IEEE UCF SKILL SERIES
PCB Design

Part 1: Intro to KiCAD and Schematic Capture
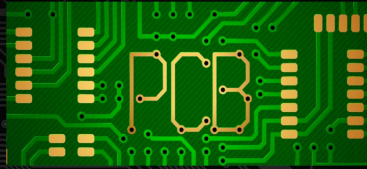
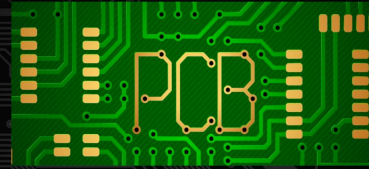**College of Engineering and Computer Science**

What You Will Learn

I. Overview of PCB design
II. KiCAD schematic and layout editor tools
III. Best practices for component selection, placement, and routing
IV. Design Rules Check (DRC)

Schedule

I. Workshop 1 (Today):
  A. What is a PCB?
  B. Setting up KiCAD
  C. Schematic capture
  D. Choosing components
II. Workshop 2 (10/6):
  A. PCB layout editor
  B. Routing traces
  C. Design Rule Check (DRC)
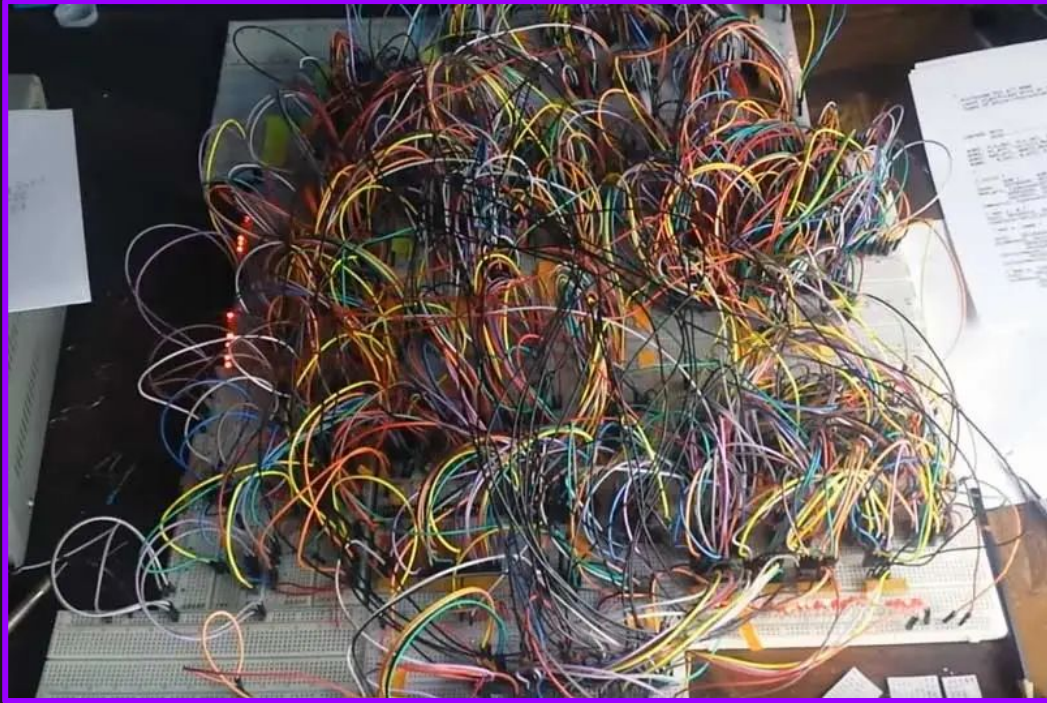
College of Engineering and Computer Science

What You'll Be Making

RGB LED strip lighting controller

- 3 analog dials for adjusting color/brightness
- IR sensor for remote control of LEDs
- Powered by a micro USB cable from a 5V wall outlet
- ON/OFF switch

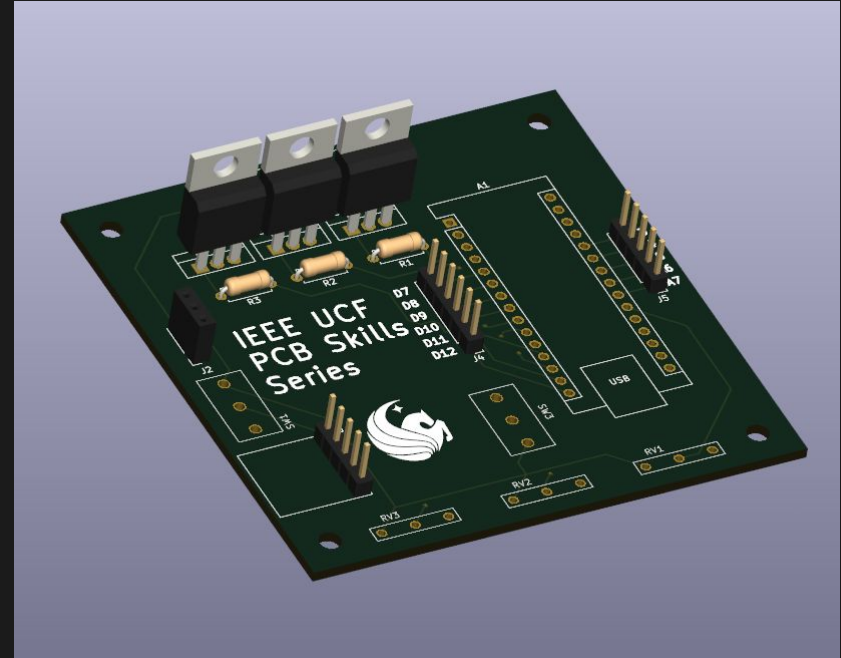# Motivation: Why make a PCB?

What is a PCB?

"Printed Circuit Board"

- Useful when circuit has too many connections to be made by hand
- Physical wire connections are replaced by conductive channels, or "traces"
- Components can be mounted through holes or pads on the PCB

Getting Started

KiCAD, pronounced Key-CAD is an EDA, or Electronic Design Automation tool.

EDA Tools are used to create electrical schematics and printed circuit boards, which can then be sent to fabs such as JLCPCB or OSHPark to be manufactured.

We will be using KiCAD for this skills series because it is free and open-source, user-friendly, and comprehensive in its schematic capture and layout features.

Downloading and Installing KiCAD

Download KiCAD:
https://www.kicad.org/download/

Github for component library:

https://github.com/cbrynds/IEEE-UCF-KiCAD-Skills-Series

Optional Getting Started:

https://docs.kicad.org/7.0/en/getting_started_in_kicad/getting_started_in_kicad.html

Useful Schematic Editor Keyboard Shortcuts

Schematic Editor:

- A: place a symbol
- W: draw a net
- L: add a label
- Q: add a no connection flag
- T: add text
- V: edit value
- U: edit reference designator

- E: Properties
- R: rotate symbol
- G: drag
- M: move

More shortcuts:
https://www.siue.edu/~gengel/ece3
26WebStuff/KiCAD-hotkeys.pdf

College of Engineering
and Computer Science

Common Terminology

- **Schematic Editor** - create symbolic representation of circuit
- **Layout Editor** - create physical PCB layout
- **Net** - signal wire connecting two circuit nodes
- **Symbol** - representation of circuit element
  - Reference Designator - unique identifier for circuit component
  - Value - resistance, capacitance, etc
  - Footprint - bounding box for component to be placed on PCB
- **Label** - links two nodes without drawing a physical net
- **Electrical rules check (ERC)** - ensures that all circuit nodes are properly linked by nets

What is a schematic?

- "Schematic" - circuit blueprint
- A graphical representation of the circuit
- All component-to-component connections are shown
- The position of symbols does not matter
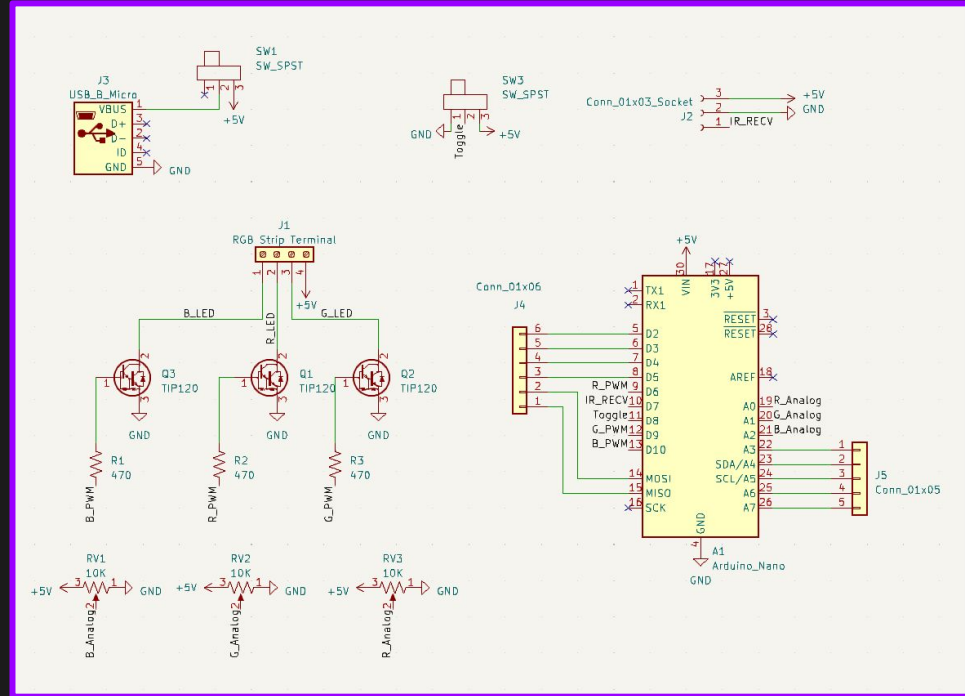
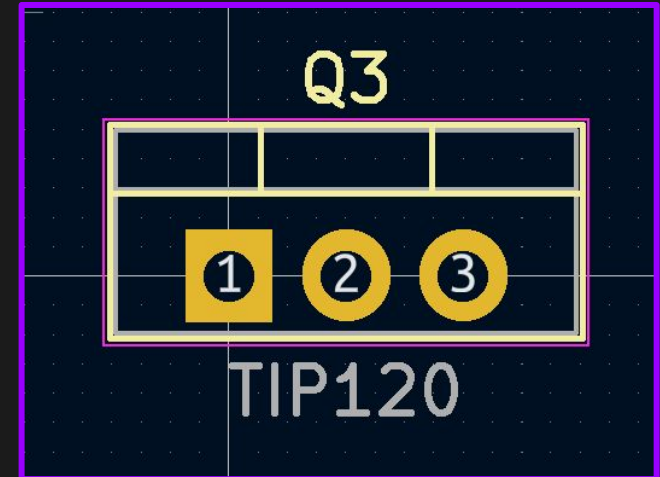# Symbols vs Footprints

- A symbol is the graphical representation of a component, while a footprint is physical component on the PCB
- A footprint has defined dimensions, pad sizes, and other manufacturing info
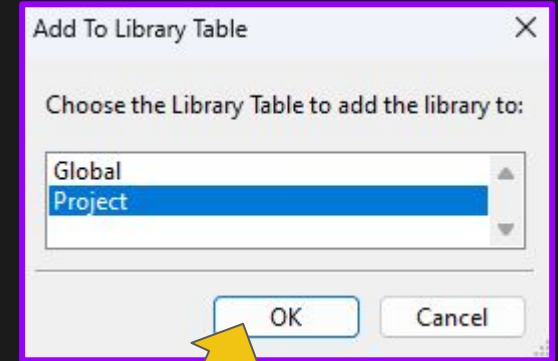- A symbol has pin connections and generic info about the component

Symbol

Footprint

College of Engineering and Computer Science
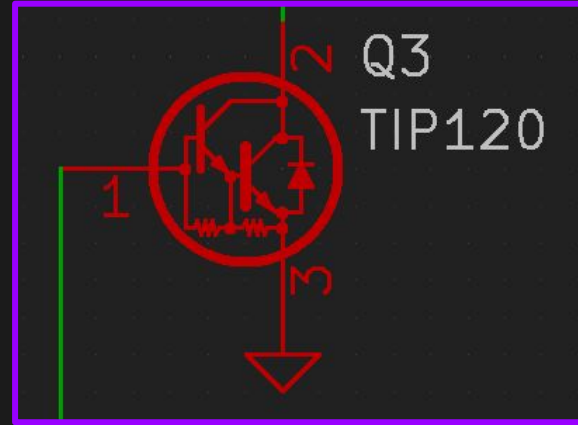
Getting started: Creating a Project

Getting Started: Importing Symbol Library

Symbol Editor
Edit global and/or project schematic symbol libraries

File	Edit	View	Place	Inspect	Preferences

New Library...
Add Library...
Save Library As...   Ctrl+Shift+S
New Symbol...   N
Save   Ctrl+S
Save As...
Save All
Revert
Import Symbol...
Export   >
Symbol Properties...
Close

Add To Library Table   ✕

Choose the Library Table to add the library to:

Global
Project

OK   Cancel

College of Engineering
and Computer Science

Getting Started: Importing Footprint Library

Footprint Editor
Edit global and/or project PCB footprint libraries

New Library...

Add Library...

New Footprint...   Ctrl+N
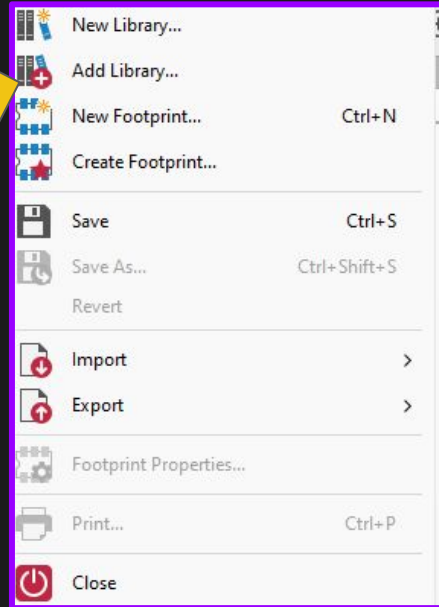
Create Footprint...

Save   Ctrl+S

Save As...   Ctrl+Shift+S

Revert

Import   >

Export   >

Footprint Properties...

Print...   Ctrl+P

Close

Add To Library Table   ×

Choose the Library Table to add the library to:

Global
Project

OK   Cancel

IEEE
UCF

UCF College of Engineering and Computer Science
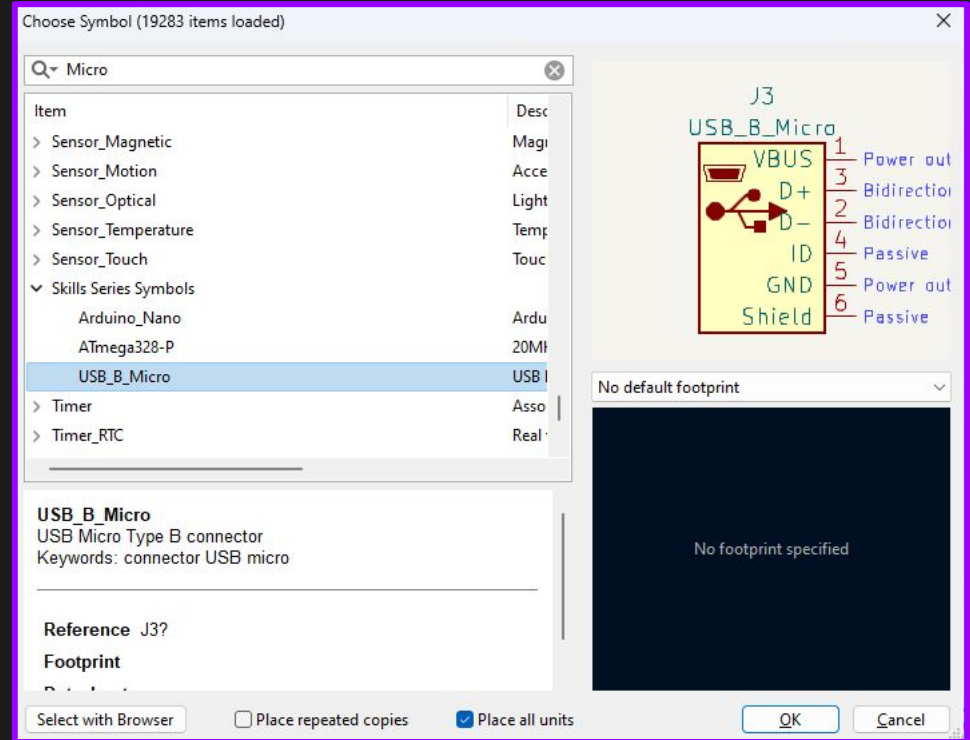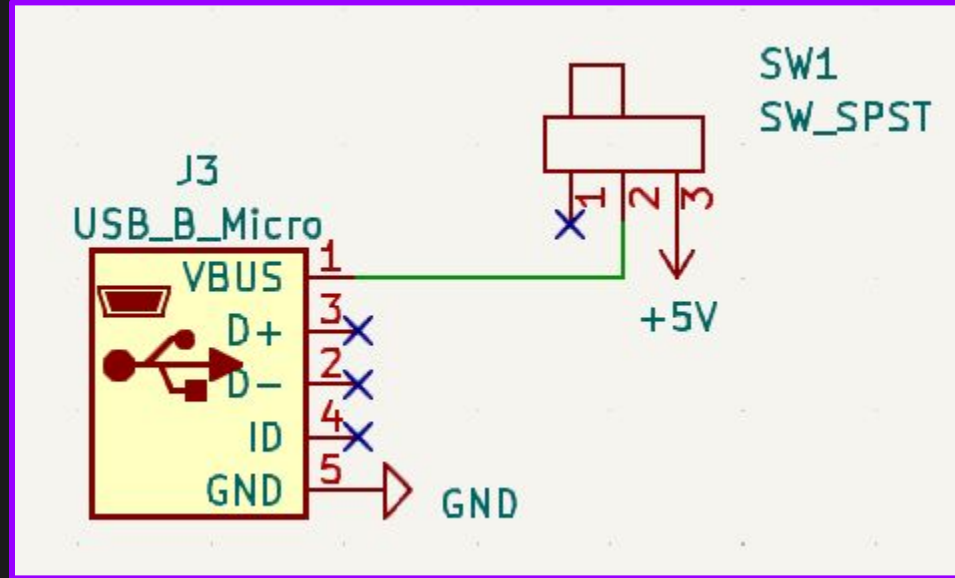
Getting started: Placing Symbols

- Access symbol menu by 'A' hotkey or place toolbar -> Add Symbol
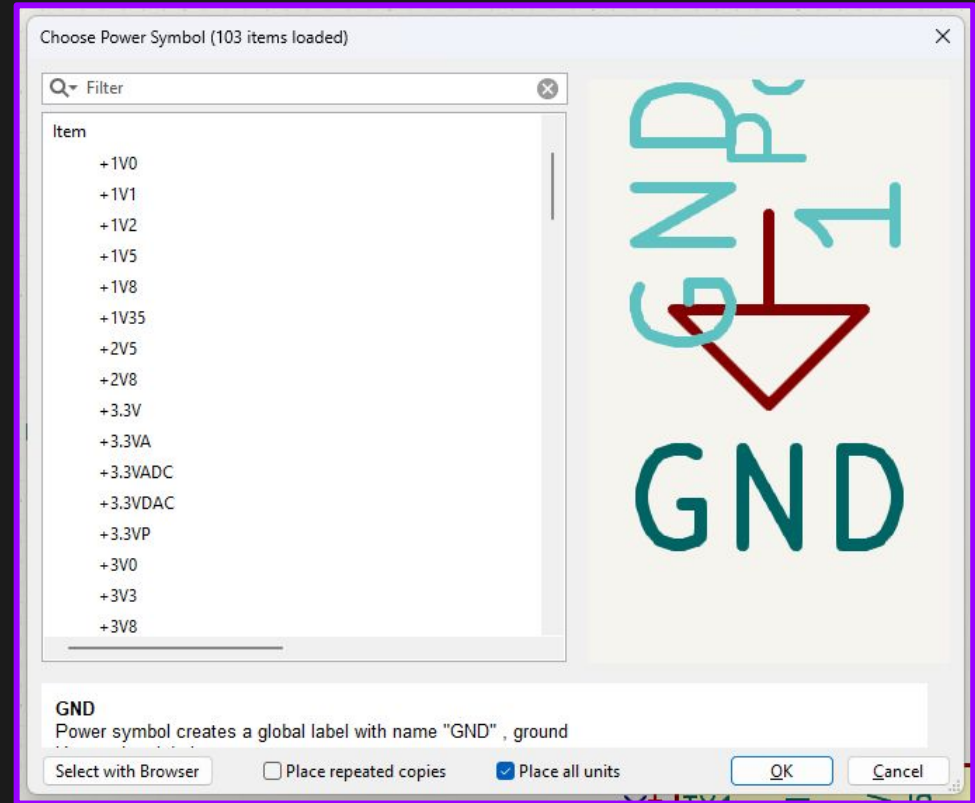- Can search by symbol name or attributes
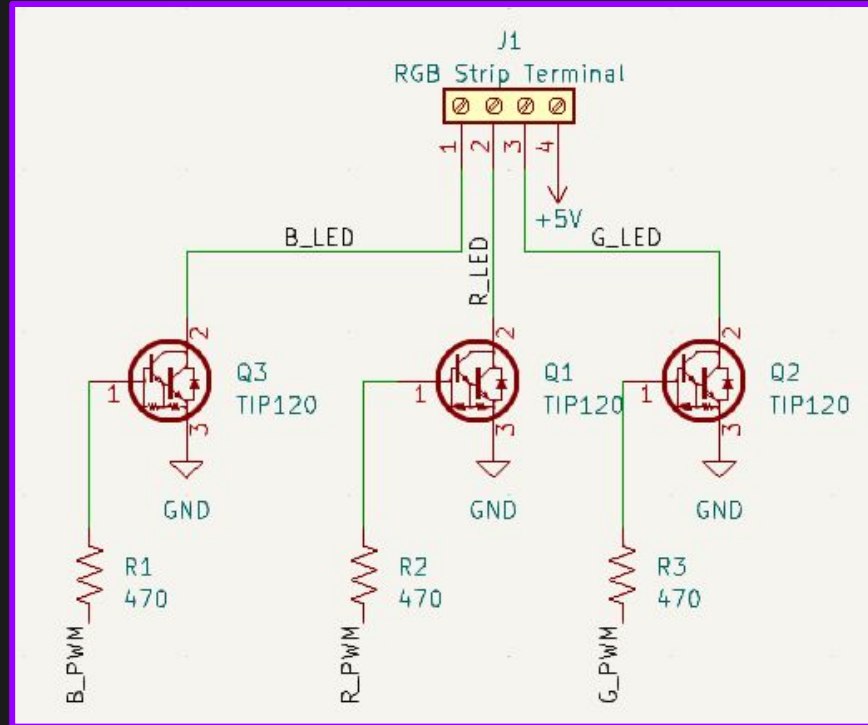
Creating Power Circuit

Power and Ground Symbols

- Access power menu by 'P' hotkey or "place" toolbar -> Add Power
- Various symbols for power and ground
  - +-X V, VCC, VDD
  - Earth Ground, Chassis Ground, Digital Ground, etc.

Creating Lighting Controller Circuit

Editing Component Values/Ref. Designators

- **A Few methods**
  - Hover over symbol, press V (or U for ref designator)
  - Double click on component, edit value/RD field
  - Right click on component -> edit main fields -> edit value/RD

College of Engineering and Computer Science
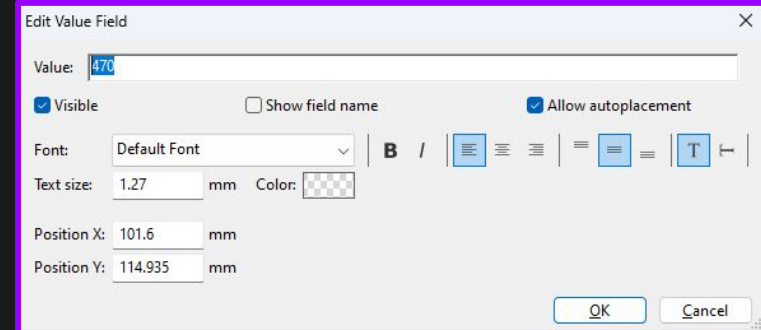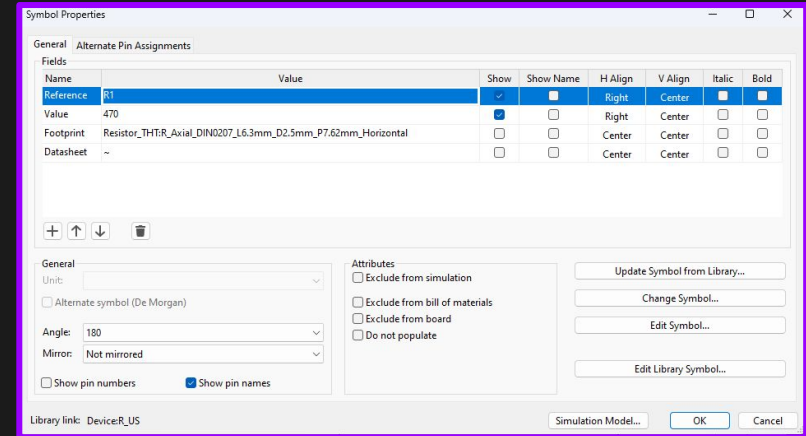
Making Connections: Pin Labels
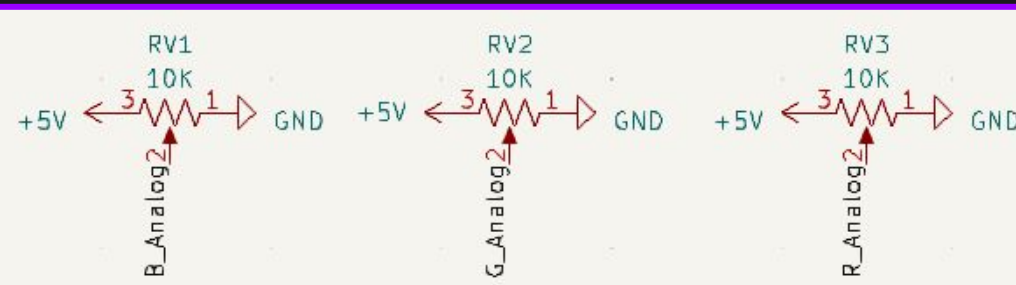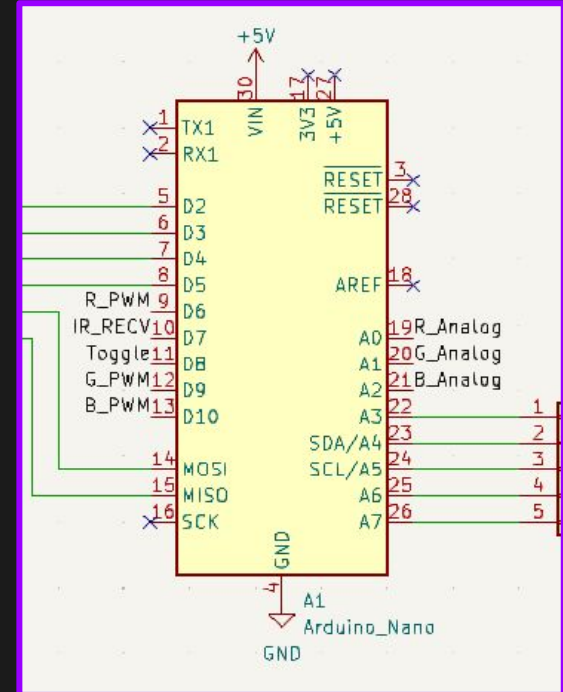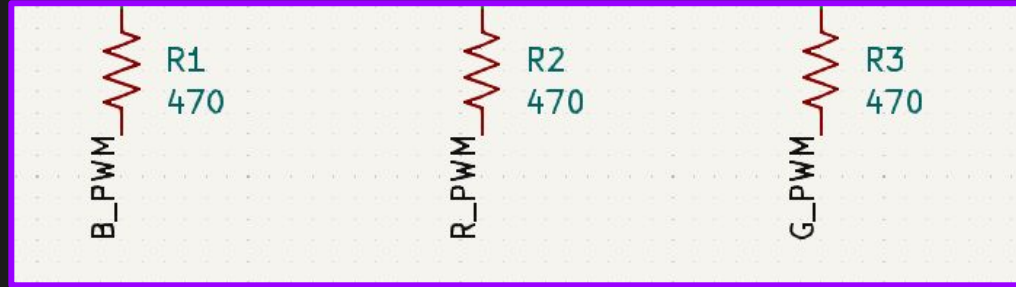
Creating Potentiometer Circuit
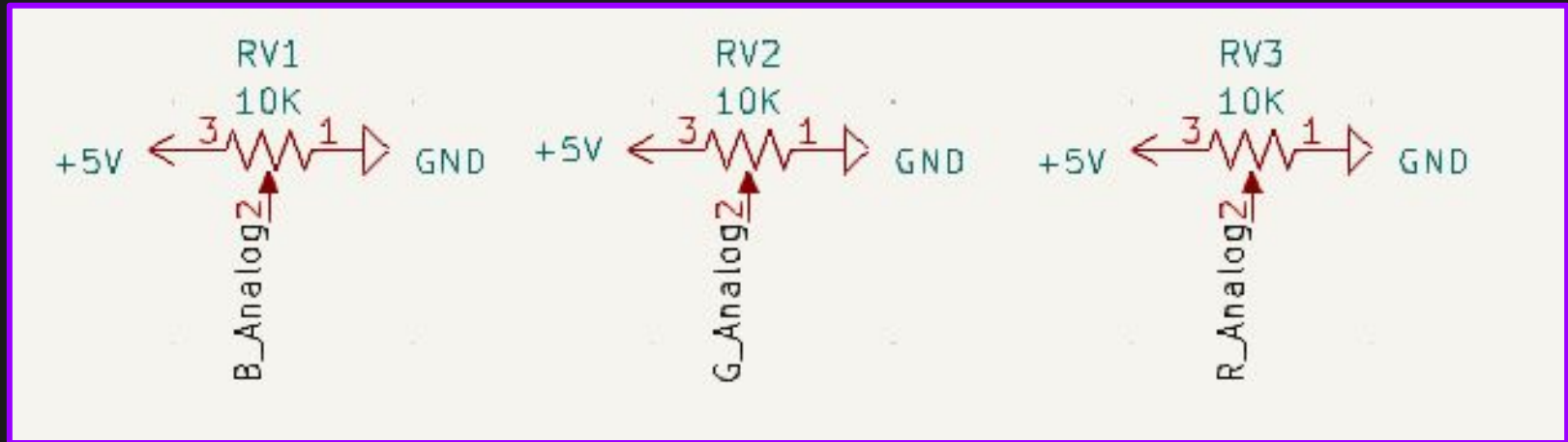
Toggle Switch and IR Receiver Circuits

Arduino Nano: Circuit vs Symbol
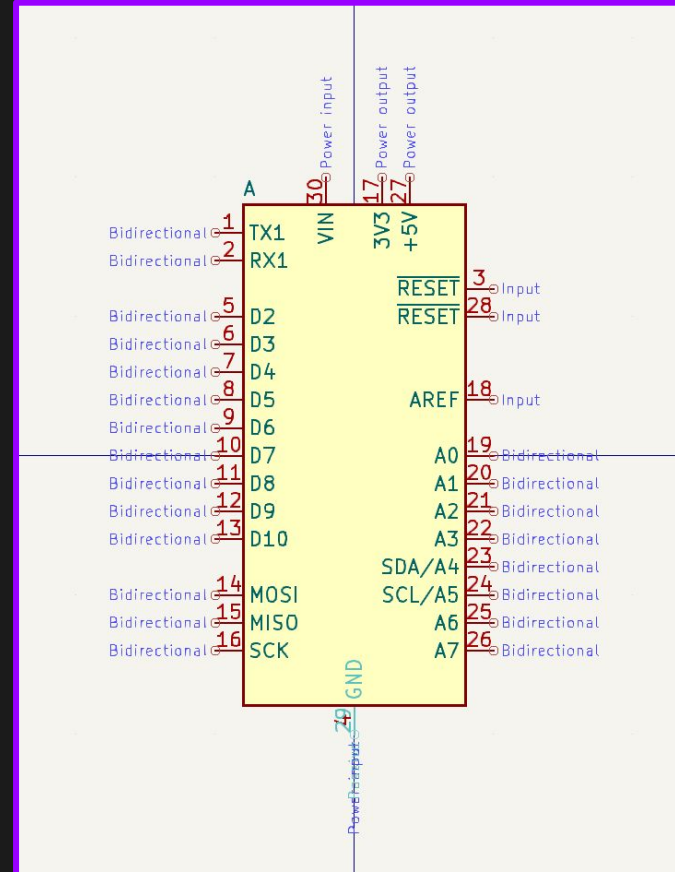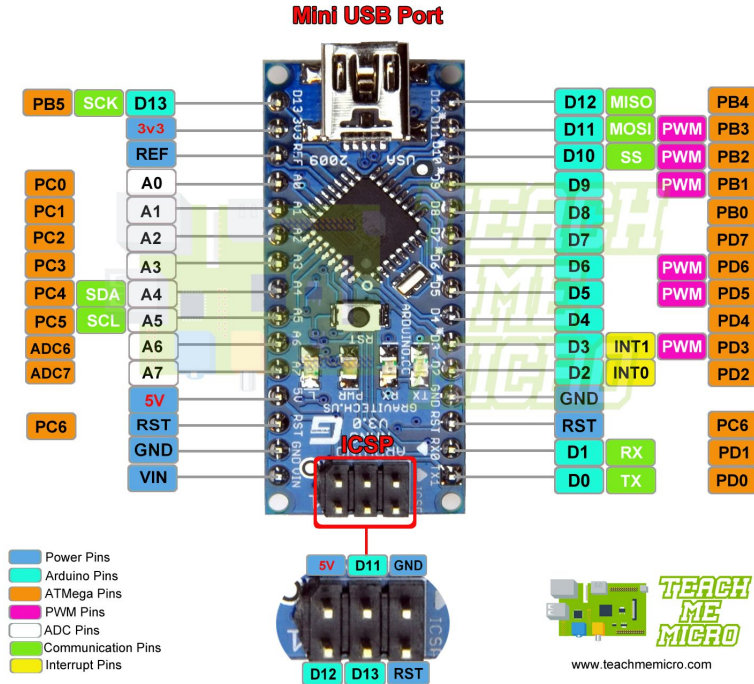
Creating Arduino Nano Circuit
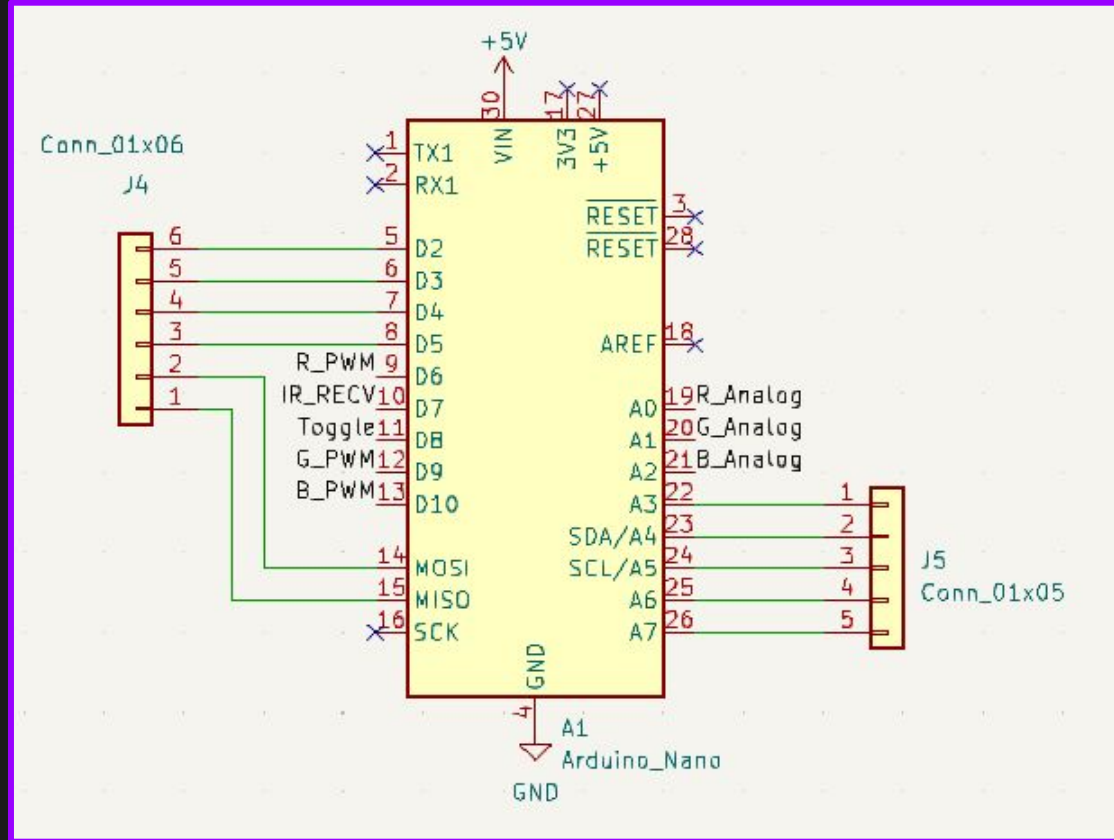
Reference Designation

- Reference designator: unique tag to identify component type and number
  - R1, R2, ... for resistor
  - C1, C2, ... for capacitor
  - U1, U2, ... for ICs
  - Etc
- Links symbol to footprint in layout editor
- Used to identify component locations when assembling PCBs
- Automatic designator assigns RDs to schematic symbols

Electrical Rules Check

- Inspect -> ERC -> Run ERC
- Ensures that schematic doesn't violate any electrical rules
  - I.e. there are no nodes missing connections
- May receive error: power input pin not driven by any output power pins
  - Solution: place a "Power Flag" on the +5V net
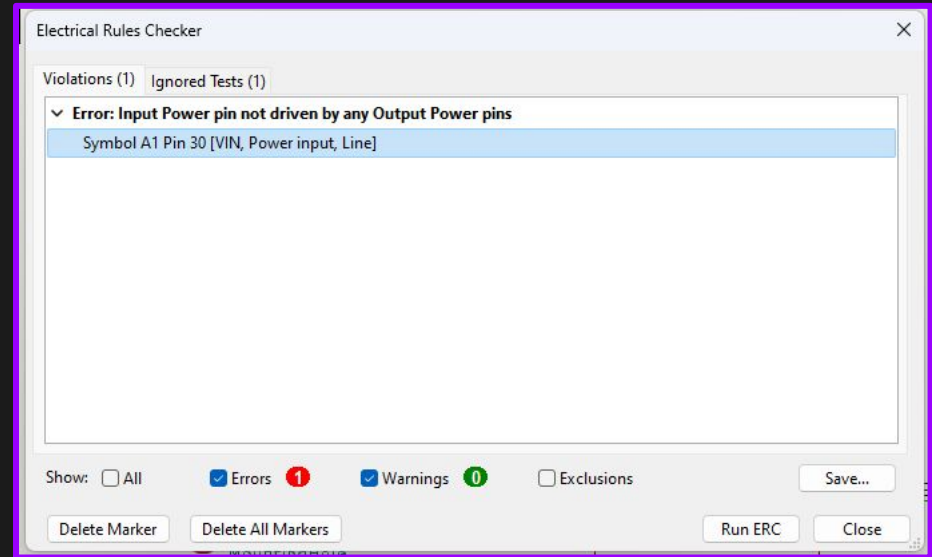
Footprint Assignment Tool

- Toolbar -> Run Footprint Assignment Tool
- Currently, symbols aren't mapped to any physical hardware
- A "Footprint" consists of the pads/holes a component is soldered to and the silkscreen boundaries on the PCB
- The footprint assignment tool maps pins from symbol to a physical footprint
- Some symbols may already have footprints linked to them


```
Symbol : Footprint Assignments
  1      A1  -        Arduino_Nano : Module:Arduino_Nano
  2      H1  -        MountingHole : MountingHole:MountingHole_3.2mm_M3
  3      H2  -        MountingHole : MountingHole:MountingHole_3.2mm_M3
  4      H3  -        MountingHole : MountingHole:MountingHole_3.2mm_M3
  5      H4  -        MountingHole : MountingHole:MountingHole_3.2mm_M3
  6      J1  - RGB Strip Terminal : Skills Series Footprints:LED Strip Terminal Block
  7      J2  -        Conn_01x03 : Connector_PinHeader_2.54mm:PinHeader_1x03_P2.54mm_Vert
  8      J3  -        USB_B_Micro : Skills Series Footprints:MicroUSB Port
  9      J4  -        Conn_01x06 : Connector_PinHeader_2.54mm:PinHeader_1x06_P2.54mm_Vert
 10      J5  -        Conn_01x05 : Connector_PinHeader_2.54mm:PinHeader_1x05_P2.54mm_Vert
 11      Q1  -            TIP120 : Package_TO_SOT_THT:TO-220-3_Vertical
 12      Q2  -            TIP120 : Package_TO_SOT_THT:TO-220-3_Vertical
 13      Q3  -            TIP120 : Package_TO_SOT_THT:TO-220-3_Vertical
 14      R1  -               470 : Resistor_THT:R_Axial_DIN0207_L6.3mm_D2.5mm_P7.62mm_Hor
 15      R2  -               470 : Resistor_THT:R_Axial_DIN0207_L6.3mm_D2.5mm_P7.62mm_Hor
 16      R3  -               470 : Resistor_THT:R_Axial_DIN0207_L6.3mm_D2.5mm_P7.62mm_Hor
 17      RV1 -               10K : Skills Series Footprints:10K Potentiometer Dial
 18      RV2 -               10K : Skills Series Footprints:10K Potentiometer Dial
 19      RV3 -               10K : Skills Series Footprints:10K Potentiometer Dial
 20      SW1 -            SW_SPST : Skills Series Footprints:Slide Switch
 21      SW3 -            SW_SPST : Skills Series Footprints:Slide Switch
```

College of Engineering and Computer Science

What is a Board Layout?

College of Engineering
and Computer Science
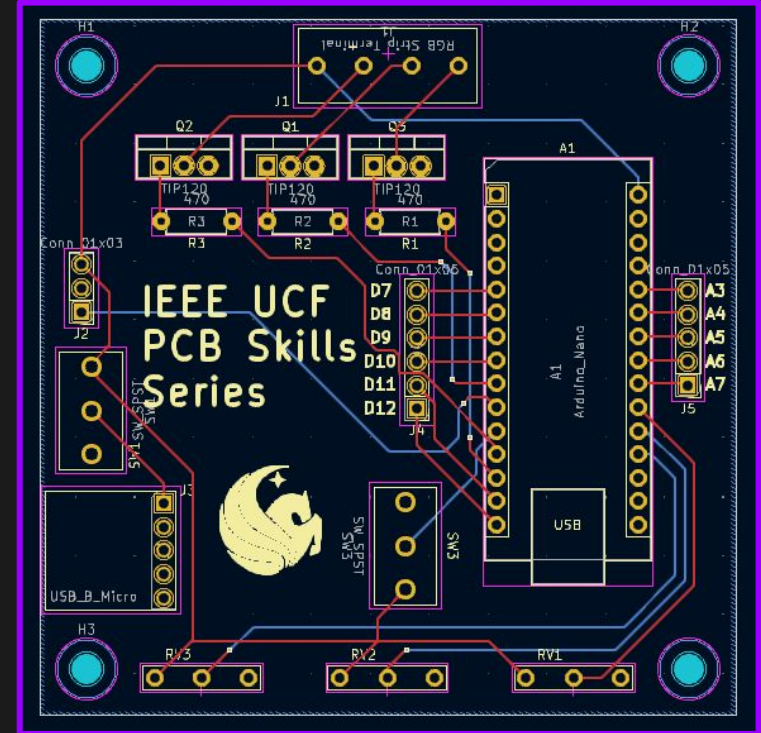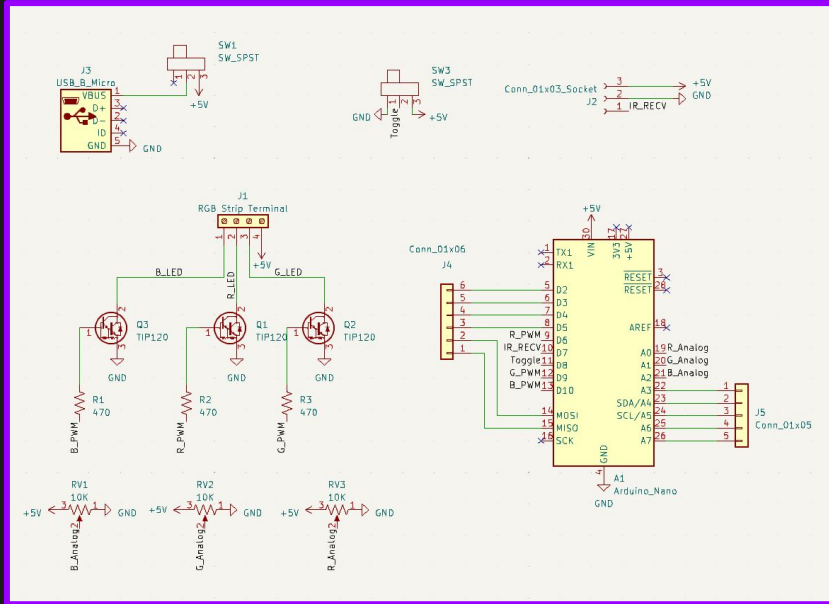
Useful Footprint Editor Keyboard Shortcuts

Footprint editor:

- A: place a footprint
- X: draw a trace
- S: redefine grid origin
- V: toggle between F.Cu and B.Cu
- Ctrl + shift + V: place a via
- Ctrl + shift + L: draw a line
- B: Fill all zones

- E: properties
- R: rotate symbol
- M: move

More shortcuts:
https://www.siue.edu/~gengel/ece3
26WebStuff/KiCAD-hotkeys.pdf

College of Engineering and Computer Science

Common Terminology

- **Layout Editor** - used to specify board boundaries and placement of footprints on a PCB
- **Trace** - copper track connecting two pads/holes
- **Via** - "Vertical interconnect access", hole through the PCB that connects two layers
- **Plane** - common connection surface on a layer
- **Footprint** - contains silkscreen boundary of a component and location of pads/holes for connections
- **Through-hole vs Surface-mount**
- **Silkscreen/Solder mask** - text and graphics printed on the PCB for user readability
- **Design Rule Check (DRC)** - ensures that PCB doesn't violate any electrical or mechanical rules set by PCB manufacturers

Board Layers

- **Front Copper/Bottom Copper**: routing layers that interconnect components
  - Components, traces, vias, planes, etc. will be on F.Cu and B.Cu
- **Solder mask**: thin protection layer applied over copper layers; gives PCB the green color
- **Silkscreen**: surface finish with text, footprint borders, and other graphics
  - Reference designators, values, and other text will be on silkscreen
- **Edge cuts**: defines board outline and other manufacturing information

F.Cu
B.Cu
F.Adhesive
B.Adhesive
F.Paste
B.Paste
F.Silkscreen
B.Silkscreen
F.Mask
B.Mask
User.Drawings
User.Comments
User.Eco1
User.Eco2
Edge.Cuts
Margin
F.Courtyard
B.Courtyard
F.Fab
B.Fab
User.1
User.2
User.3
User.4
User.5
User.6
User.7
User.8
User.9
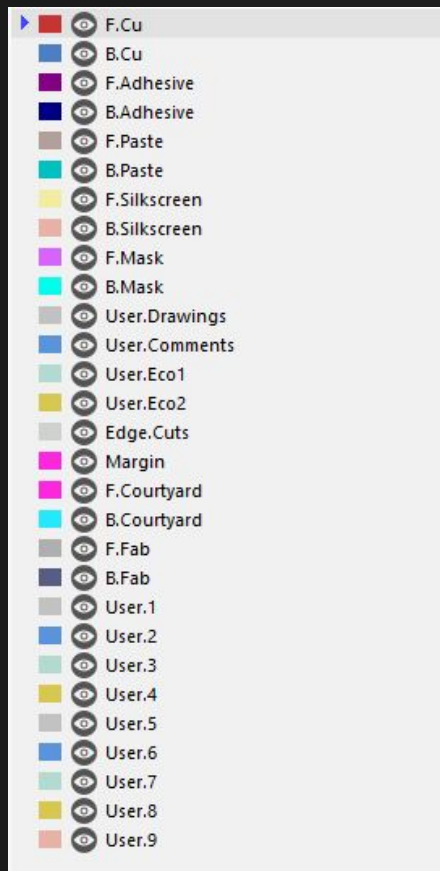
Nets vs Airwires vs Traces

- **Net**: connection between two pins on a schematic
- **Airwire**: Net translated to the PCB layout to show connection between pads
- **Trace**: physical copper connection between pads

Net

Trace

Airwire
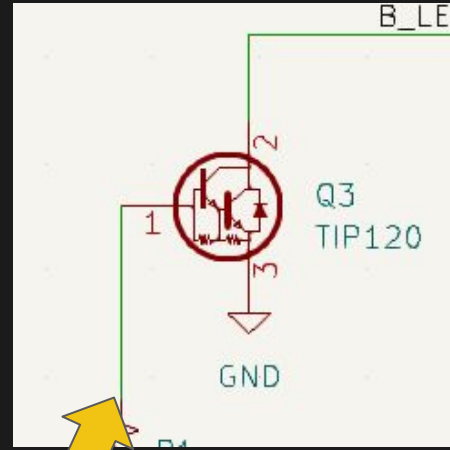
College of Engineering and Computer Science

Grid Units

- Pin spacing specified in non-metric units: 2.54mm, 5.08mm, etc
- Board outlines and component placement often specified in metric units
- 1 mil = 1/1000 of an inch
  - Ex: 2.54mm = 100 mil
- Change grid units with drop down or "N" hotkey

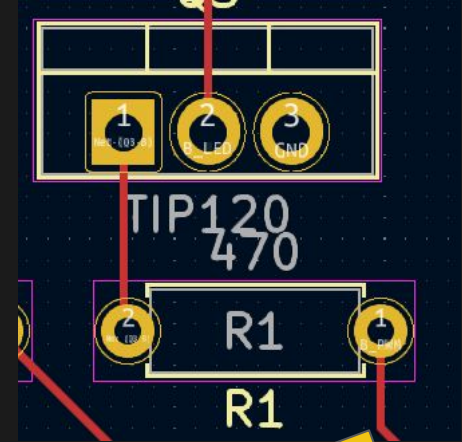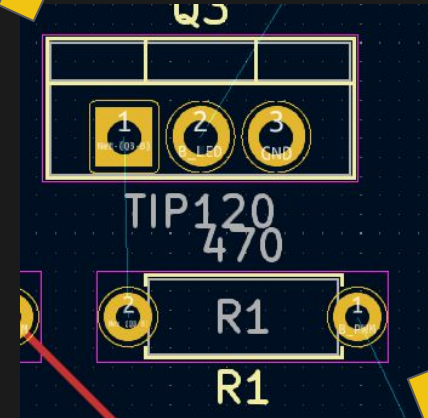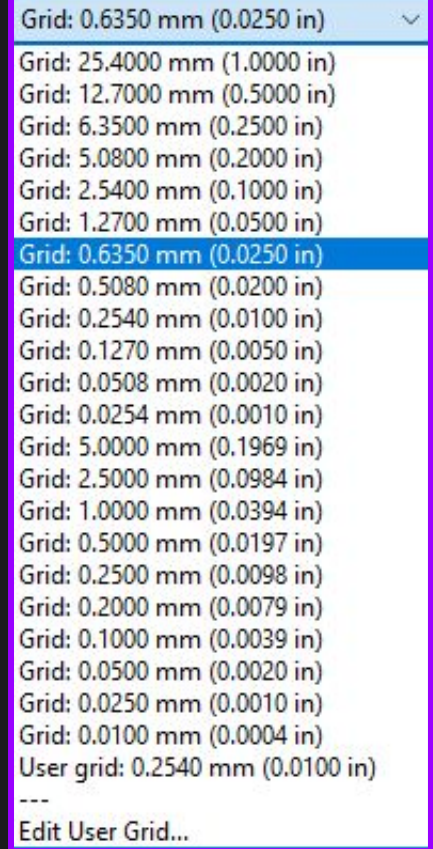| Grid: 0.6350 mm (0.0250 in) ▼ |
|---|
| Grid: 25.4000 mm (1.0000 in) |
| Grid: 12.7000 mm (0.5000 in) |
| Grid: 6.3500 mm (0.2500 in) |
| Grid: 5.0800 mm (0.2000 in) |
| Grid: 2.5400 mm (0.1000 in) |
| Grid: 1.2700 mm (0.0500 in) |
| Grid: 0.6350 mm (0.0250 in) |
| Grid: 0.5080 mm (0.0200 in) |
| Grid: 0.2540 mm (0.0100 in) |
| Grid: 0.1270 mm (0.0050 in) |
| Grid: 0.0508 mm (0.0020 in) |
| Grid: 0.0254 mm (0.0010 in) |
| Grid: 5.0000 mm (0.1969 in) |
| Grid: 2.5000 mm (0.0984 in) |
| Grid: 1.0000 mm (0.0394 in) |
| Grid: 0.5000 mm (0.0197 in) |
| Grid: 0.2500 mm (0.0098 in) |
| Grid: 0.2000 mm (0.0079 in) |
| Grid: 0.1000 mm (0.0039 in) |
| Grid: 0.0500 mm (0.0020 in) |
| Grid: 0.0250 mm (0.0010 in) |
| Grid: 0.0100 mm (0.0004 in) |
| User grid: 0.2540 mm (0.0100 in) |
| --- |
| Edit User Grid... |

Pads	Vias	Track Segments	Nets	Unrouted		
87	7	118	40	10		
F Z 1.36		X 105.2350 Y 134.1700		dx 105.2350 dy 134.1700 dist 170.5168	grid X 0.6350 Y 0.6350	mm

Getting Started: Define Board Dimensions

Draw Rectangle
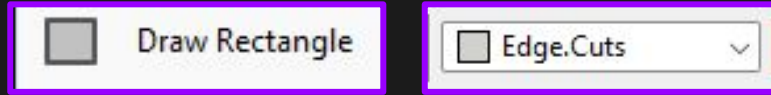
Edge.Cuts

- Dimensions used: 75 x 75 mm
- Draw a rectangle on "Edge cuts" layer
- Tip: change grid size (in bottom righthand corner) to 1mm
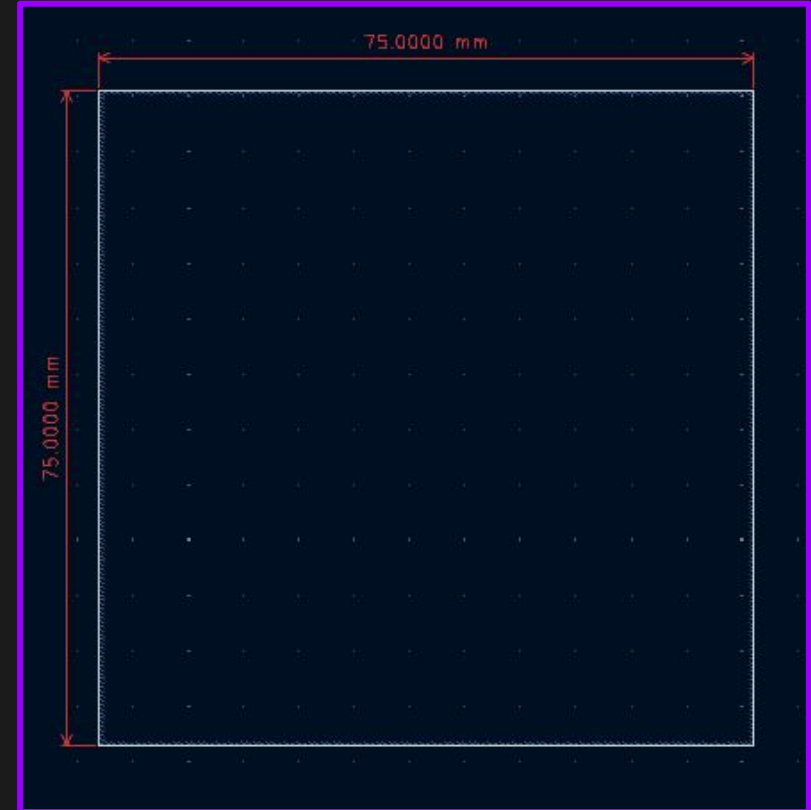
Getting Started: Update PCB from Schematic

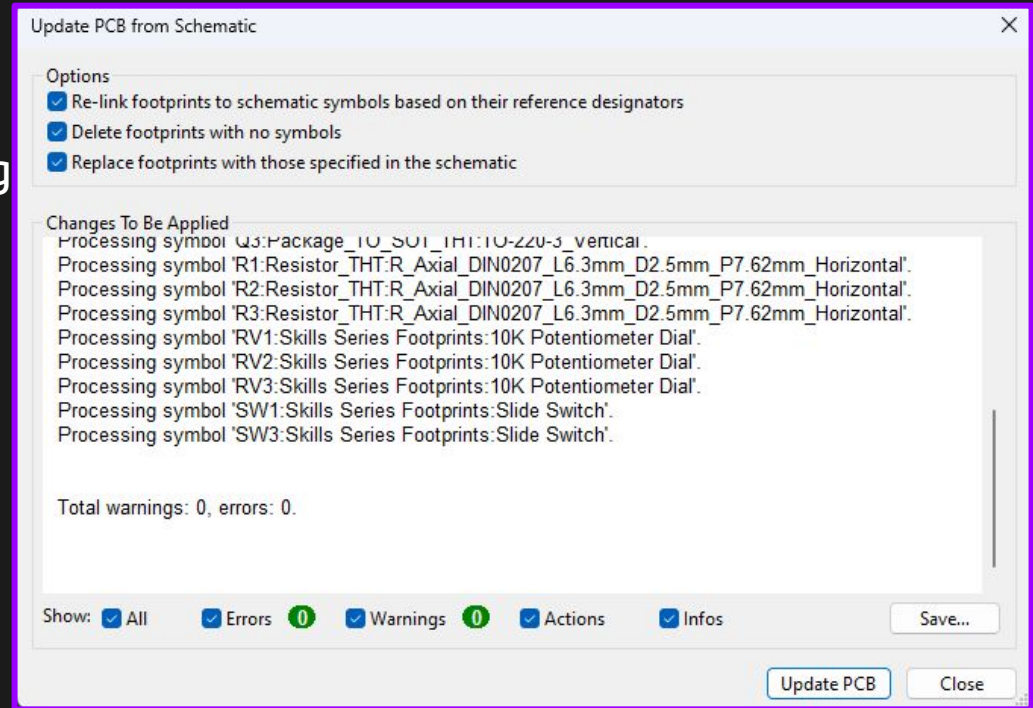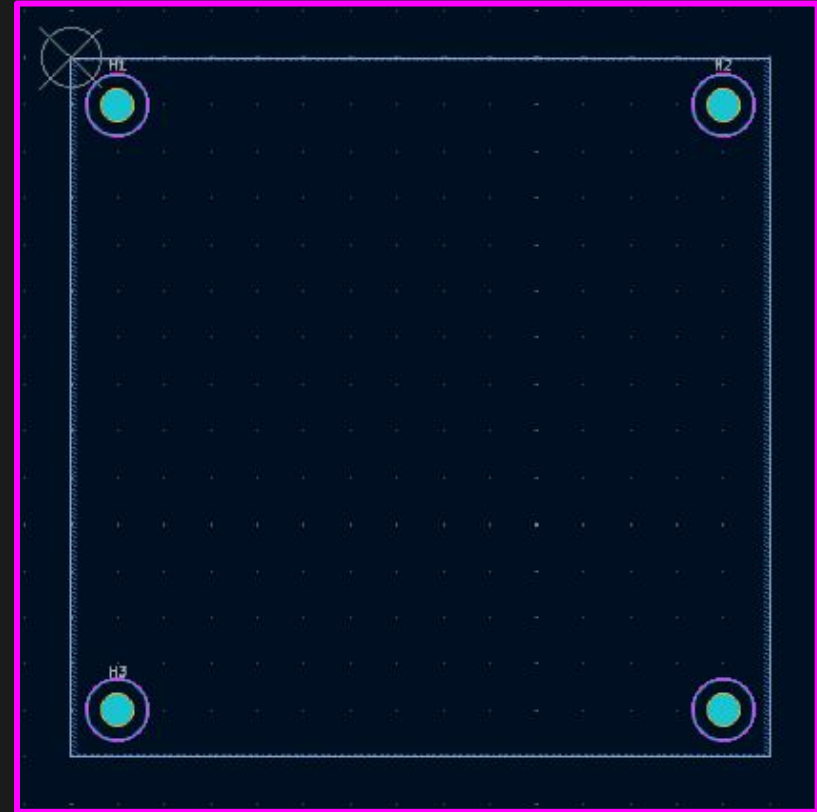- Used to import footprint assignments from schematic
- Must be re-run any time chang[es] are made to the schematic

Update PCB from Schematic   ✕

Options
☑ Re-link footprints to schematic symbols based on their reference designators
☑ Delete footprints with no symbols
☑ Replace footprints with those specified in the schematic

Changes To Be Applied
Processing symbol 'Q3:Package_TO_SOT_THT:TO-220-3_Vertical'.
Processing symbol 'R1:Resistor_THT:R_Axial_DIN0207_L6.3mm_D2.5mm_P7.62mm_Horizontal'.
Processing symbol 'R2:Resistor_THT:R_Axial_DIN0207_L6.3mm_D2.5mm_P7.62mm_Horizontal'.
Processing symbol 'R3:Resistor_THT:R_Axial_DIN0207_L6.3mm_D2.5mm_P7.62mm_Horizontal'.
Processing symbol 'RV1:Skills Series Footprints:10K Potentiometer Dial'.
Processing symbol 'RV2:Skills Series Footprints:10K Potentiometer Dial'.
Processing symbol 'RV3:Skills Series Footprints:10K Potentiometer Dial'.
Processing symbol 'SW1:Skills Series Footprints:Slide Switch'.
Processing symbol 'SW3:Skills Series Footprints:Slide Switch'.

Total warnings: 0, errors: 0.

Show: ☑ All   ☑ Errors ⓪   ☑ Warnings ⓪   ☑ Actions   ☑ Infos   [Save...]

[Update PCB] [Close]
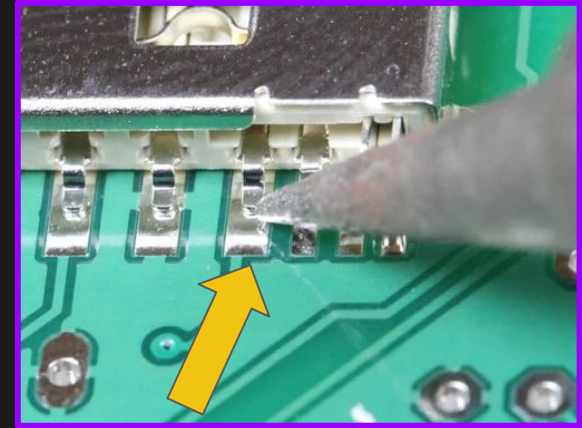
Placing Mounting Holes

- Place M3 mounting holes in each of the four corners of the board
- Inset the center of each hole 5mm from the corner of the board outline
- Easiest way: offset each hole from the corner
  - Set grid origin to the corner of the board ("S" key)
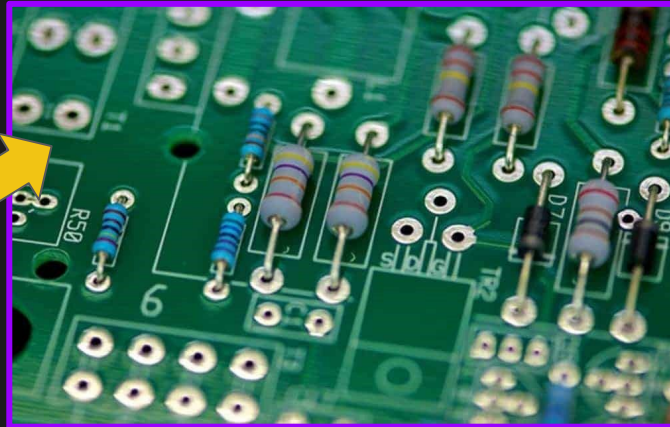 - Set grid units to 5mm

Component Placement

Design principles:

- Place I/O components in convenient access locations
- Place active logic (MCUs) and components with high connectivity in accessible areas
- Isolate high-power or high-speed components and signals (not relevant to this PCB design)
- Minimize the amount of airwire crossing/distance

Footprint type: Through-Hole vs Surface-Mount

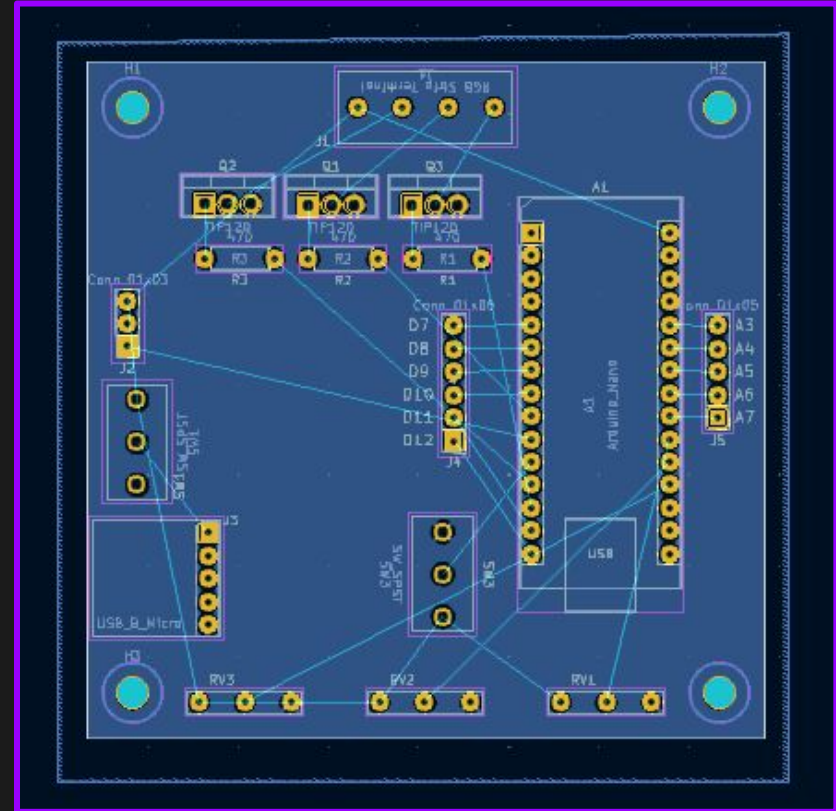
PTH/DIP   SMD
ANOD   CATHOD
+   −
VS

Through-hole

Surface mount

Ground Planes

- Continuous area of copper on a PCB layer
- Used to provide a return path (common ground) to the supply of a circuit
- Also helps to reduce noise and dissipate heat in high-power, high-speed circuits
- Greatly simplifies trace routing in a circuit
- Usually created on the bottom layer of a two-layer PCB

Creating a Ground Plane

1. Click "Add Filled Zone"
2. Select B.Cu and GND net
3. Draw an outline around board, extending beyond edge cuts
4. Right click and press "close outline"
5. Click "Fill All Zones" or press "B" to fill in ground plane
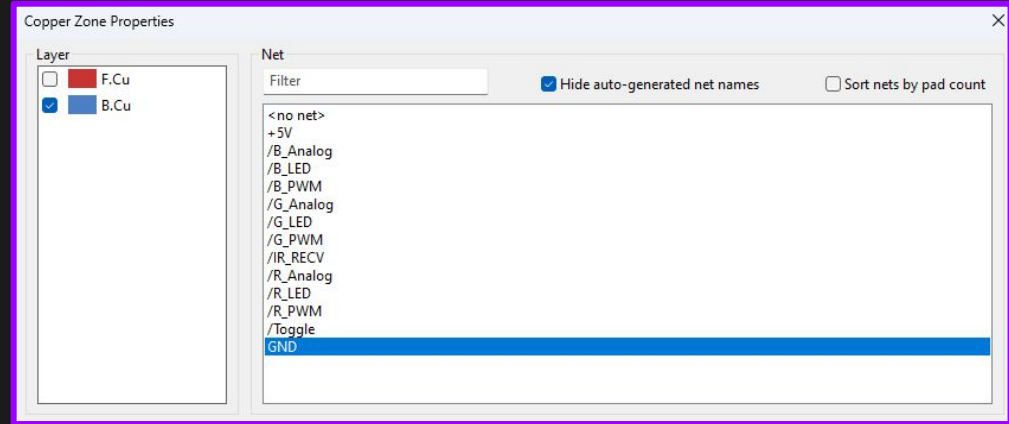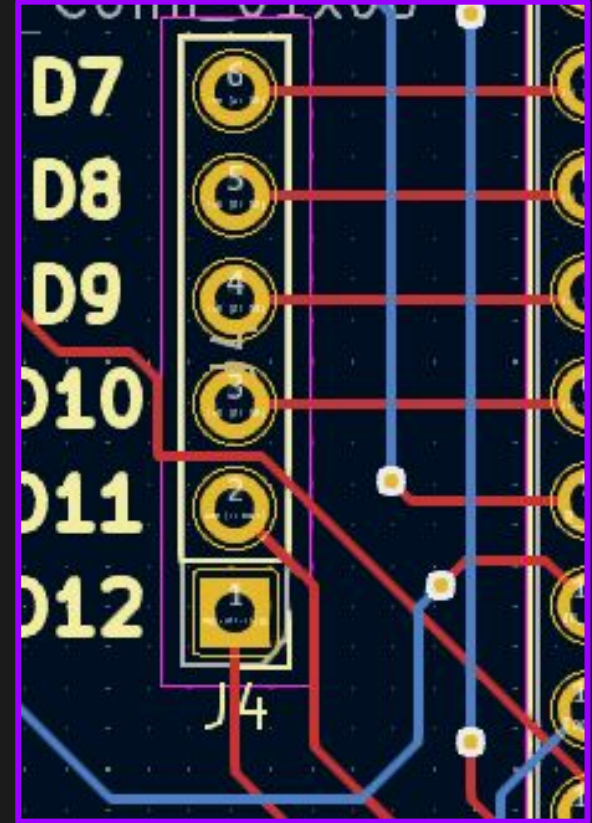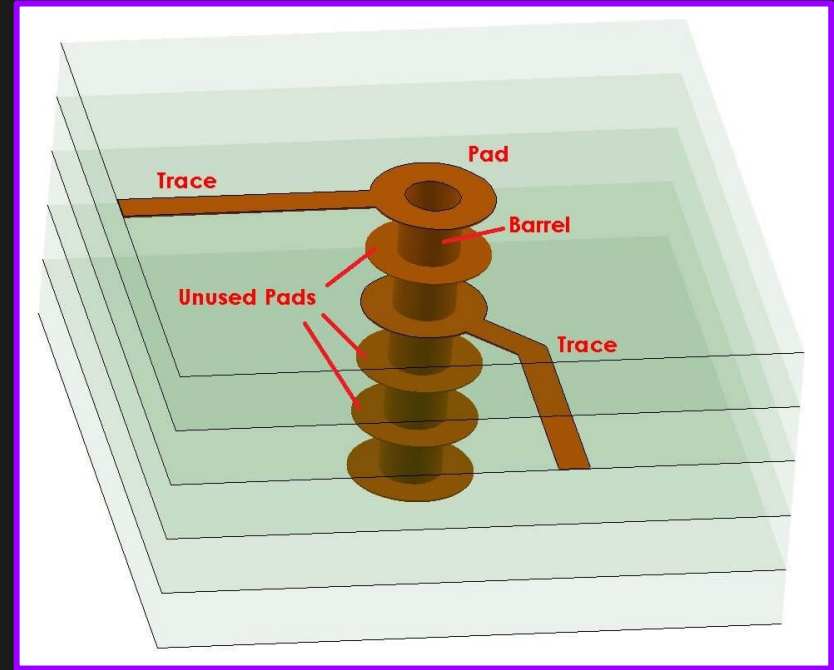
Trace Routing

- Most involved part of PCB design
- **Trace**: copper track that connects the pads of two components
- Two traces on the same layer cannot cross each other - short circuit!
- Goal: minimize the length of connections between components
  - Shorter trace = smaller signal propagation delay
- Other considerations: trace width and spacing
- Further reading

Vias

- Vertical Interconnect Access (VIA)
- Key idea: expand trace routing from 2->3 dimensions
- Solution to intersecting traces
- Copper tunnel between board layers
- Different types of vias: through hole, buried, blind, micro
- Further reading

Trace Routing: Live Demo

- Routing considerations:
  - No trace overlap
  - No sharp turns: drag corners to smooth turns
  - Minimize path length
  - When not possible to route on the top layer, add a via and switch to the bottom layer
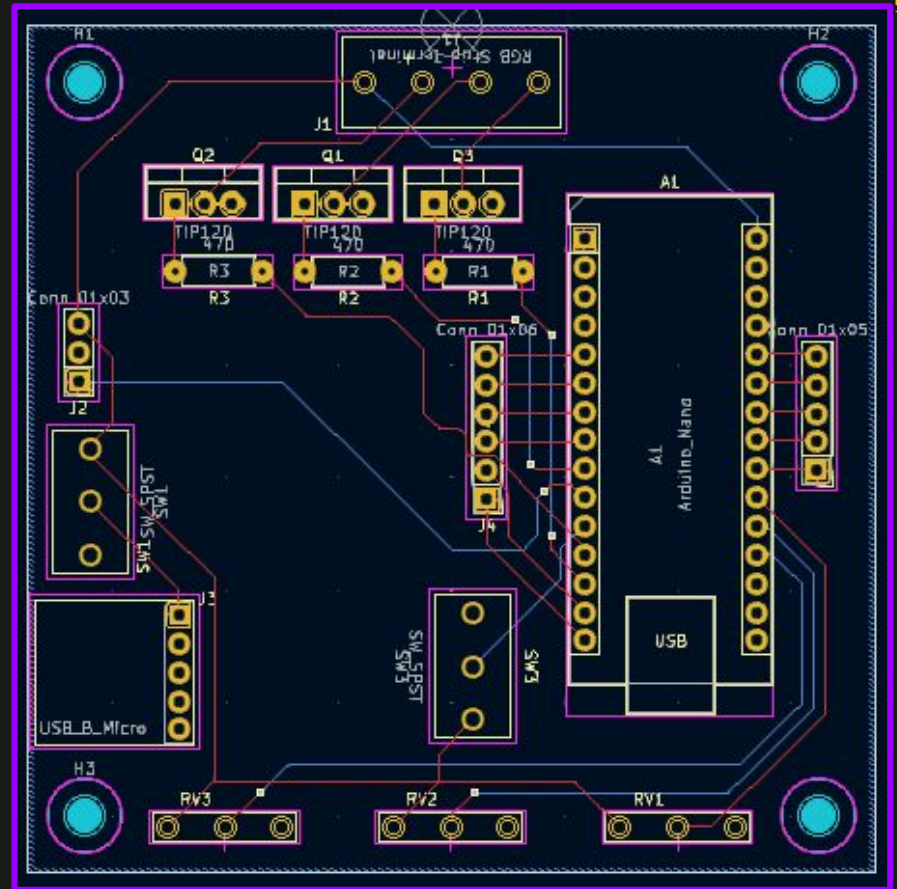
Adding Silkscreen Text

- **Silkscreen**: text/graphics layer of the PCB that conveys assembly information such as
  - Component placement location
  - Component reference designators
  - Component values
  - Company logos
  - Electrical properties/operating conditions
  - Component pinouts
- Main purpose is to enhance readability during assembly

# Design Rules Check

- Used to ensure that electrical and mechanical properties of the board do not violate any PCB manufacturers' standards
- Common errors/issues:
  - Unconnected trace: can usually be deleted
  - Silkscreen overlap: not a functional error, won't prevent manufacturing
  - Courtyard overlap: if two components are overlapping on the PCB

College of Engineering and Computer Science
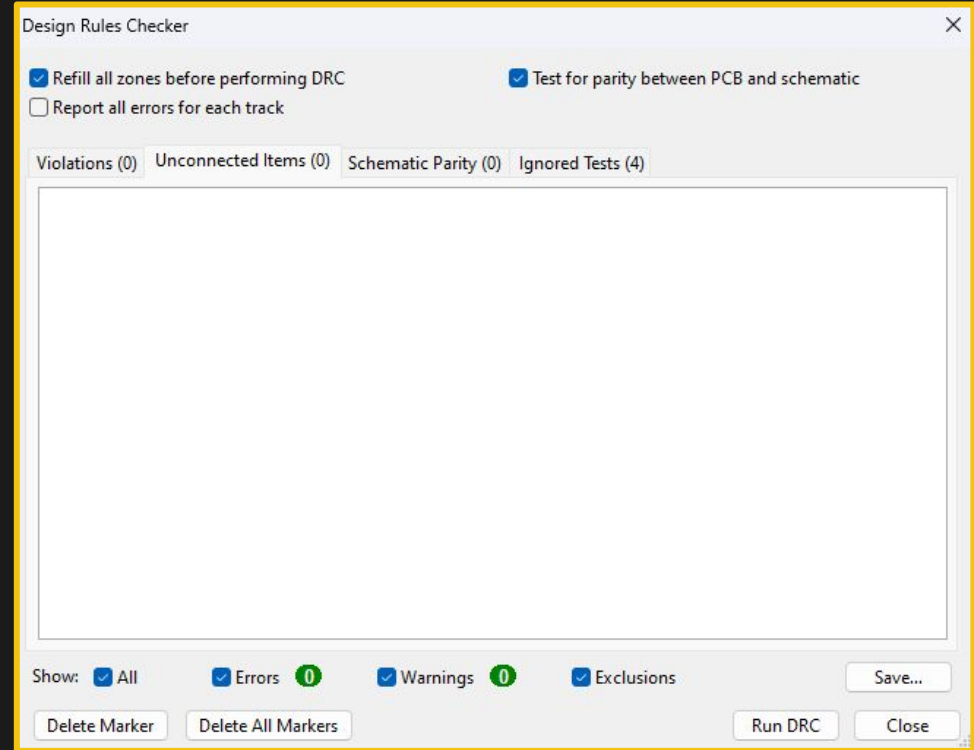
Gerber File Generation

- Gerber file: file format with all of drilling, etching, and silkscreen information from various layers
- Gerber file is sent to manufacturers such as JLCPCB, OSHPARK, and PCBWay
- Extra: can also auto-generate a bill of materials if footprints have linked part numbers
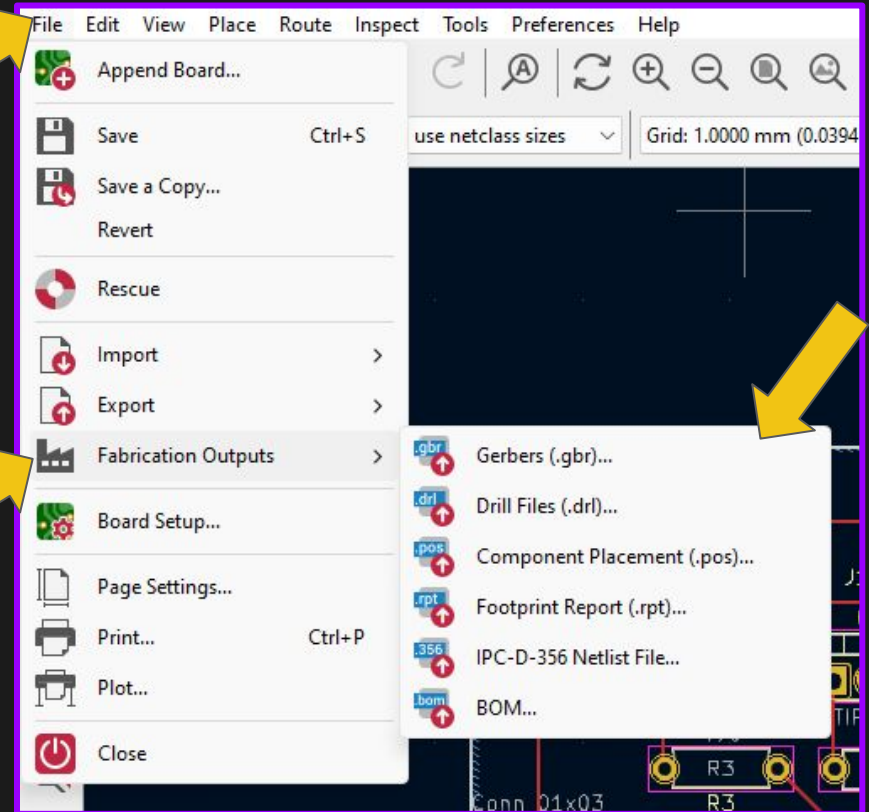
Plot

Plot format: Gerber   Output directory: C:\Users\Cory\Documents\GitHub\IEEE-KiCAD-Workshop\RGB LED Controller\

Include Layers
- ☑ F.Cu
- ☑ B.Cu
- ☐ F.Adhesive
- ☐ B.Adhesive
- ☑ F.Paste
- ☑ B.Paste
- ☑ F.Silkscreen
- ☑ B.Silkscreen
- ☑ F.Mask
- ☑ B.Mask
- ☐ User.Drawings
- ☐ User.Comments
- ☐ User.Eco1
- ☐ User.Eco2
- ☑ Edge.Cuts
- ☐ Margin
- ☐ F.Courtyard
- ☐ B.Courtyard

Plot on All Layers
- ☐ F.Cu
- ☐ B.Cu
- ☐ F.Adhesive
- ☐ B.Adhesive
- ☐ F.Paste
- ☐ B.Paste
- ☐ F.Silkscreen
- ☐ B.Silkscreen
- ☐ F.Mask
- ☐ B.Mask
- ☐ User.Drawings
- ☐ User.Comments
- ☐ User.Eco1
- ☐ User.Eco2
- ☐ Edge.Cuts
- ☐ Margin

General Options
- ☐ Plot drawing sheet
- ☑ Plot footprint values
- ☑ Plot reference designators
- ☐ Force plotting of invisible values / refs
- ☐ Mirrored plot
- ☐ Sketch pads on fabrication layers
- ☑ Check zone fills before plotting

Drill marks: None
Scaling: 1:1
Plot mode: Filled
- ☐ Use drill/place file origin
- ☐ Negative plot
- ☐ Do not tent vias

Gerber Options
- ☐ Use Protel filename extensions
- ☑ Generate Gerber job file
- ☑ Subtract soldermask from silkscreen

Coordinate format: 4.6, unit mm
- ☑ Use extended X2 format (recommended)
- ☑ Include netlist attributes
- ☐ Disable aperture macros (not recommended)

Output Messages
Plotted to 'C:\Users\Cory\Documents\GitHub\IEEE-KiCAD-Workshop\RGB LED Controller\RGB LED Controller-Edge_Cuts.gbr'.
Created Gerber job file 'C:\Users\Cory\Documents\GitHub\IEEE-KiCAD-Workshop\RGB LED Controller\RGB LED Controller-job.gbrjob'.
Done.

Show: ☐ All   ☑ Errors 0   ☑ Warnings 0   ☑ Actions   ☑ Infos   Save...

Run DRC...   Plot   Close   Generate Drill Files...

IEEE UCF

UCF College of Engineering and Computer Science