size is 0.1 in inches
 - Alt is 0.01 in inches
- The previously blank page should now have grids

Step 2: Library Settings

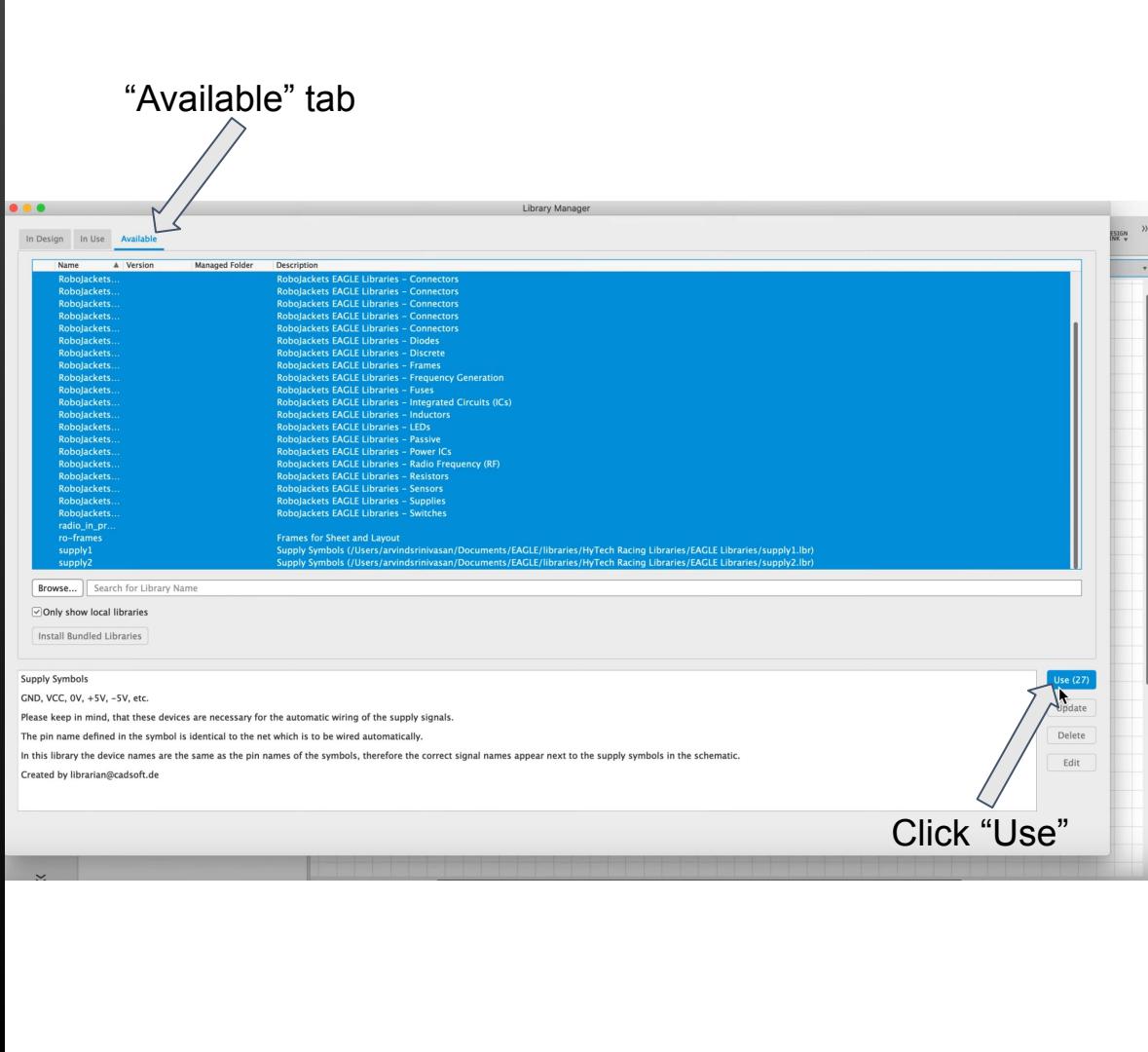
The point of this step is to make sure we are using the RoboJackets library for parts.

Open library and then
“Open library manager”



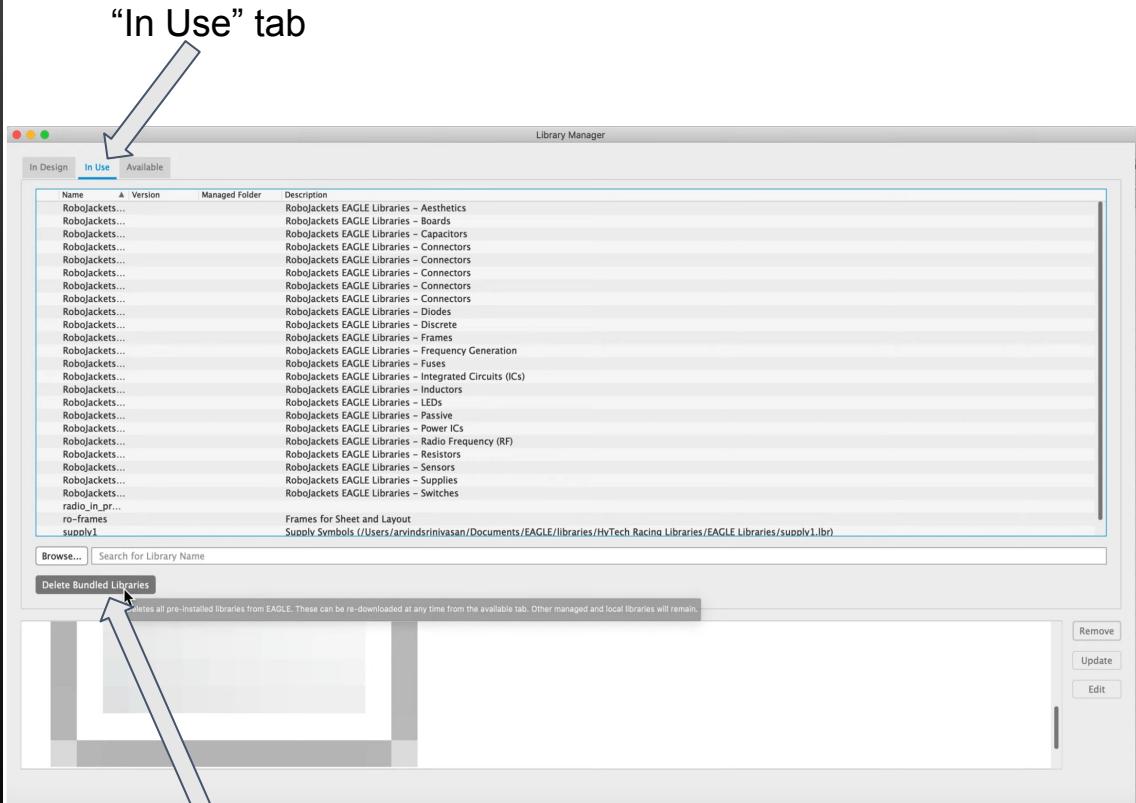
Step 2: Library Settings

*Click the
“Available” tab and
click & drag the
RoboJackets parts.
Then click “Use” on
the bottom right
side of page*



Step 2: Library Settings

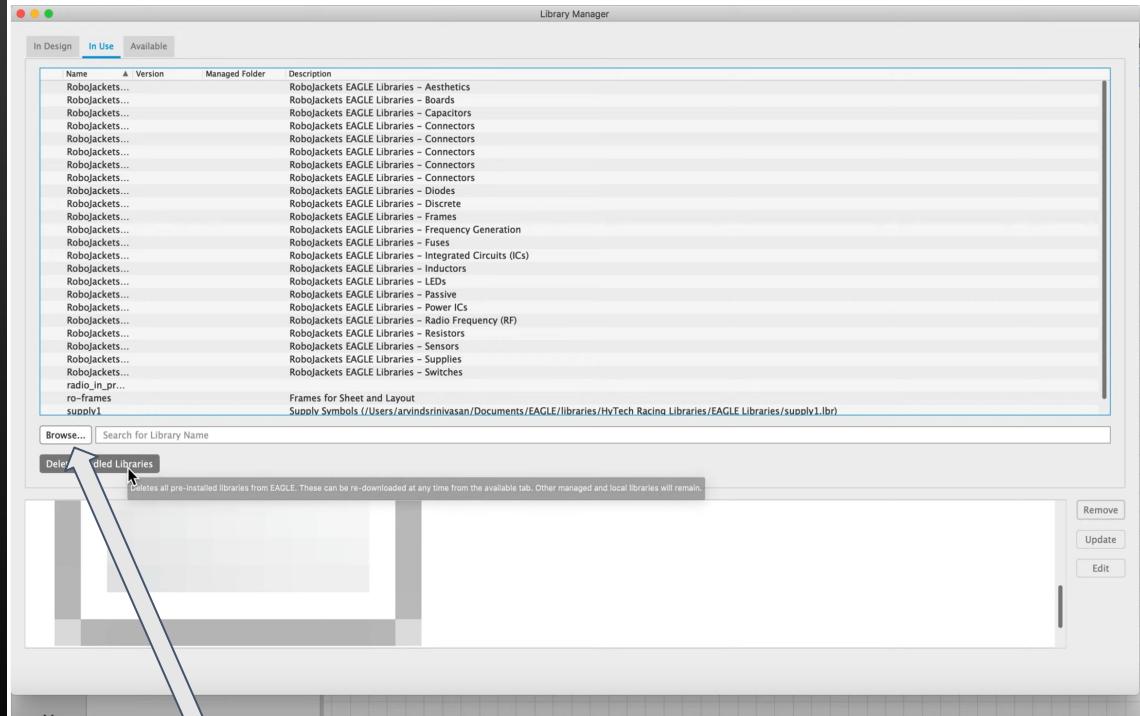
*Click on “In Use”
tab and then click
“Delete Bundled
Libraries”*



**“Delete Bundled
Libraries”**

SideNote

You may or may not have to use the “Browse” Feature to see the parts



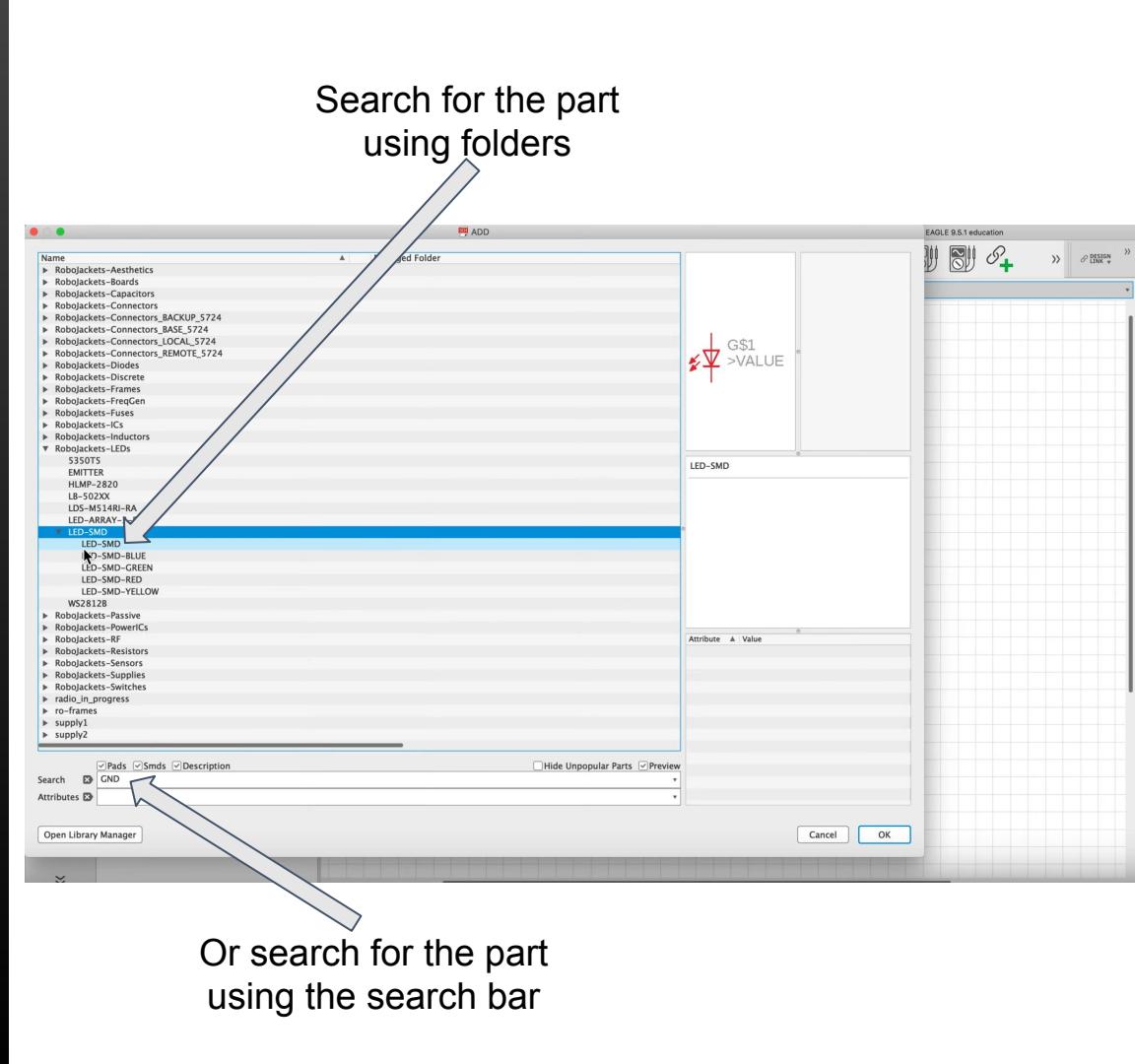
“Browse” Feature

Step 3: Adding Parts

- This step is where we'll be adding the parts that we'll need
- Click on "Add Part" from the bar at the left or type "add" in the command bar
- You can search for the part through the folders or typing it in the search bar
 - If you can't find the exact part through the search bar, remember that wildcard exists

Step 3: Adding Parts

Double click on the part and then click on the grid line where you want the part to be.



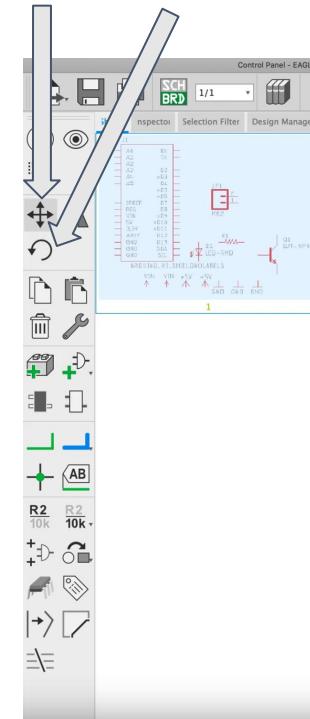
Step 4: Rearranging Parts

- After you've found most or all of your parts, we may need to rearrange their positions or orientation
- You can move your parts around the grid by clicking “Move” on the left bar or typing “move” in the command bar (You can also right click to rotate while part is selected and being moved)
- You can rotate your parts by clicking “Rotate” on the left bar or typing “rotate” in the command bar

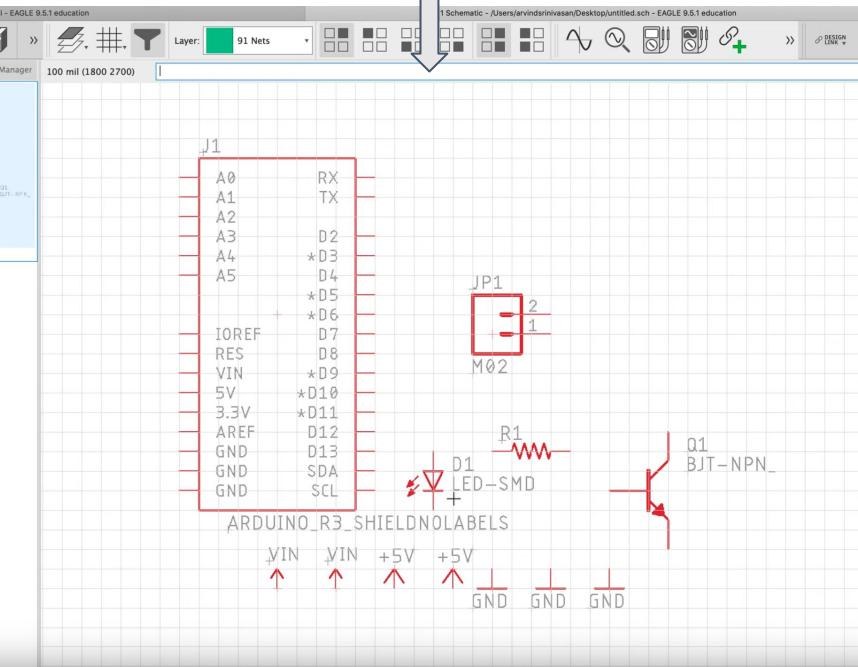
Step 4: Rearranging Parts

*Once you click
move or rotate,
click on the part
to interact.*

Click
Move Click
Rotate



Or type “move” or
“rotate”

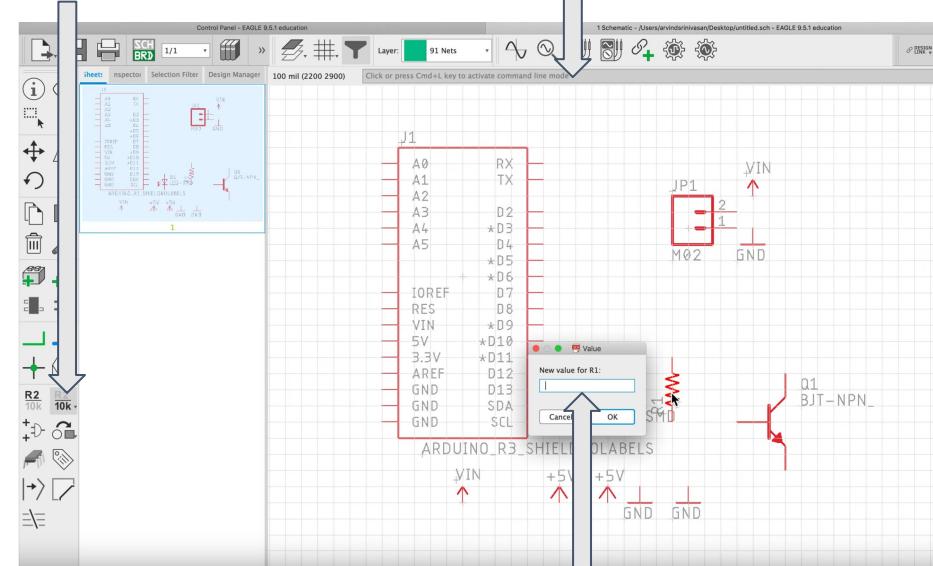


Step 5: Setting Value of Parts

- To understand the schematic as a whole and how it interacts with other parts, some parts may need values to them such as resistors
- Click “Value” on the left bar or type “value” in the command bar
- Left click on the part to add the value and type the amount

Step 5: Setting Value of Parts

Click
Value



Or type “value”

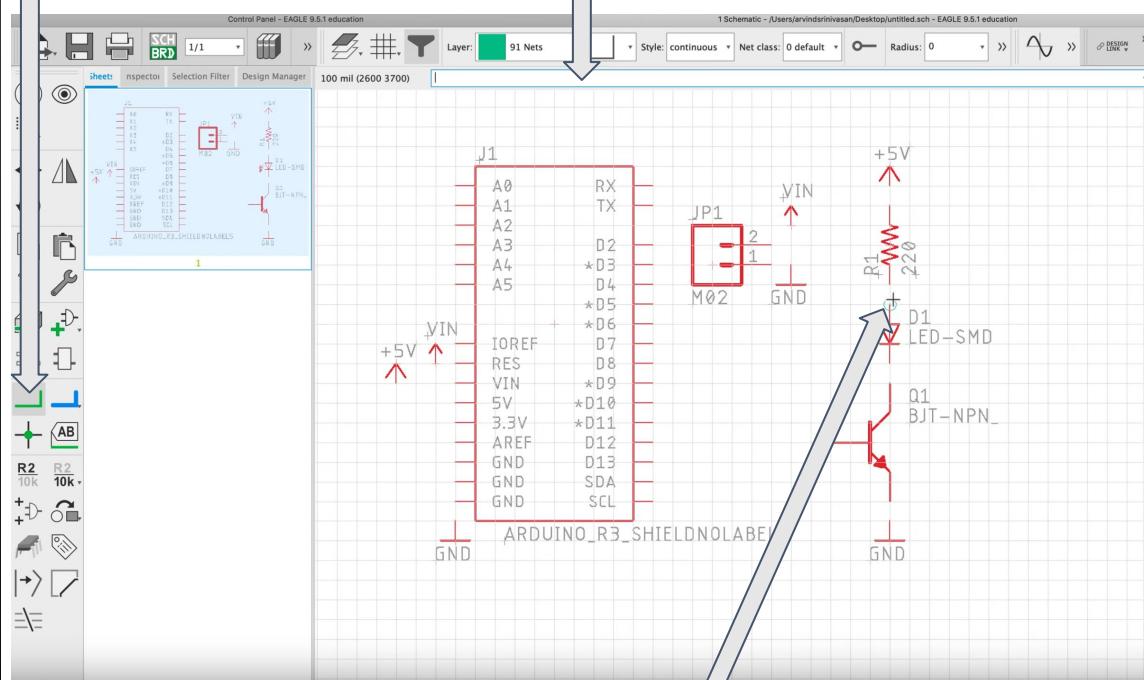
Set the amount and
then click “ok”

Step 6: Adding & Deleting Nets

- To connect the parts, we need to add nets to them
- Click on “Net” on the left or type “net” in the command bar
 - You can change the orientation of a net by right clicking it
 - If you hover over a part’s pin, you should see a small circle
 - Click on the circle and drag it to another part’s pin and then you should see the connection
- If you made a mistake, click on “Delete” on the left or type “delete” in the command bar and then left click on the net

Step 6: Adding & Deleting Nets

Click
Net



Or type “net”

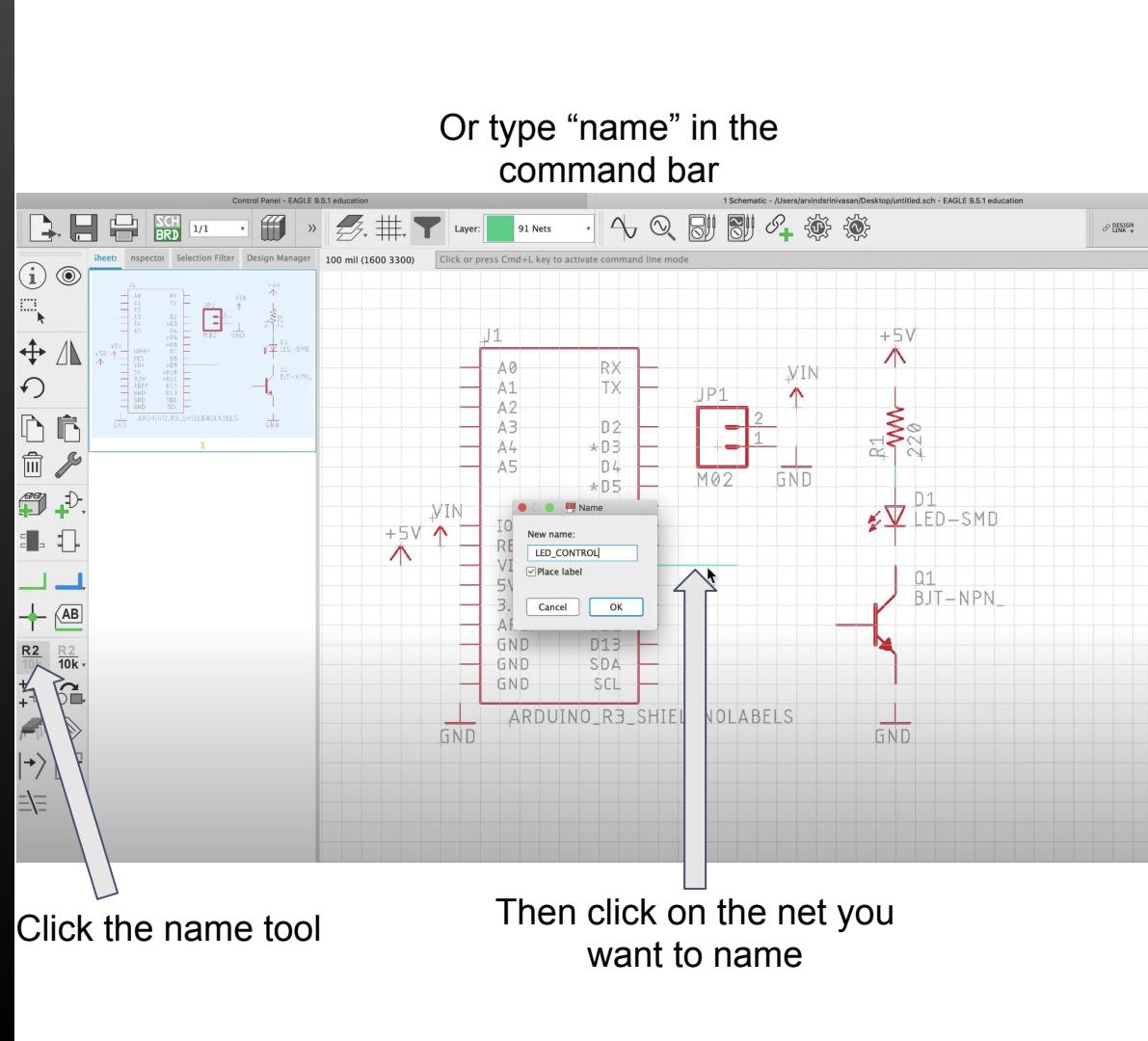
Click on the circle and
drag it to another pin to
create a connection

Optional Step: Name and Label

- In certain scenarios, you may want to name where your net is leading up to and display it on the schematic
 - The name tool must be clicked first and then the net you want to name
 - Once a descriptive name is typed, we can use the label button on the toolbar to display the name
 - Make sure the label setting is in trapezoidal label and size is set to 0.05

Optional Step: Naming

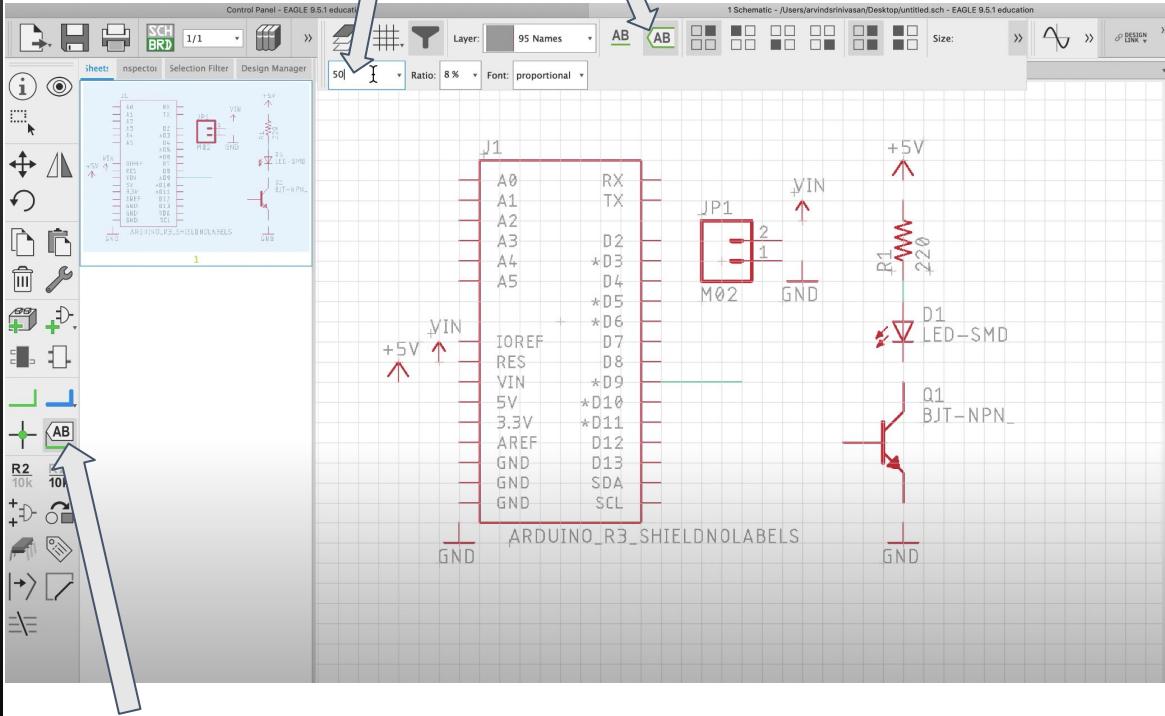
*Click the name tool,
then click on the net
you want to name, and
type a descriptive name
for the net.*



Optional Step: Label Settings

Click the label tool and make sure to set the label to trapezoid and size of 0.05 under the dropdown menu as shown in the arrows.

Make sure to set the label to trapezoid and size of 0.05

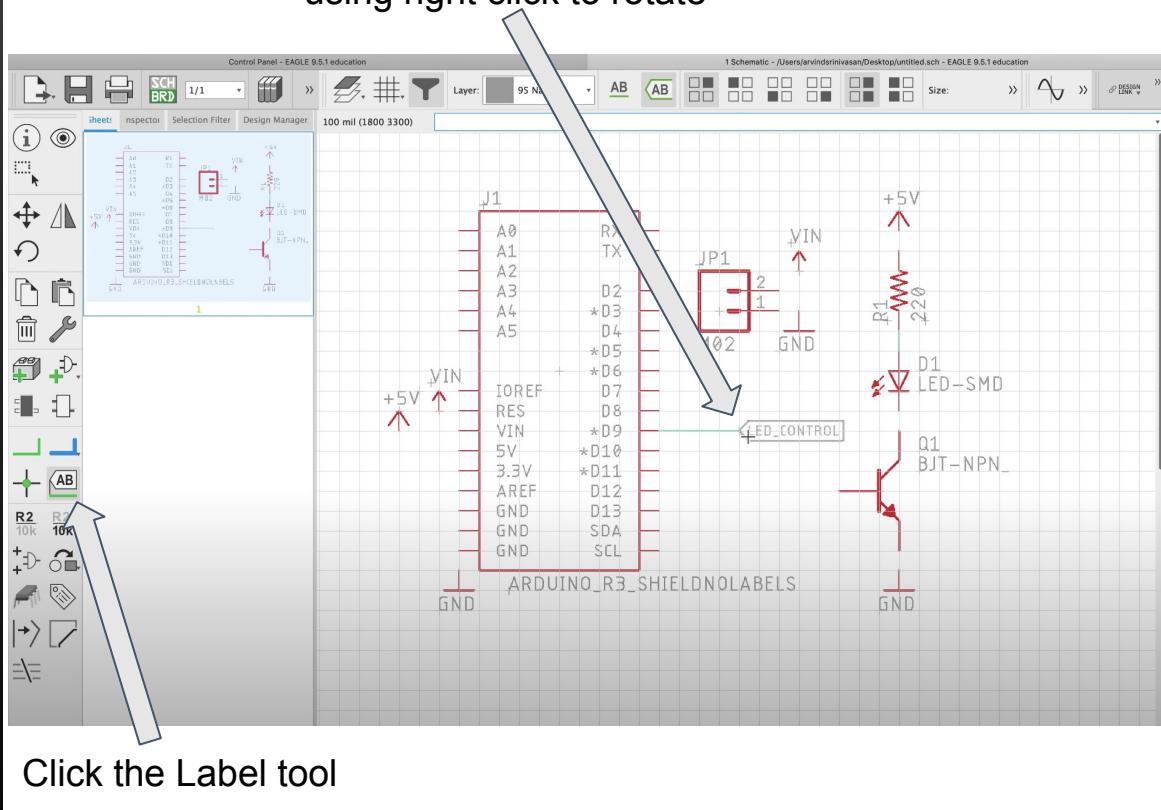


Click the Label tool

Optional Step: Label Settings

After adjusting the settings, click on the net and line up the label at the end of the net using right click to rotate.

Click on the net and line up the label at the end of the net using right click to rotate



Step 7: Electrical Rules Check (ERC)

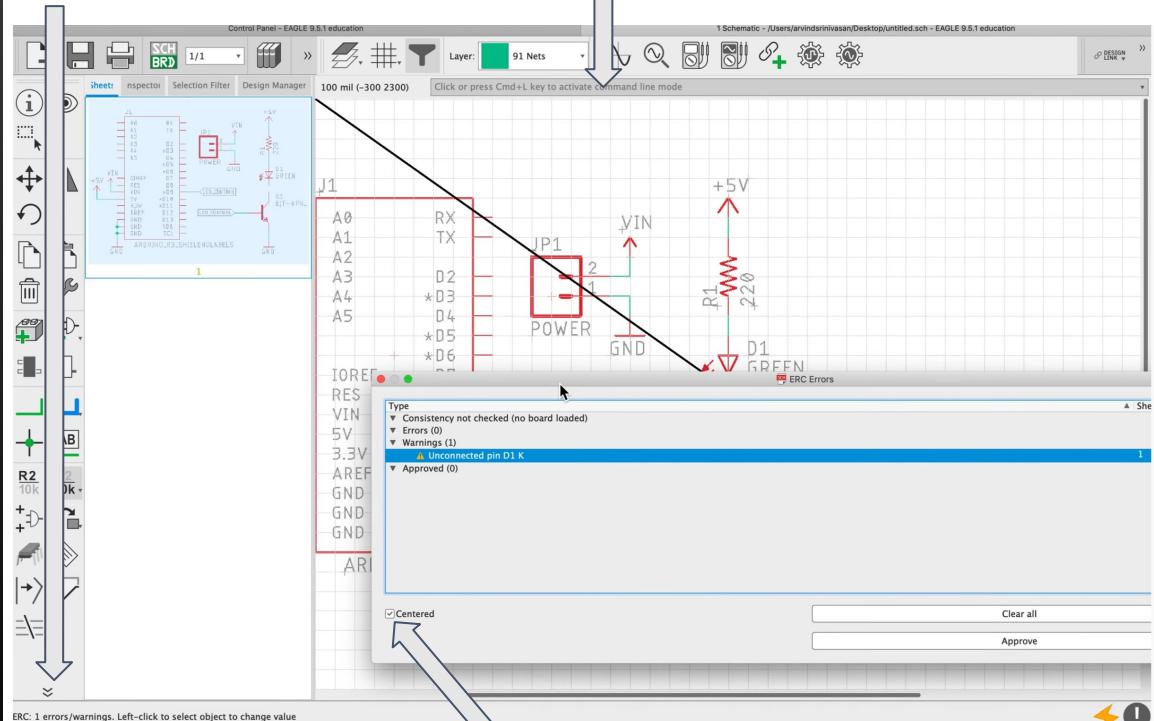
- ERC is to ensure our schematic is good and all the connections are made
- Click on “ERC” on the left bar or type “erc” in the command bar
 - If there are no warnings or errors, the bottom left corner should say so
 - If there is a warning or error, an ERC error page will pop up
 - Check the centered box and then click on the error or warning which will bring you to the site of the error
 - Fix the issues and rerun ERC again

Step 7: Electrical Rules Check (ERC)

*If there are
errors or
warnings.*

Click the
dropdown
and then
ERC

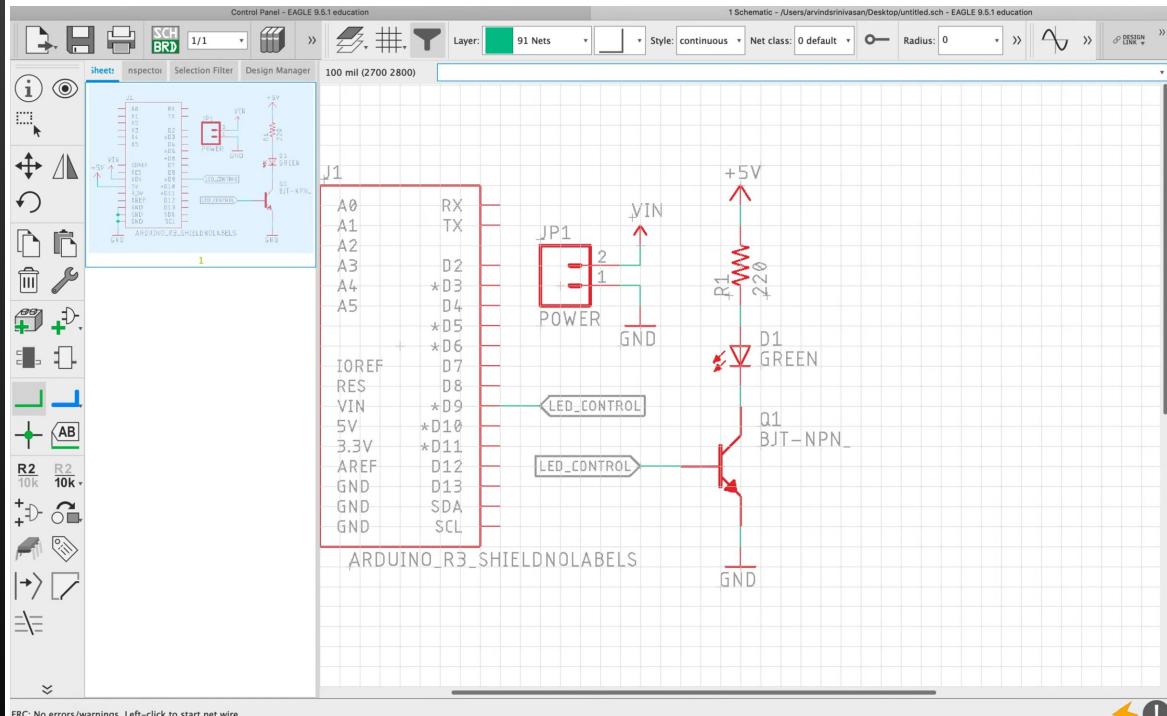
Or type “erc”



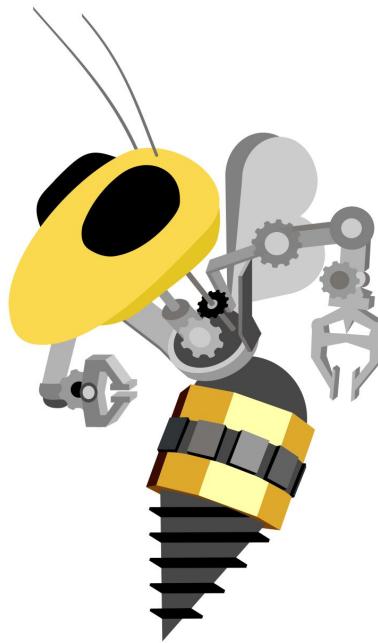
Click on the centered box
and then click the error or
warning to fix it

Step 7: Electrical Rules Check (ERC)

*If there is no
error or
warning.*



If there's no warning or error, nothing should pop up and the bottom left corner should be like this



Lab!

Creating a Schematic in EAGLE