Schematic Capture Workshop
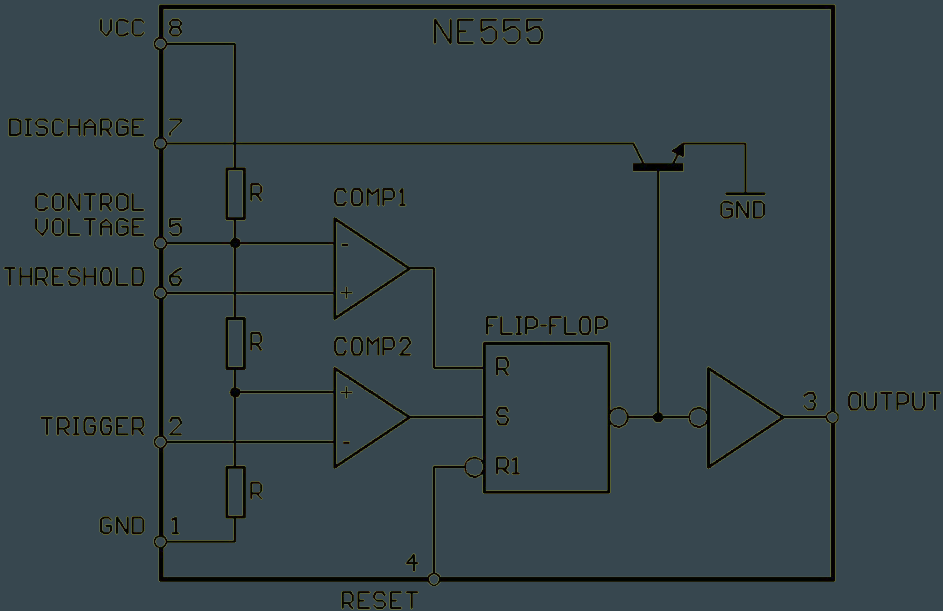## (Using KiCAD)

Hosted By:
NEU Wireless
NU IEEE

Please sign in:

kicad.org/download

Workshop Goals:

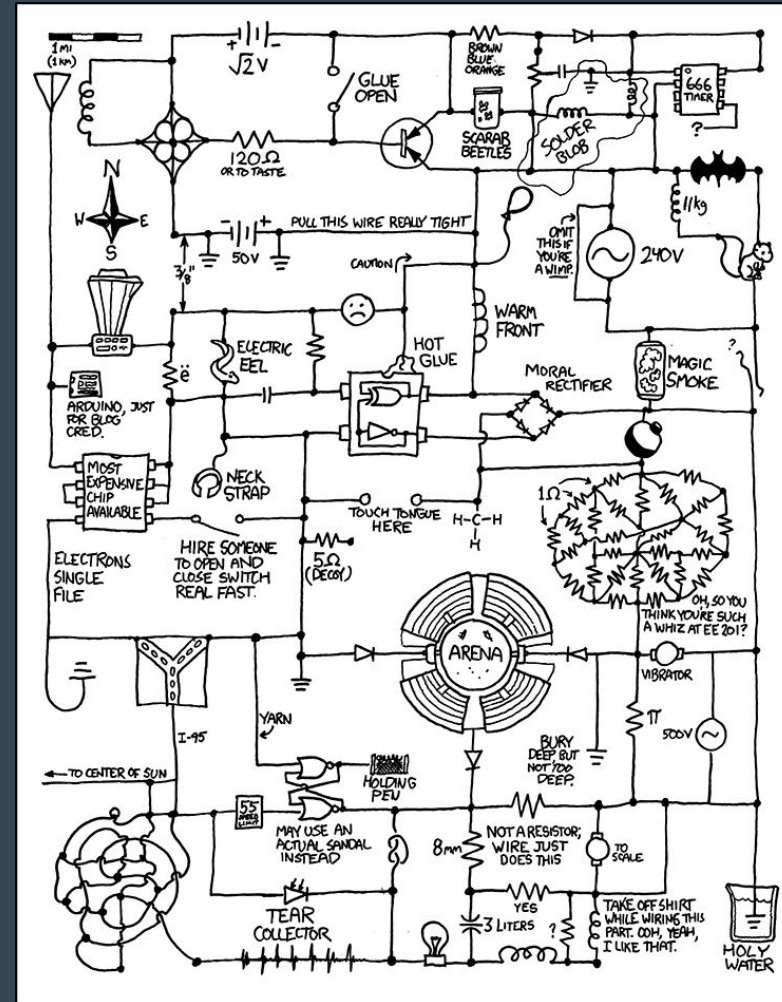- Learn to design a 555-timer based circuit schematic using KiCAD.


555 timer block schematic

What's a Schematic?

- A visual diagram with symbols and lines that shows how electronic components are electrically connected.
- Helps engineers plan and understand systems. Communication for engineers, data entry for PCB software.
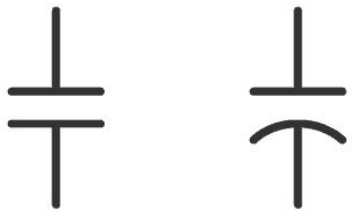- Think of it as a map for electronics.

KiCAD Project Setup

1. Open KiCAD
2. Click "Create A New Blank Project"
3. Name/Save your project
4. Click on the `.kicad_sch` file
5. You should see this:
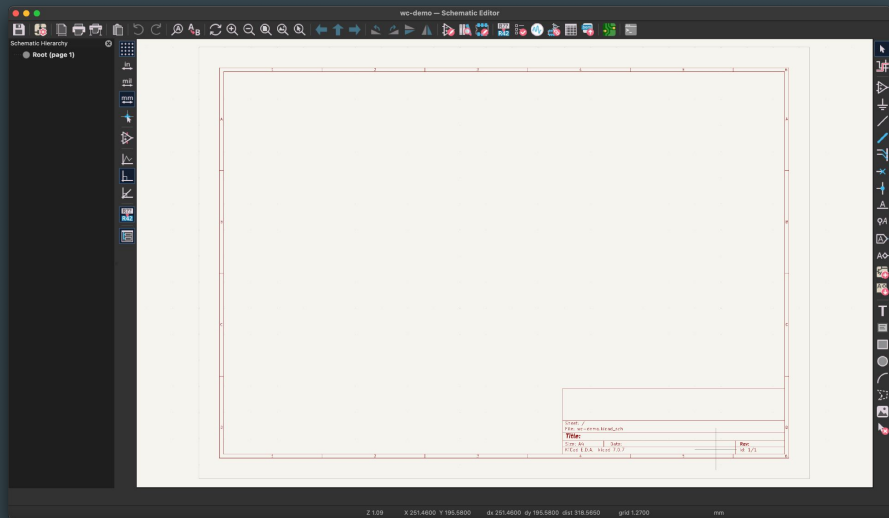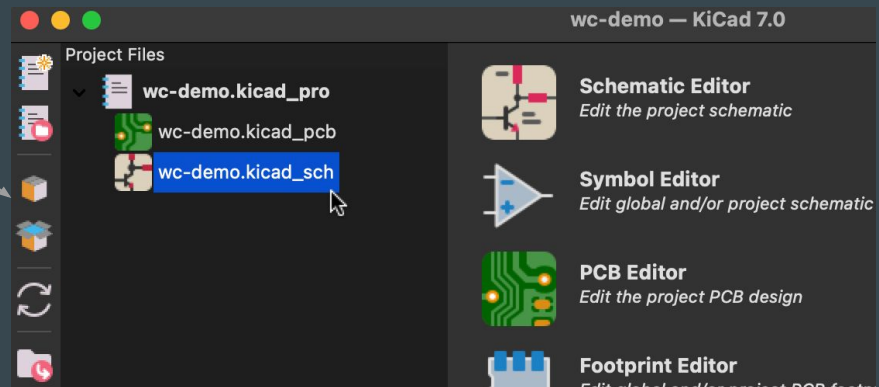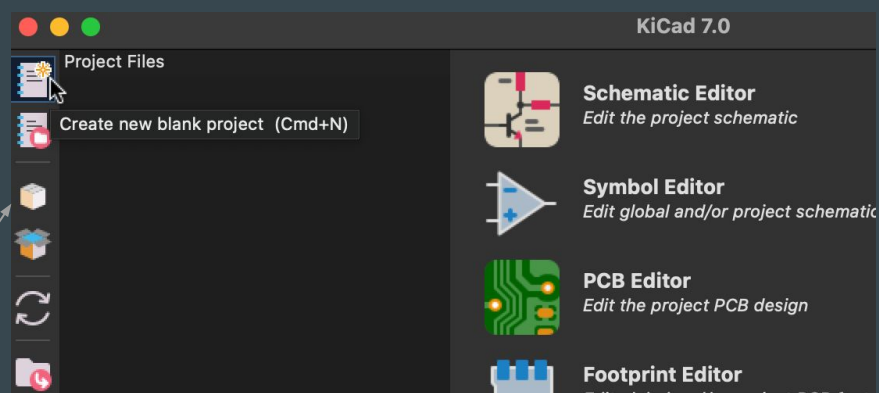
Schematic Symbols (ANSI + IEC)
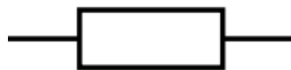

Non-polarized Capacitor

Polarized Capacitor

Battery

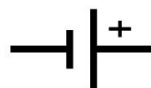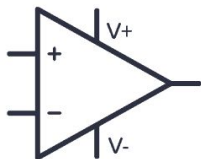American style resistor

European style resistor

Operational Amplifier

V+

V-

+

−

VDD

Power symbols for Ground and VDD
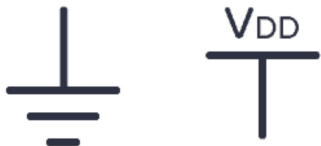
LED   Diode

Photodiode   Zener   Schottky

Diode

Capacitor

Inductor

Resistor

DC voltage source

AC voltage source

Place a Resistor

IEC (International)   ANSI (US)
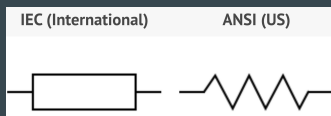
1. Press 'a' to bring up the <u>symbol library</u>
2. Type "R_US" to search for Resistor and click "OK"
   a. In this workshop, we'll utilize ANSI symbols, not IEC.
3. Press 'r' to rotate the resistor
4. Place the component by clicking on schematic

R1
R_US

'r'

R1
R_US
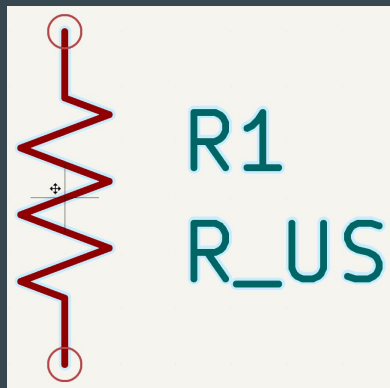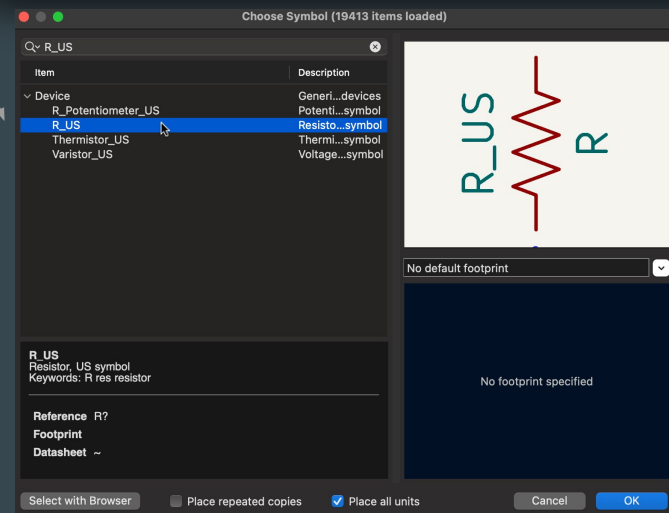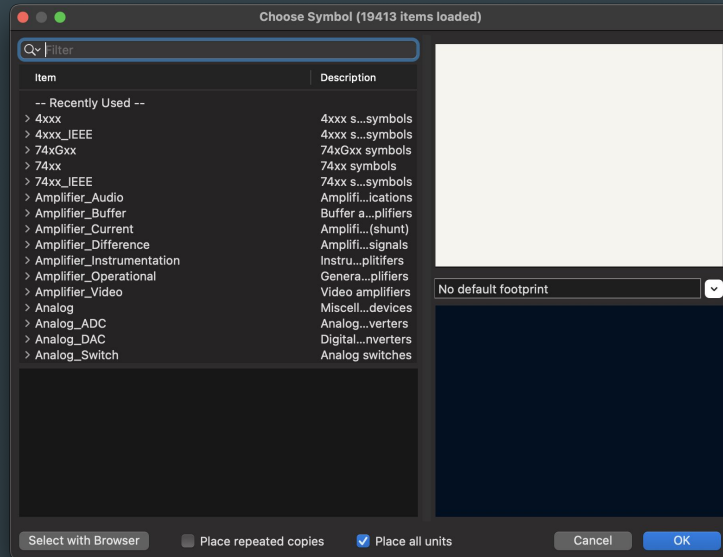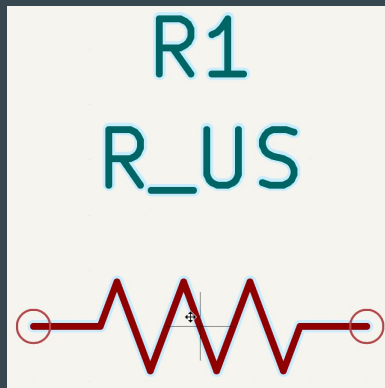
Schematic Connections (Wires)

- Represent a connection between components.
- A green dot ● represents a connection between two wires
- Wires that cross without that dot are not connected.
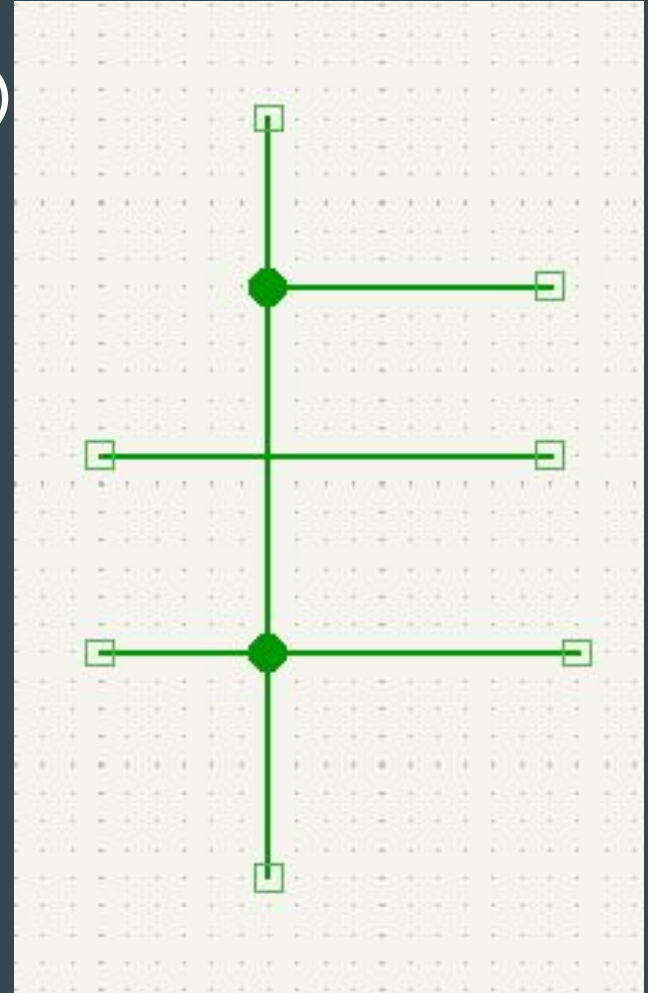
Place a Wire

1. Press 'w' to start a wire. If you messed up, hit "escape" to reset your wire start point.
2. Move the mouse to draw
3. Press 'k' to end a wire

Image: Resistor with a wire →

R1
R_US
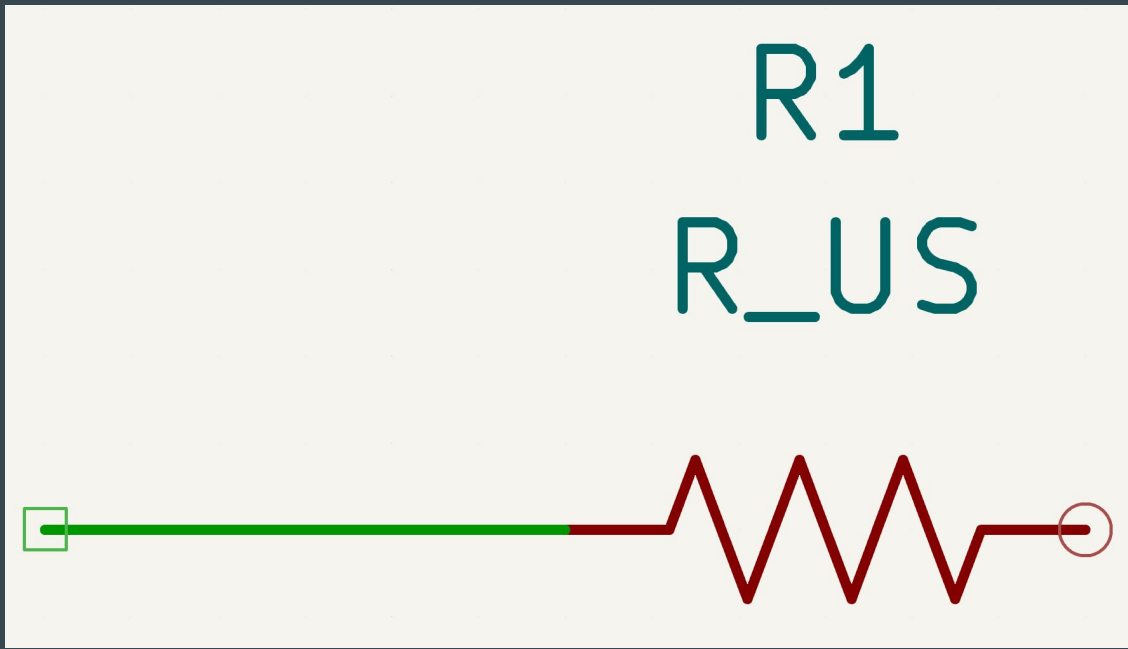
Power Ports

- Power ports represent a common connection
- Think of it like a wire connecting all of the same ports
- Typically used for nodes that always have the same voltage, like +5V or GND

+5V
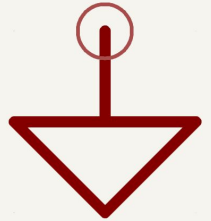
GND

Place a Power Port

1. Press 'p' to bring up the <u>power symbol library</u>
2. Type "+5V" and click "OK"
3. Place the component at the end of the wire
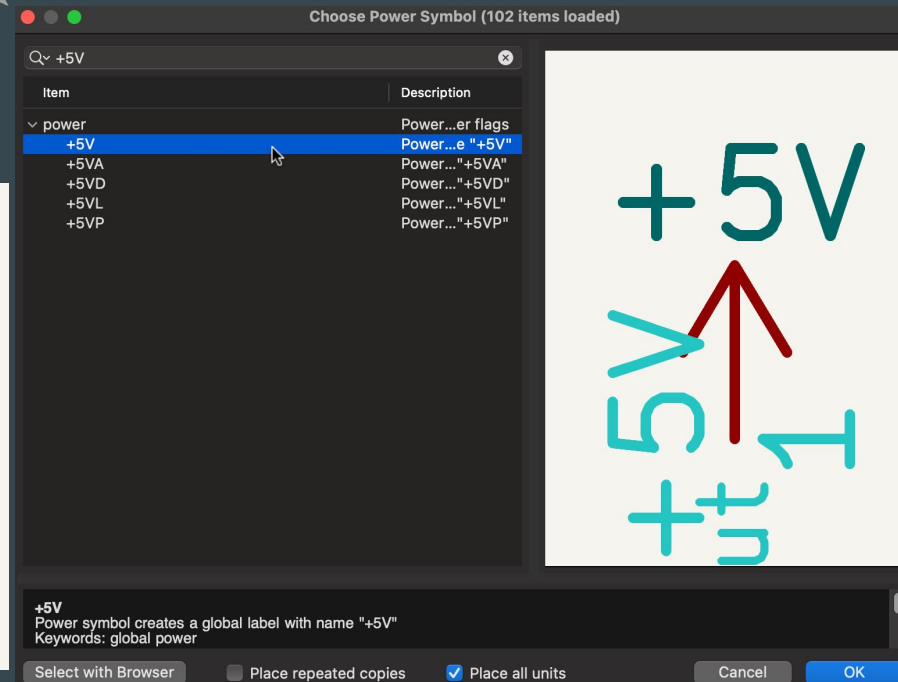
Component Values

- Component values are essential parameters defining each component. Without them, your schematic is useless.
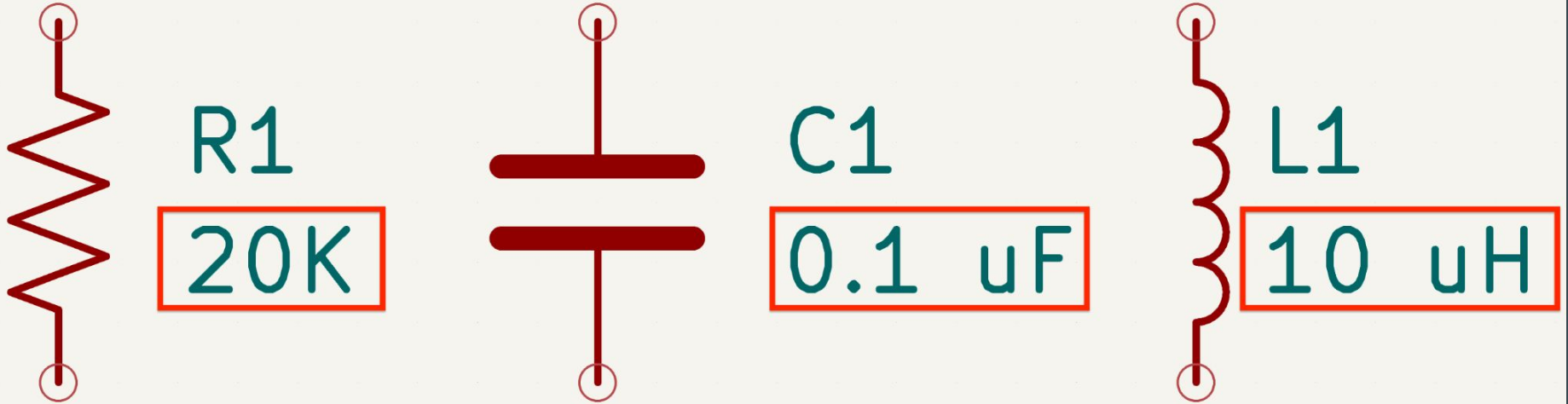
Useful Hotkeys

- 'm' to pick up a component, click to drop it
- 'g' to drag a component (wires stay connected)
- "del"/"delete" to delete a component
- 'x/y' to flip a component while it is being moved
- 'q' for the "No Connect Flag" (blue X symbol on schematic)

All hotkeys have a corresponding button on the top/right of your screen

Most tools in KiCad either have default hotkeys assigned, or can have custom hotkeys assigned. To view all hotkeys, go to Help → List Hotkeys…. Hotkeys can be changed in Preferences → Preferences… →

Put it all together
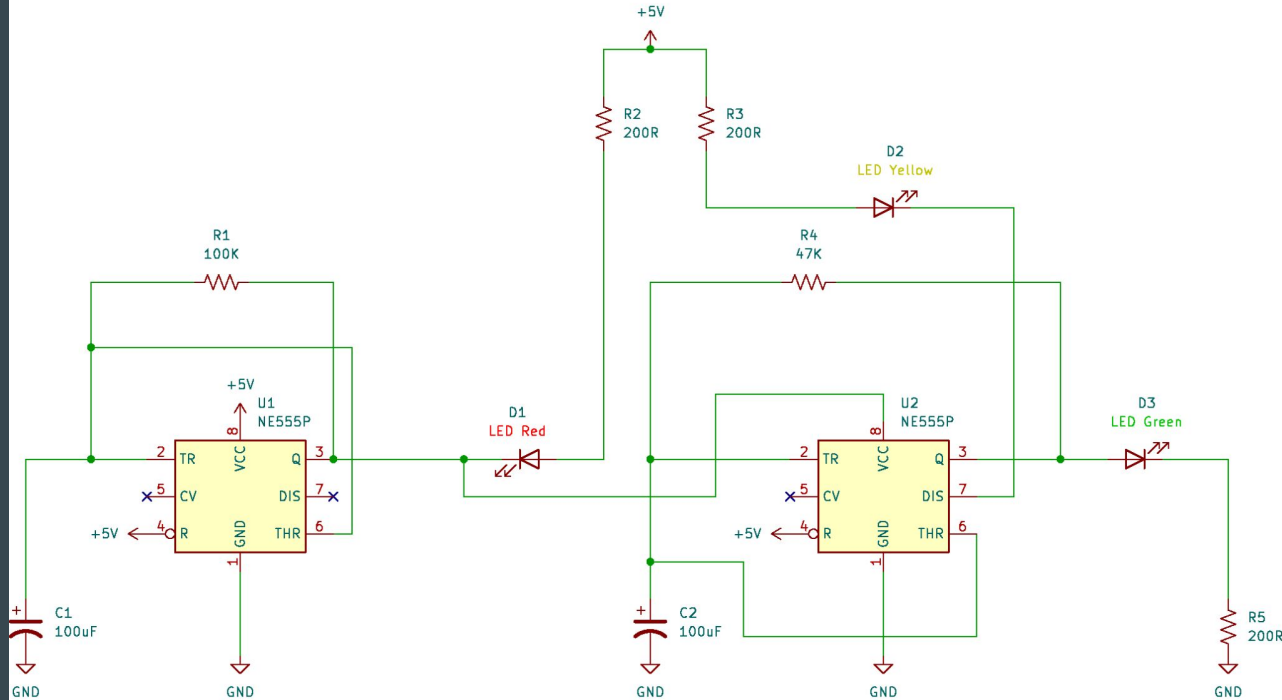
Press 'q' on your keyboard to access the blue X, which signifies the "No Connect Flag."

Check your work

Component Count:	12						
Ref	Qnty	Value	Cmp name	Footprint	Description	Vendor	DNP
C1, C2	2	100uF	C_Polarized_US		Polarized capacitor, US symbol		
D1	1	LED Red	LED		Light emitting diode		
D2	1	LED Yellow	LED		Light emitting diode		
D3	1	LED Green	LED		Light emitting diode		
R1	1	100K	R_US		Resistor, US symbol		
R2, R3, R5	3	200R	R_US		Resistor, US symbol		
R4	1	47K	R_US		Resistor, US symbol		
U1, U2	2	NE555P	NE555P	Package_DIP:DIP-8_W7.62mm	Precision Timers, 555 compatible, PDIP-8		

Questions?

{elarbi.m, annapragada.s, aviedov.v}@northeastern.edu

References/Attributions

- https://upload.wikimedia.org/wikipedia/commons/2/2e/555_esquema.png
    - Own work based on: NE555 astable.png, CC BY-SA 3.0 <http://creativecommons.org/licenses/by-sa/3.0/>, via Wikimedia Commons
- https://www.kicad.org/img/kicad_logo_paths.svg
    - KiCad Developers Team, GPLv3 <http://www.gnu.org/licenses/gpl-3.0.html>, via Wikimedia Commons
- KiCAD Screenshots by Muhammad Elarbi, 2023, licensed under CC BY-SA 4.0.