

# Introduction to Electronics Design Automation (EDA) Schematic Capture

Medical Electrical Equipment (BME590L)

Mark L. Palmeri, M.D., Ph.D.

January 18, 2023

# Outline

Overview

Schematics

Power

ERC

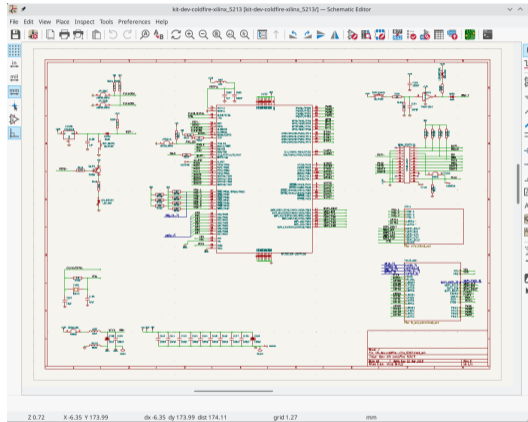
Best Practices

In-class / Lab Exercise



- ▶ Installation: <https://kicad.org>
  - ▶ Certain video cards will not work well with the PCB tool. In that case, disable the *Accelerated* toolset and choose the *Fallback* graphics toolset.
  - ▶ On Macs, your part / footprint / model libraries might not be detected automatically after installation. You might need to manually point Kicad at them in a path similar to: `Macintosh/Library/Application Support/kicad/templates/sym-lib-table/`.
- ▶ Documentation: [https://docs.kicad.org/6.0/en/getting\\_started\\_in\\_kicad/getting\\_started\\_in\\_kicad.html](https://docs.kicad.org/6.0/en/getting_started_in_kicad/getting_started_in_kicad.html)

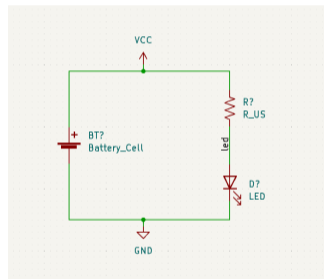
## UI/UX



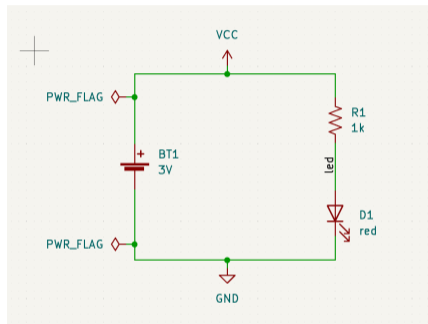
<https://docs.kicad.org/6.0/en/eeschema/eeschema.html#>

# Schematic Capture

1. Create a New Project, which creates a schematic file.
2. **DO NOT CHANGE THE GRID SPACING!!**
3. Setup Page Settings, using semantic versioning for "Revision".
4. Place component / part (Place Symbol) using default library. If component doesn't exist, then you either need to:
  - ▶ Download the part from an online database (e.g., SnapEDA)
  - ▶ Create part using the Library Editor.

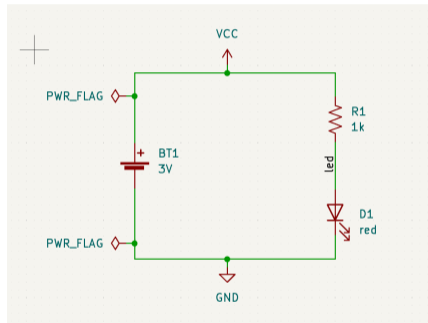


5. Annotate components (give each component a unique label)
6. Assign component values
7. Label nets with meaningful names
  - ▶ Nets are like nodes; common voltage connections.
  - ▶ Use Global net labels to avoid connection chaos.



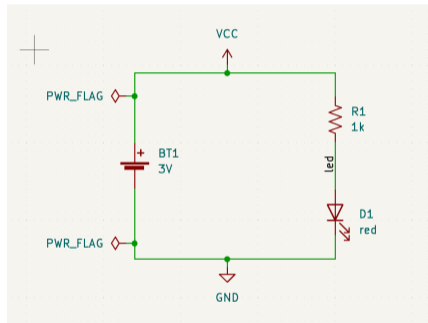
# Power Ports

- ▶ Power ports, including ground references, are also global net labels.
- ▶ Component pins can be explicitly designated power in/out (in contrast to signal).
- ▶ Power, ideally, cascades top-to-bottom (+ → GND (→ -)) on the schematic.



## Electrical Rules Check (ERC)

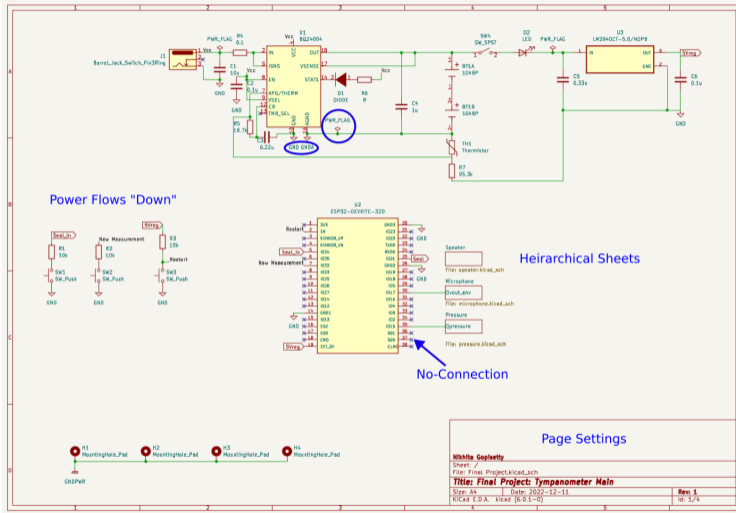
- ▶ Check the validity of the schematic
- ▶ Common error: “Input Power pin not driven by any Output Power pins”
  - ▶ KiCad checks to make sure that power can “drive” components that demand that input.
  - ▶ You can explicitly indicate this in the schematic using the `PWR_FLAG` symbol attached to the net in question.





## Best Practices

- ▶ Signal, ideally, flows left-to-right (input → output).
- ▶ Outline and label functional blocks. Use `Heirarchical Sheets` to organize more clearly-defined sub-circuits.
- ▶ Can include non-electrical items, like `Mounting Holes`.
- ▶ Add Test Pins/Pads to nets you will want to verify during testing.
  - ▶ Power nets
  - ▶ Signal I/O
- ▶ Use `No-Connection` flags for pins that are intentionally not connected to other components.
- ▶ You can create a Bill of Materials (BOM) based on your schematic.



# Create a variable voltage divider driving an LED

- ▶ File → Page Settings...
- ▶ 2 AA battery
- ▶ Use power ports for battery voltage and GND.
- ▶ 10k resistor and 20k potentiometer
- ▶ Annotate components
- ▶ Test pin on each net
- ▶ Label  $V_{out}$  net.
- ▶ Create a Hierarchical Sheet for an LED output.<sup>1</sup>

---

<sup>1</sup>This is complete overkill...

# In lab this week...

More schematic generation fun!