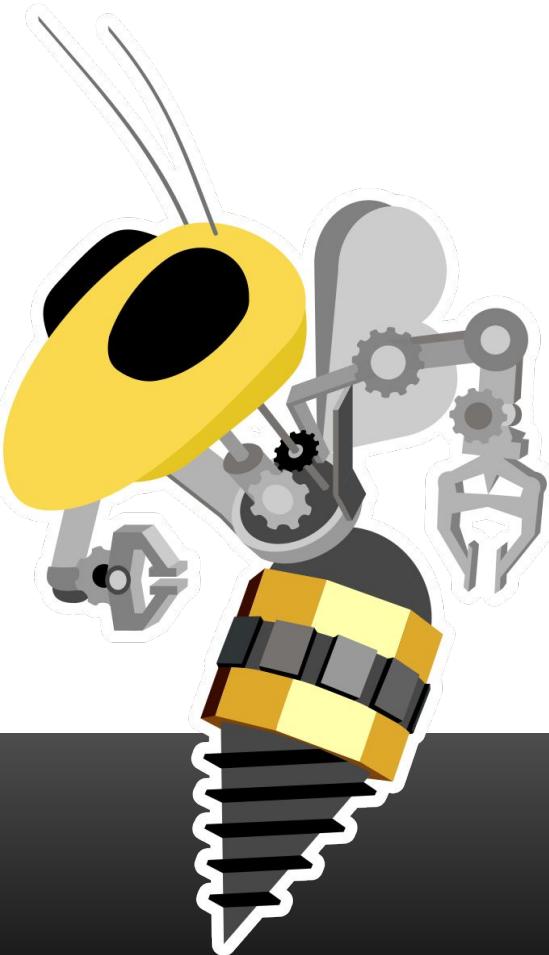


# Welcome!

Electrical Training  
Week 3

**ROBOJACKETS**  
COMPETITIVE ROBOTICS AT GEORGIA TECH

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

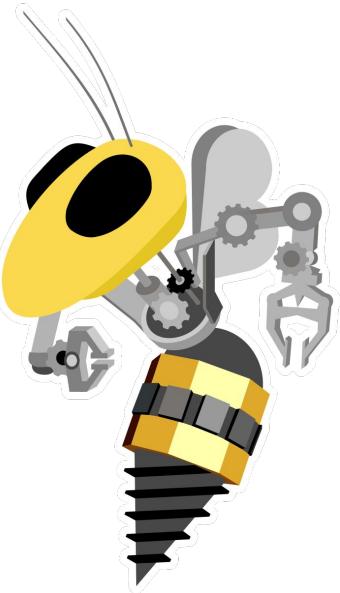


# Last Week!

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

# This Week!

- Communication Systems
- KiCAD Schematics

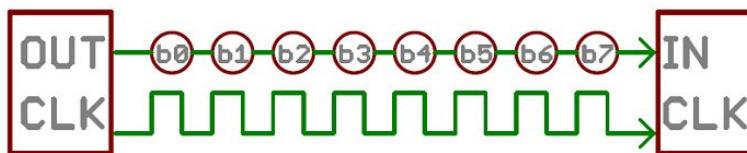


# Communication Systems

How many are there again...?

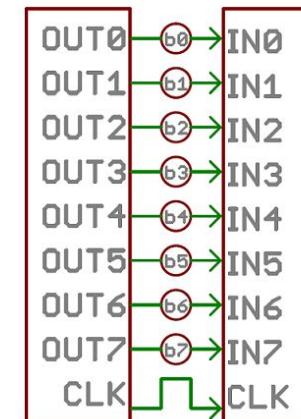
# Serial vs Parallel Communication

Serial



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

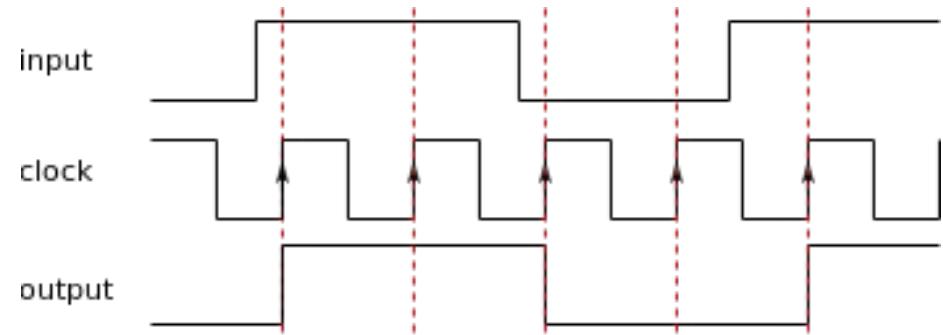
Parallel



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

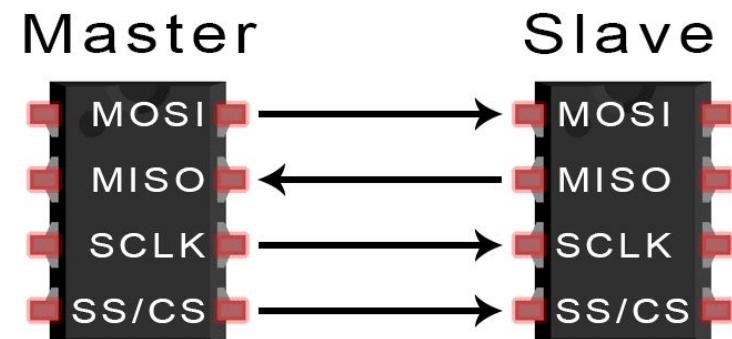
# Clock Signals

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



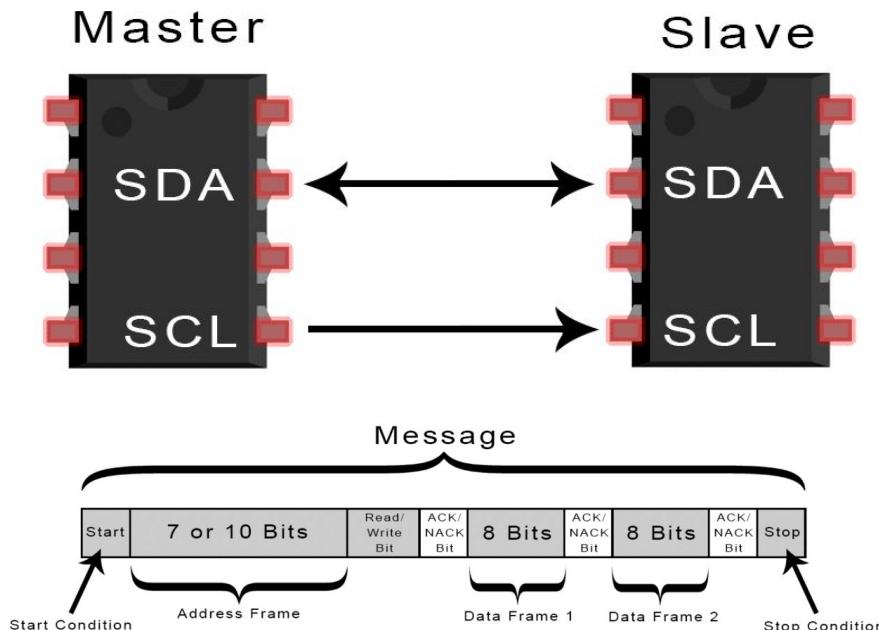
# SPI (Serial Peripheral Interface)

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



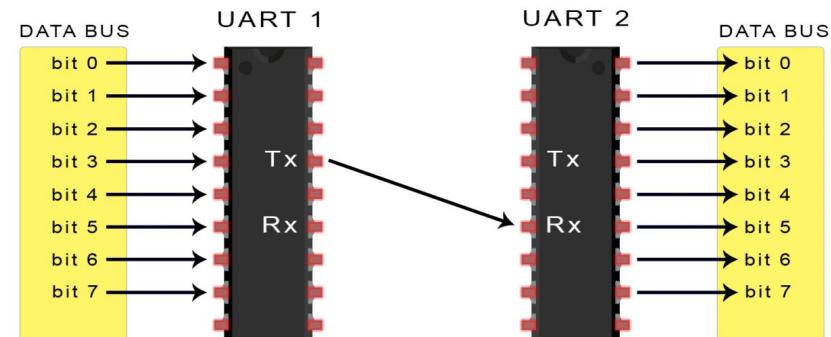
# I2C (Inter-Integrated Circuit)

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



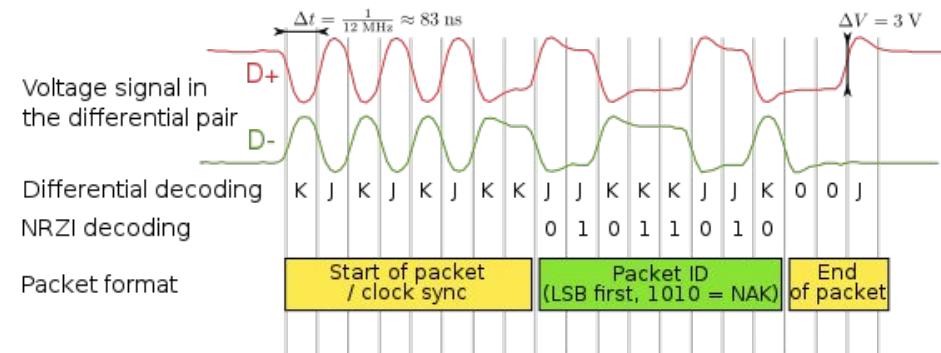
# UART (Universal Asynchronous Receiver/Transmitter)

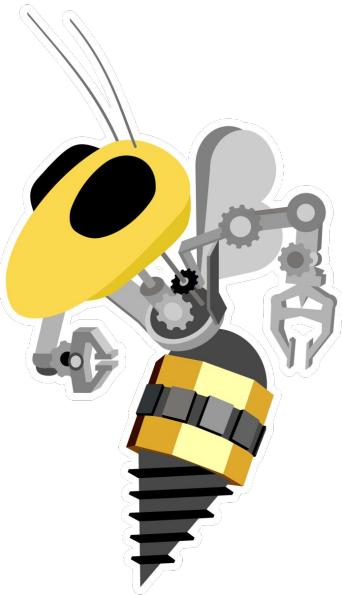
- Asynchronous (no clock needed)
- Uses 2 wires
- Need same baud rate



# USB (Universal Serial Bus)

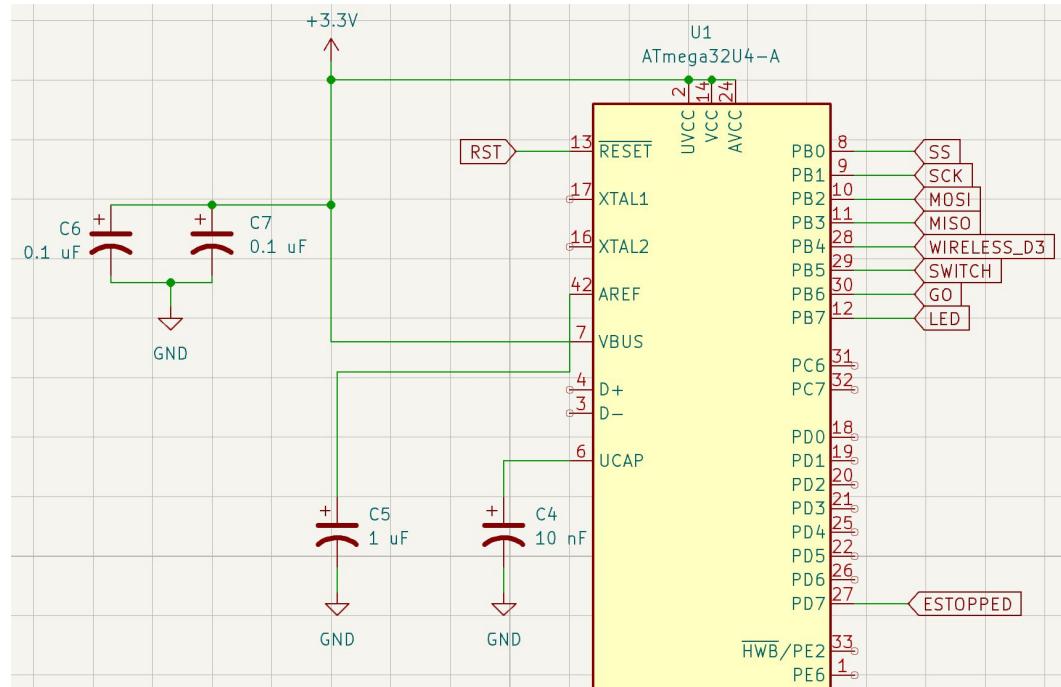
- Differential Pair signal
- Defines rate in “clock sync” phase
- Useful for computer-device communication





# KiCAD Cont.

# Goals by End of Today (Schematics)

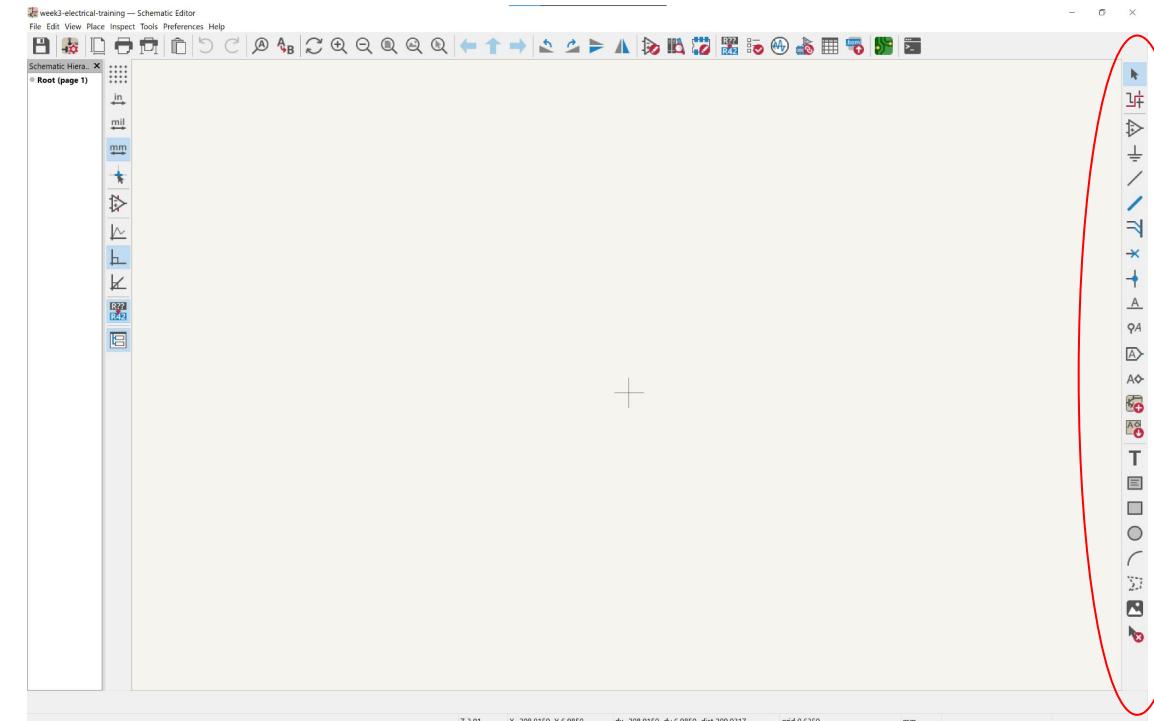


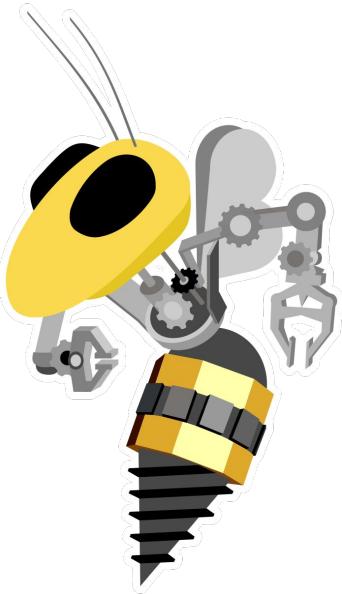


# Some Useful Features to Know

# SideBar Tools

*Most actions for drawing in the schematic are in the bar on the right (ex. add symbol, draw wire)*



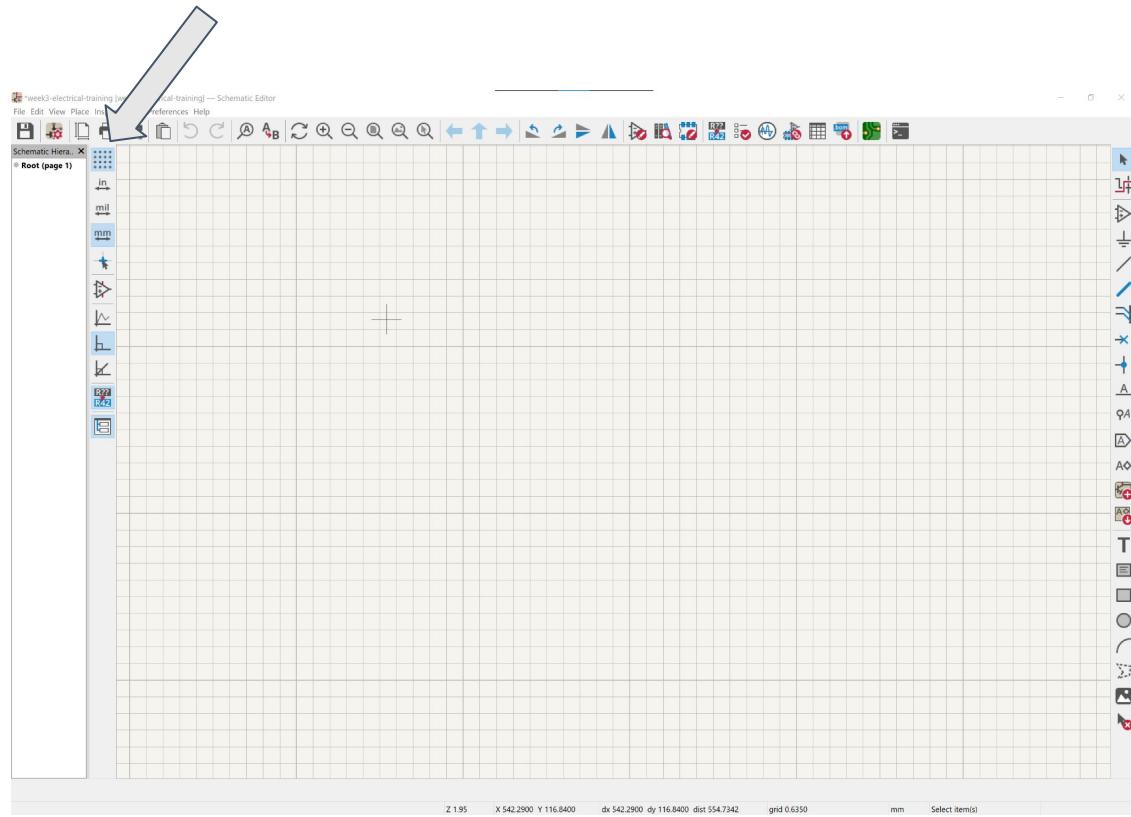


# How to Build your Own Schematic

# Step 1: Grid Lines

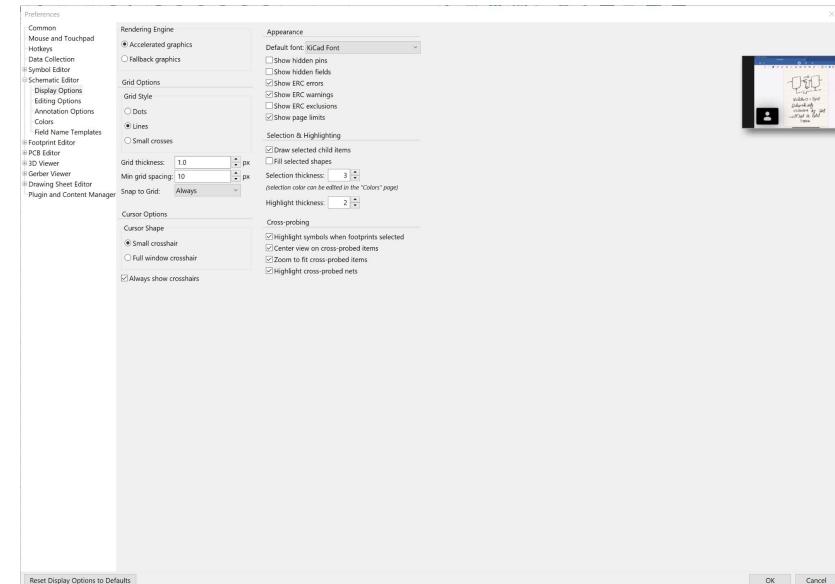
*Your screen  
should look  
like this.*

Make sure this is selected (blue)



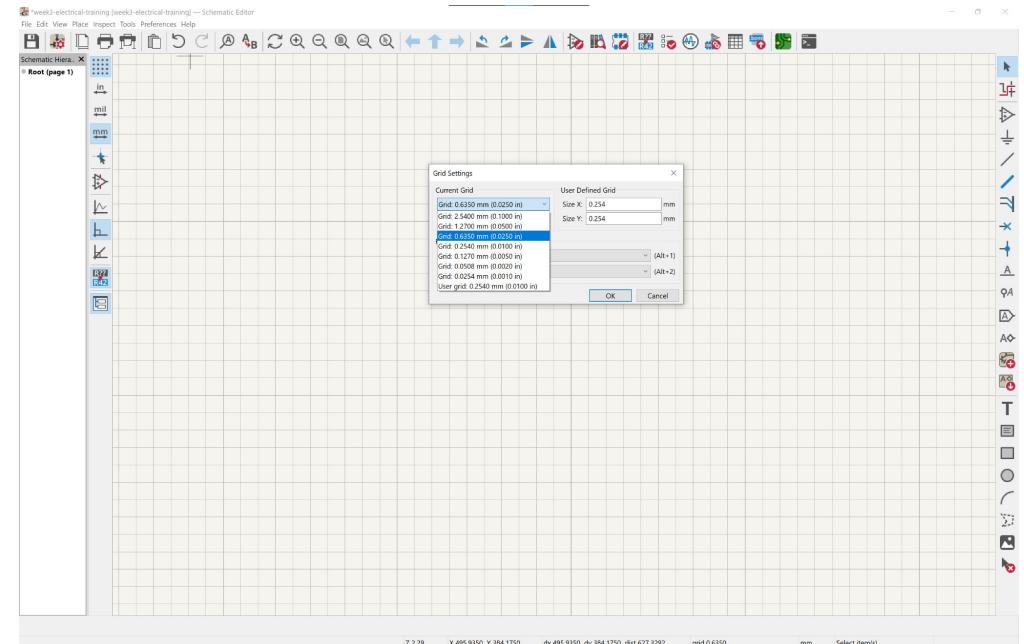
# Step 1: Grid Lines

- If nothing shows up, you may need to change your grid style to lines since dots is the default
- Go to Preferences > Preferences > Schematic Editor > Display Options
  - Change Grid Style to Lines
- The previously blank page should now have grids



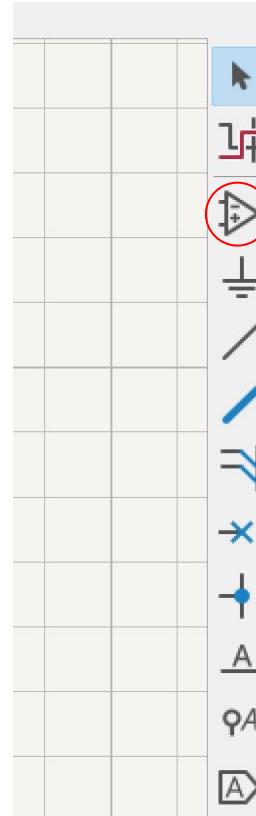
# Step 1: Grid Lines

- You can change the size of your grid in View > Grid Properties
- There are some predefined grid sizes, but you can also define your own
  - Make sure to select User grid to use your custom grid size



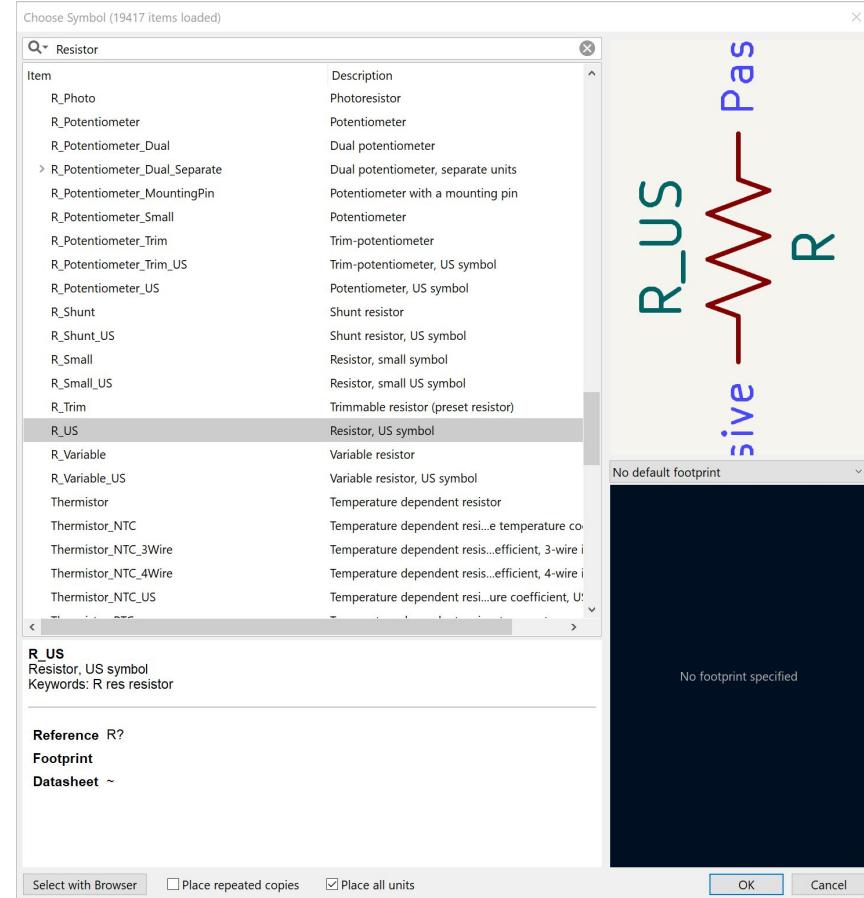
# Step 2: Adding Symbols

*Select the 'Add a  
Symbol' icon or  
press 'A'*



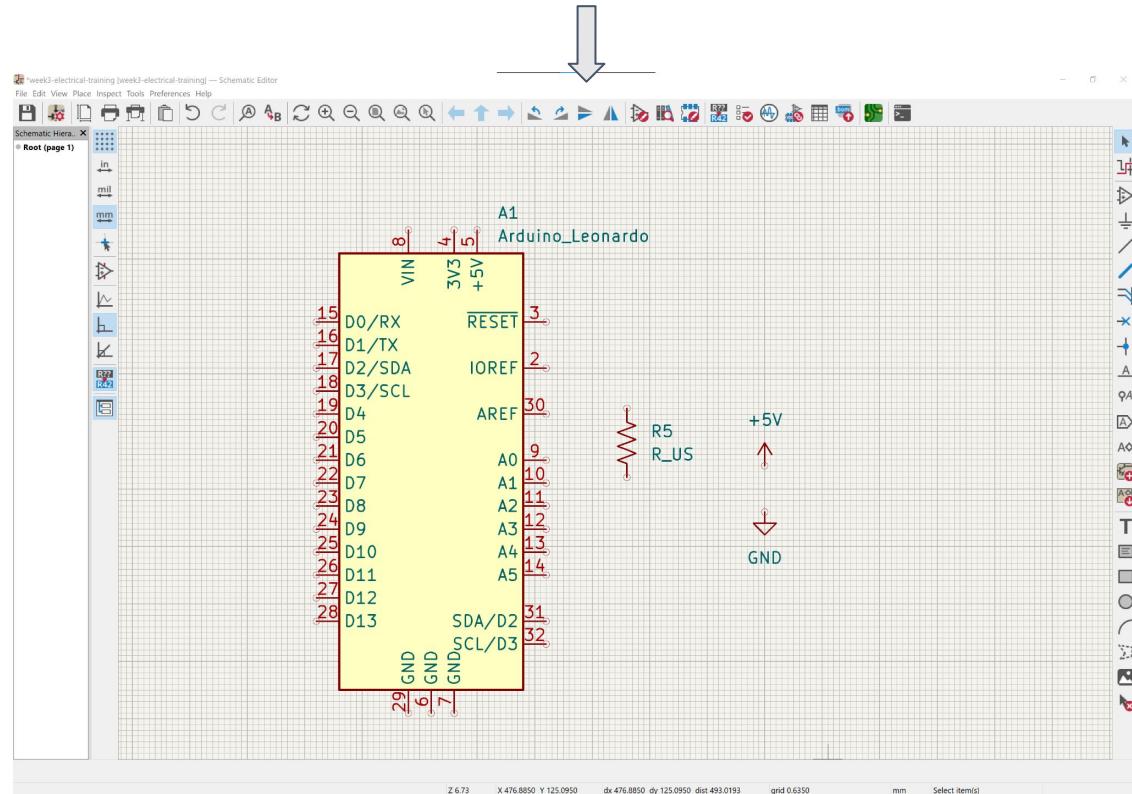
# Searching for Parts

*Use keywords  
to find what  
you're looking  
for (ex.  
resistor, relay)*



# Step 3: Rearranging Parts

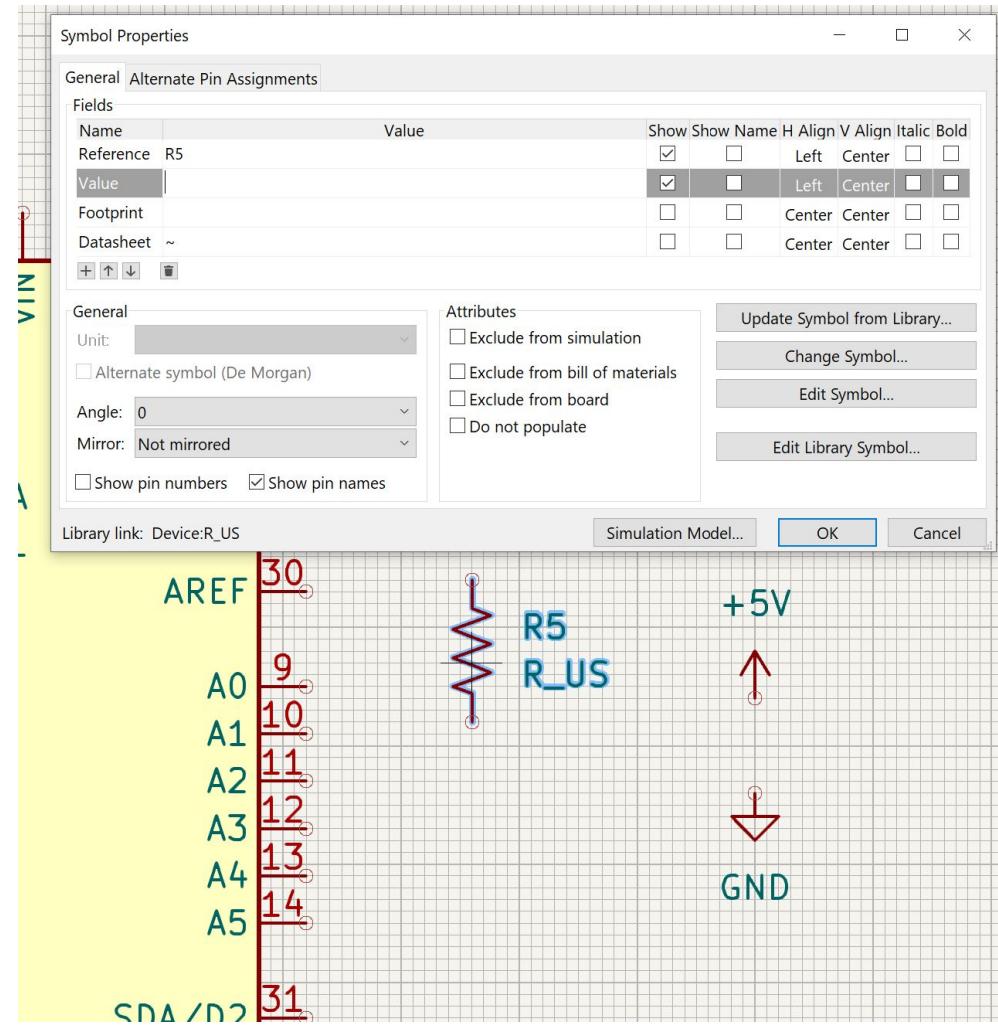
Click to rotate or flip



# Step 3: 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 by clicking on a part and dragging it to where you want to place it
- You can rotate your parts by clicking "Rotate" in the top bar or clicking on a part and pressing 'R' (CCW)

# Step 4: Setting Value of Parts

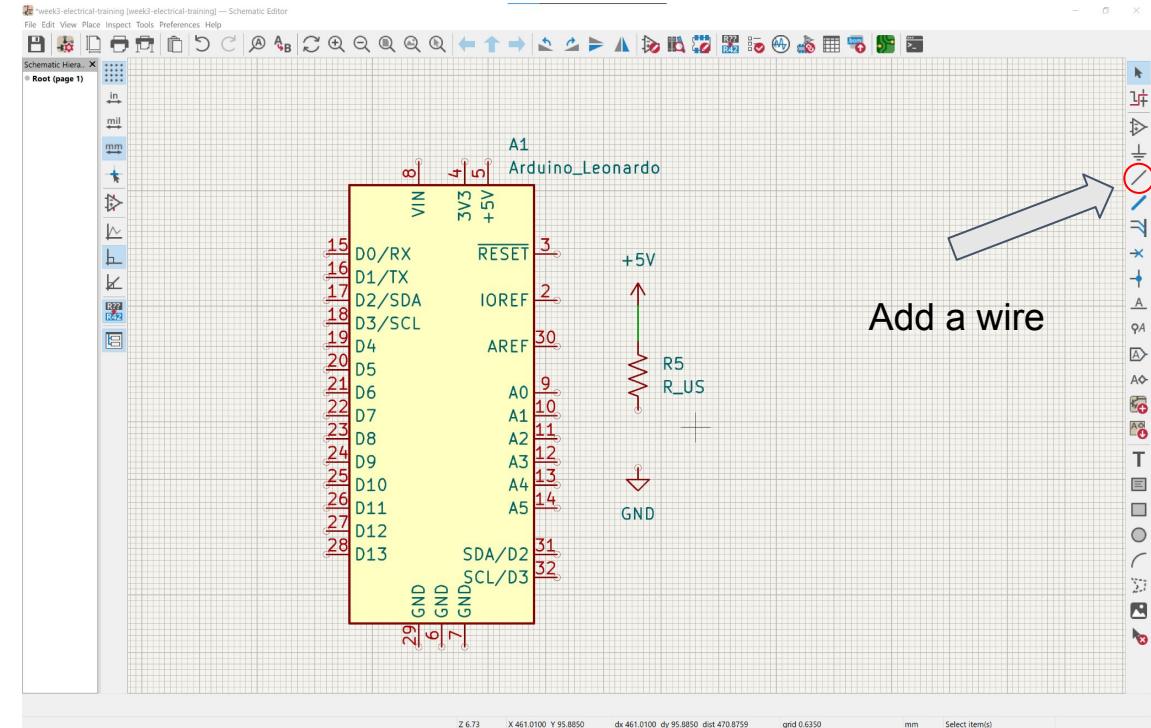


# Step 4: Setting Value of Parts

- Right click on a part and select ‘Properties’ to change a part’s value or left click on a part and press ‘E’ to open the Properties menu
- You can enter any text into the ‘Value’ field, but RJ convention is to just write a number (if possible, abbreviate like 10K) for resistors and include the units for capacitors and inductors

# Step 5: Adding & Deleting Nets

*Same as adding  
and deleting wires*

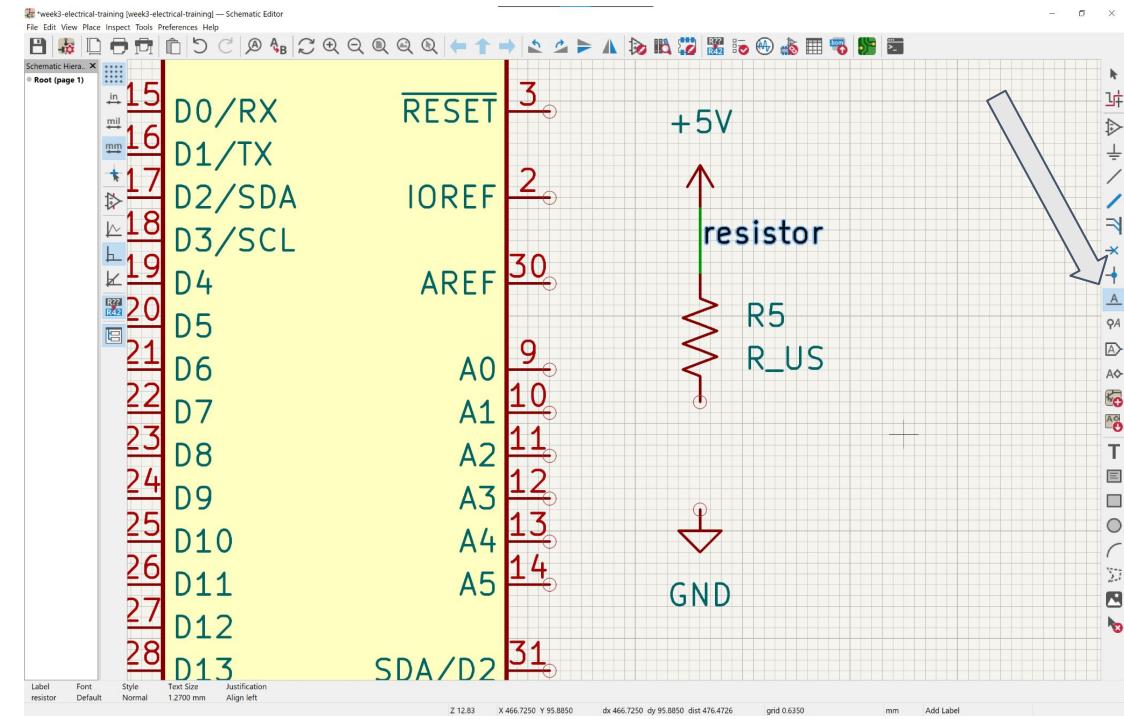


# Step 5: Adding & Deleting Nets

- To connect the parts, we need to add wires between them
- Click on ‘Add a wire’ in the right bar and click on the points you want to connect
  - If you want to stop drawing a wire, you can right click and select ‘Finish wire’ to keep what you’ve drawn so far
- You can also click on any unconnected pins of parts to start drawing a wire from that point

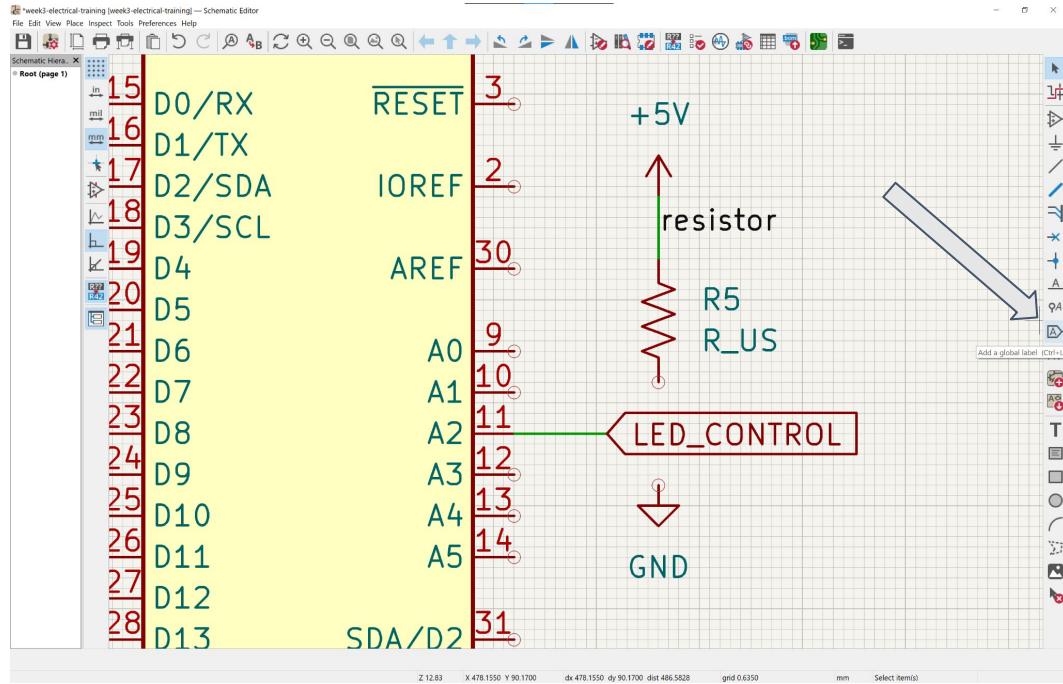
# Optional Step: Naming

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



# Optional Step: Label Settings

*Click the 'Add a global label' button, type a name for what the signal represents, and connect it to your desired net*

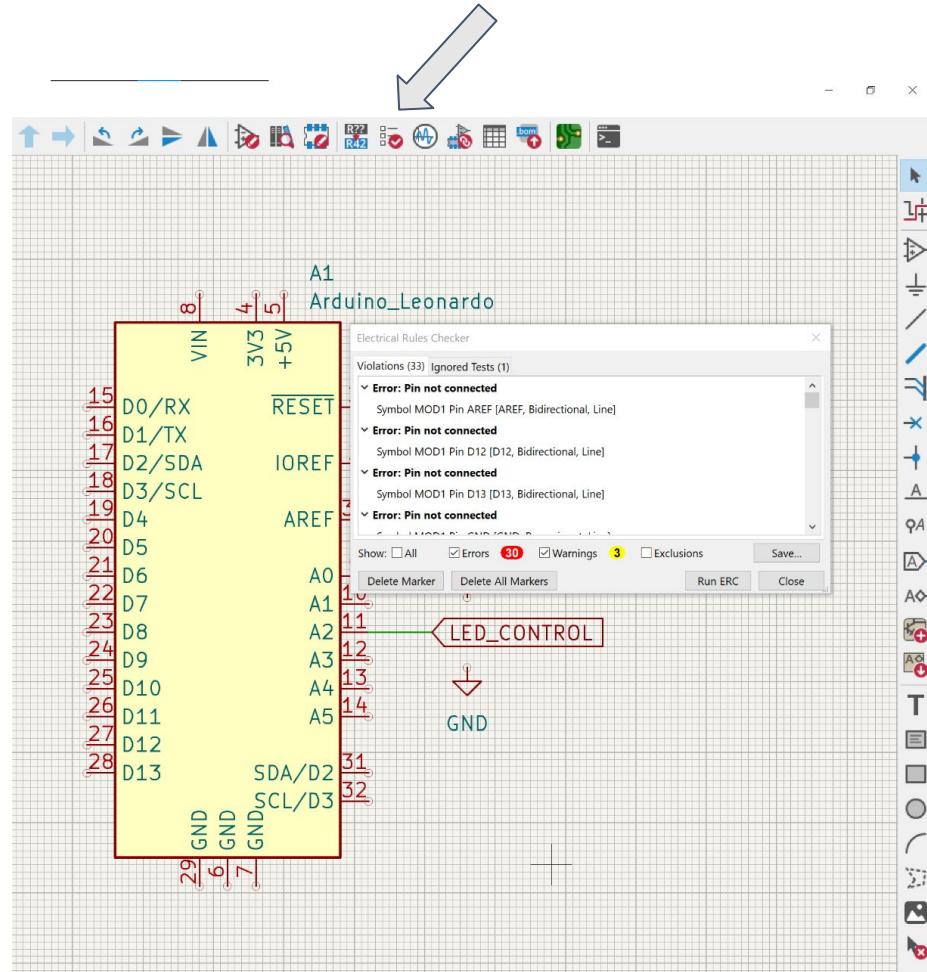


# Optional Step: Name and Label

- In certain scenarios, you may want to break up your schematic graphically to represent different functions
  - Using the global label tool helps with this so any wires connected to the same label are connected in the schematic even if they're not touching

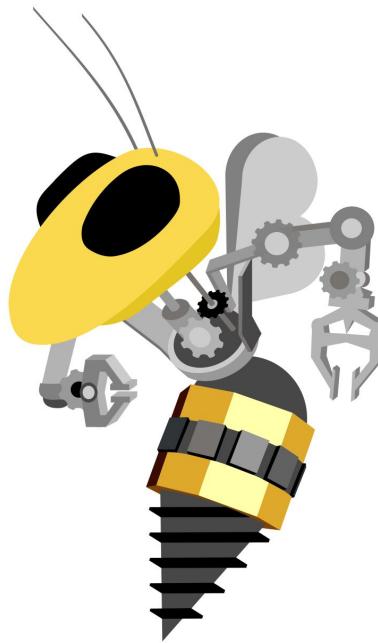
# Step 6: Electrical Rules Check (ERC)

*Check if there  
are errors or  
warnings.*



# Step 6: Electrical Rules Check (ERC)

- ERC is to ensure our schematic is good and all the connections are made
- Click on “Perform electrical rules check” in the top bar
  - You have to click on Run ERC and it will tell you how many warnings and errors you have
  - It will add arrows in your schematic pointing to all of the locations with issues to help you diagnose where they are



# Lab!

Creating a Schematic in KiCAD