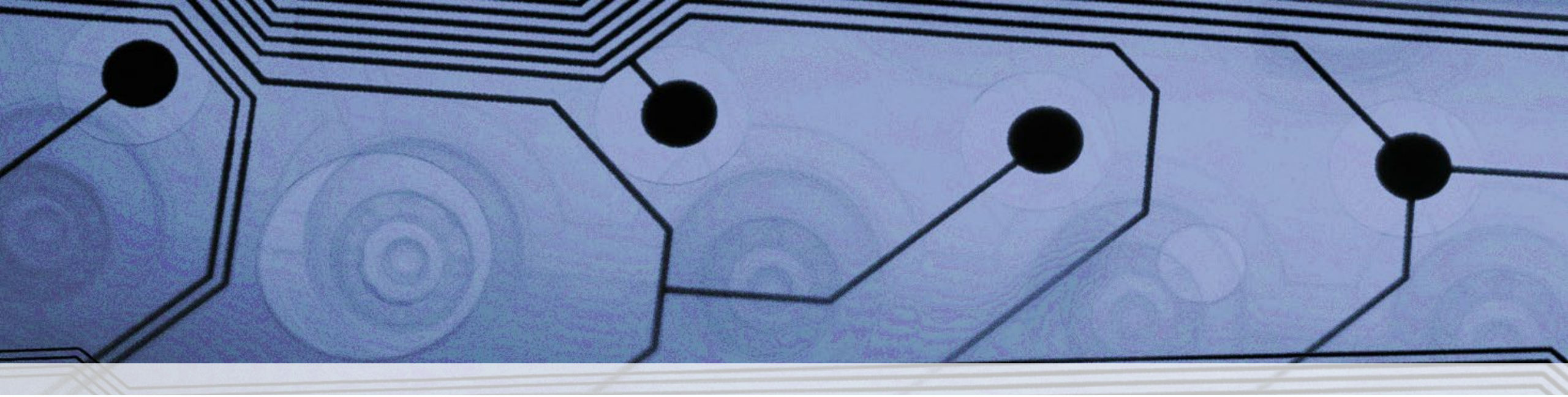


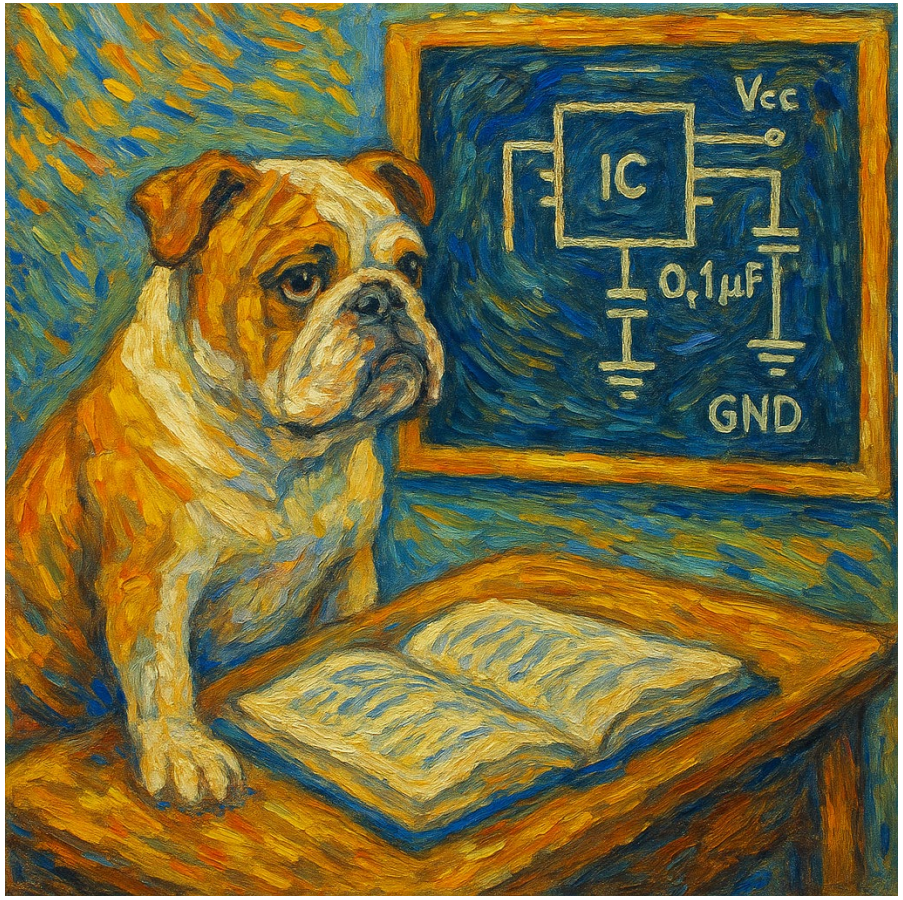
Hands-on PCB Design and Electronic Prototyping with Autodesk Fusion

SUMMER ENRICHMENT- LYLES COLLEGE OF ENGINEERING

REVISION 1.0

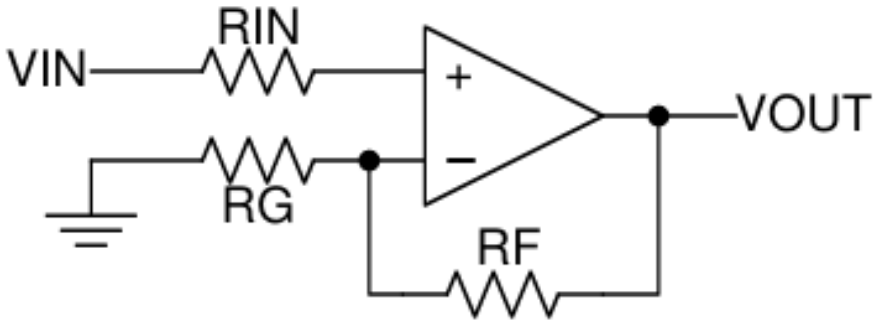
Day 2 Review

- Schematic Capture Essentials
  - Creating and saving a new project and schematic
  - Component libraries and management
  - Placing and wiring components
    - Netlists
- Hands-on Activity
  - Creating a basic schematic circuit- LM358 Non-Inverting Amplifier
  - Include decoupling capacitor and 4 pin header



Create a Schematic of a Circuit

- Op Amp Non-Inverting Amplifier
- Basic Schematic Included on this Slide
- Use a LM358 Op Amp
- Add a 4-pin header
- Add a +5V supply
- Add GND symbols to ground connections
- Add a ceramic decoupling capacitor between +5V and GND
- Configure the unused amplifier for non-inverting unity gain and add a 4.7k Ω resistor to GND



The Fusion Schematic Environment

- Schematic Navigation
- Library Management

Day 3: Advanced Schematic Techniques

- Complex Schematics
  - Creating custom symbols and parts
  - Hierarchical and modular schematic designs
- Schematic Validation
  - Error checking and electrical rule checking (ERC)
- Hands-on Activity
  - Creating a schematic symbol for the connector to be used in a Raspberry Pi Expansion Card

AI Minute

ChatGPT

Grok

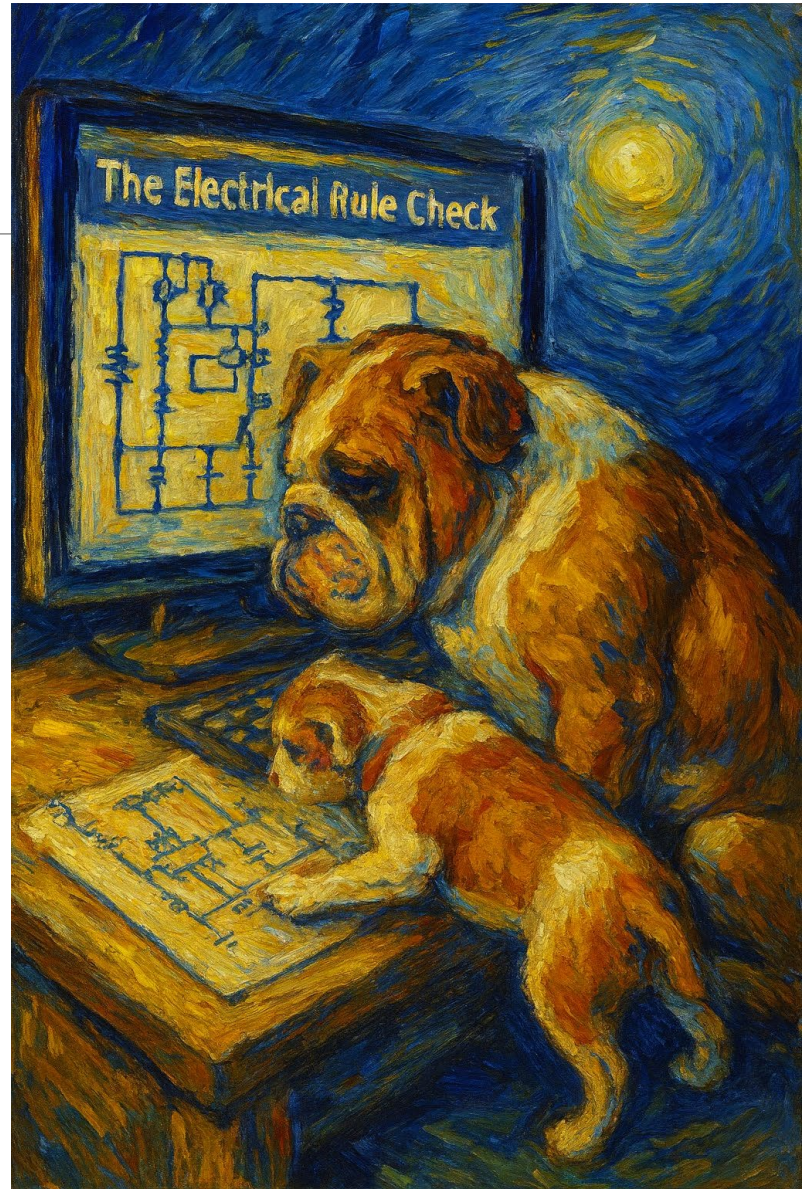
- Please type the following prompts into the AI Tool of your choice, Chat GPT, Gemini, Grok, Copilot, etc…
  - **What are hierarchical and modular schematic designs? How are they different?**

Schematic ERC Check

- **The Electrical Rule Check (ERC)** in AutoDesk Fusion Electronics helps detect logical and electrical errors in your schematic design.
- Common types of errors it can detect:
  - Electrical Connectivity Issues
    - Unconnected Pins – Pins that should be connected (e.g., power or ground) but are left floating.
    - Unconnected Nets – Wires that don't go anywhere or end abruptly.
    - Multiple Net Names on the Same Wire – Two or more different net labels assigned to the same wire.
    - Short Circuits – Conflicting connections, like power and ground tied together.

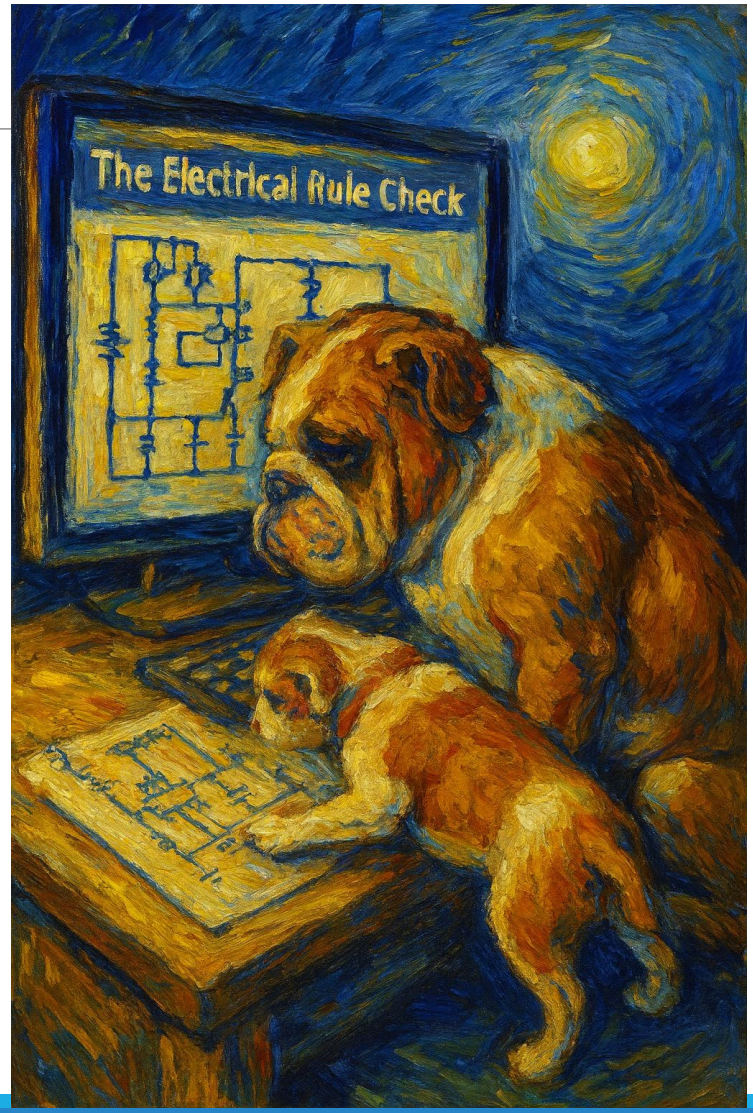
Schematic ERC Check

- Pin Usage Conflicts
  - Power Pin Not Connected to Power – VCC or GND pins not actually connected to appropriate nets.
  - Output Connected to Output – Two output pins directly connected, which could cause contention.
  - Input Not Driven – Input pins with no signal source (e.g., not connected to an output or net).
  - Conflict Between Passive and Active Pins – Violations of pin direction compatibility.
- Logic & Netlist Problems
  - Duplicate or Missing Net Names – Nets that are misnamed, creating logic errors.
  - Multiple Global Labels on One Net – e.g., two different labels like VCC and 5V on the same wire.



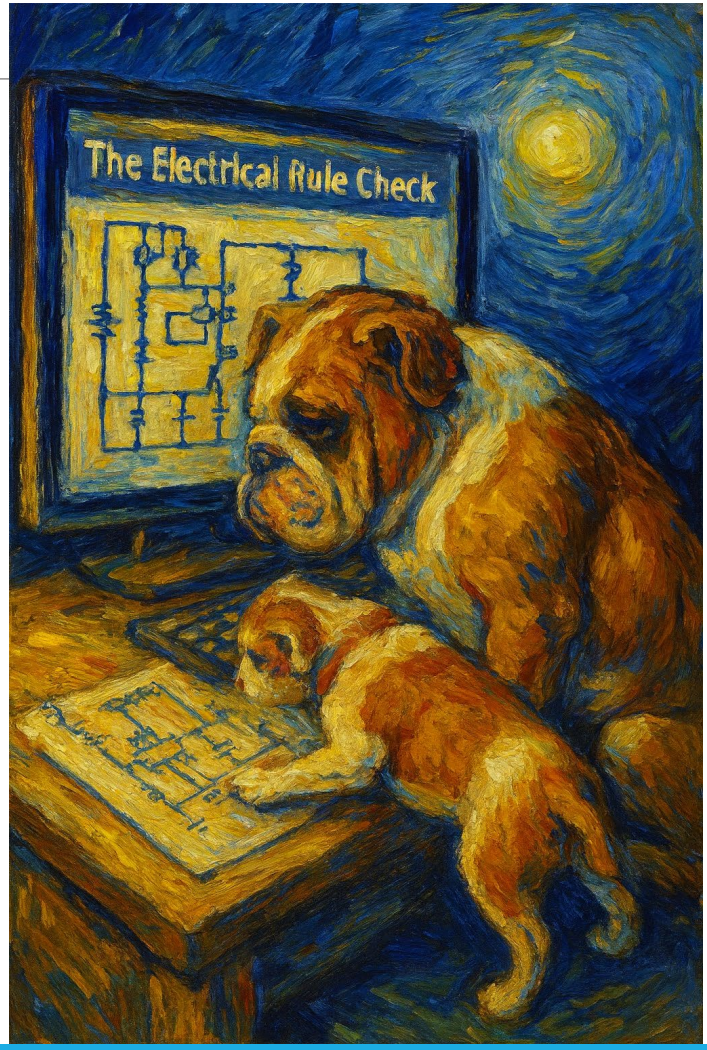
Schematic ERC Check

- Component Definition Errors
  - Missing Part Values or Names – Components without identifiers or missing values (e.g., a resistor without a resistance).
  - Improper Pin Types in Symbols – Symbols defined with incorrect or incomplete pin types.
- Miscellaneous Wrong or Conflicting Attributes
  - Conflicts in user-defined attributes (like voltage ranges or footprint mismatches).
  - Duplicate Part Designators – Two parts with the same name, e.g., both named R1.



Schematic ERC Check Summary

- Summary:
  - ERC in Fusion Electronics focuses on logical and electrical integrity, not physical layout.
  - It ensures your schematic is electrically and logically valid before proceeding to the PCB layout phase.



AI Minute

ChatGPT

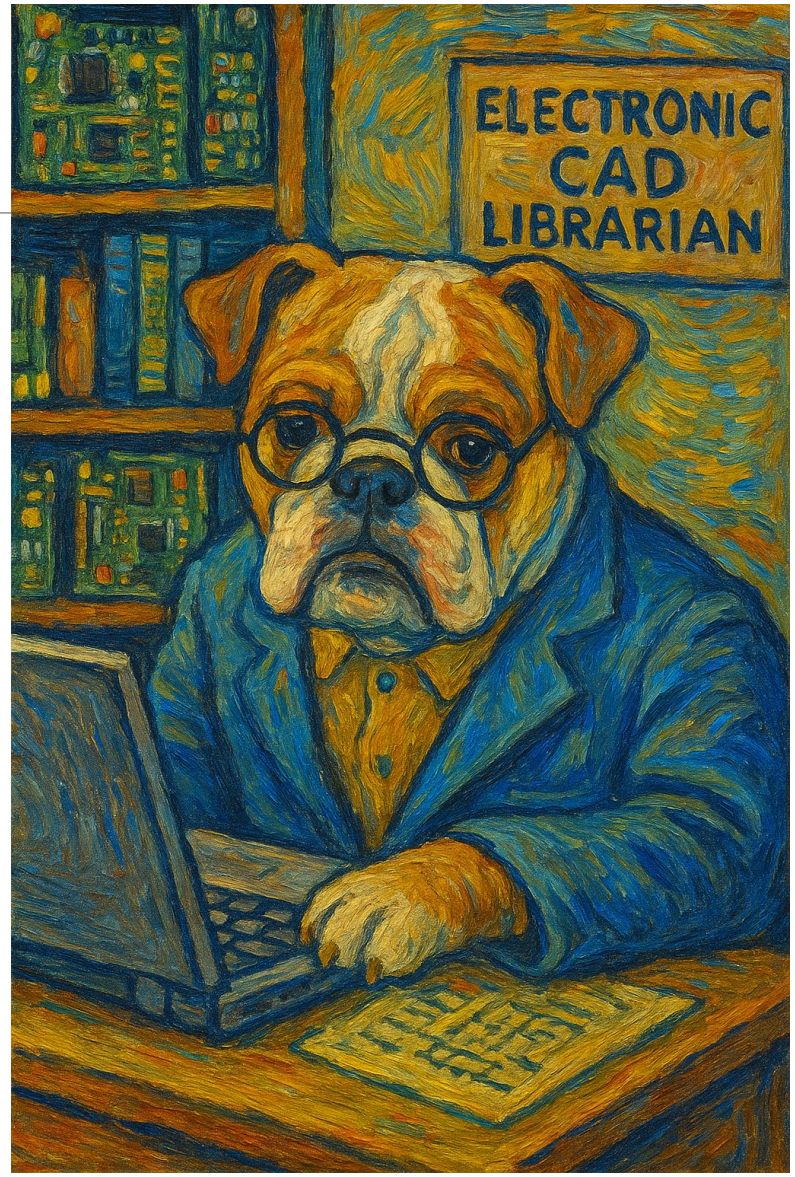
Grok

- Please type the following prompts into the AI Tool of your choice, Chat GPT, Gemini, Grok, Copilot, etc…

  - **What is the difference between Hub Libraries, Public Libraries, and Private Libraries in AutoDesk Fusion Electrical?**
  - **Considering an AutoDesk Fusion Electrical Library, describe in detail what a schematic symbol, a PCB footprint, and a Component are.**
  - **In AutoDesk Fusion, what is an ERC check and what types of errors does an ERC check identify ?**

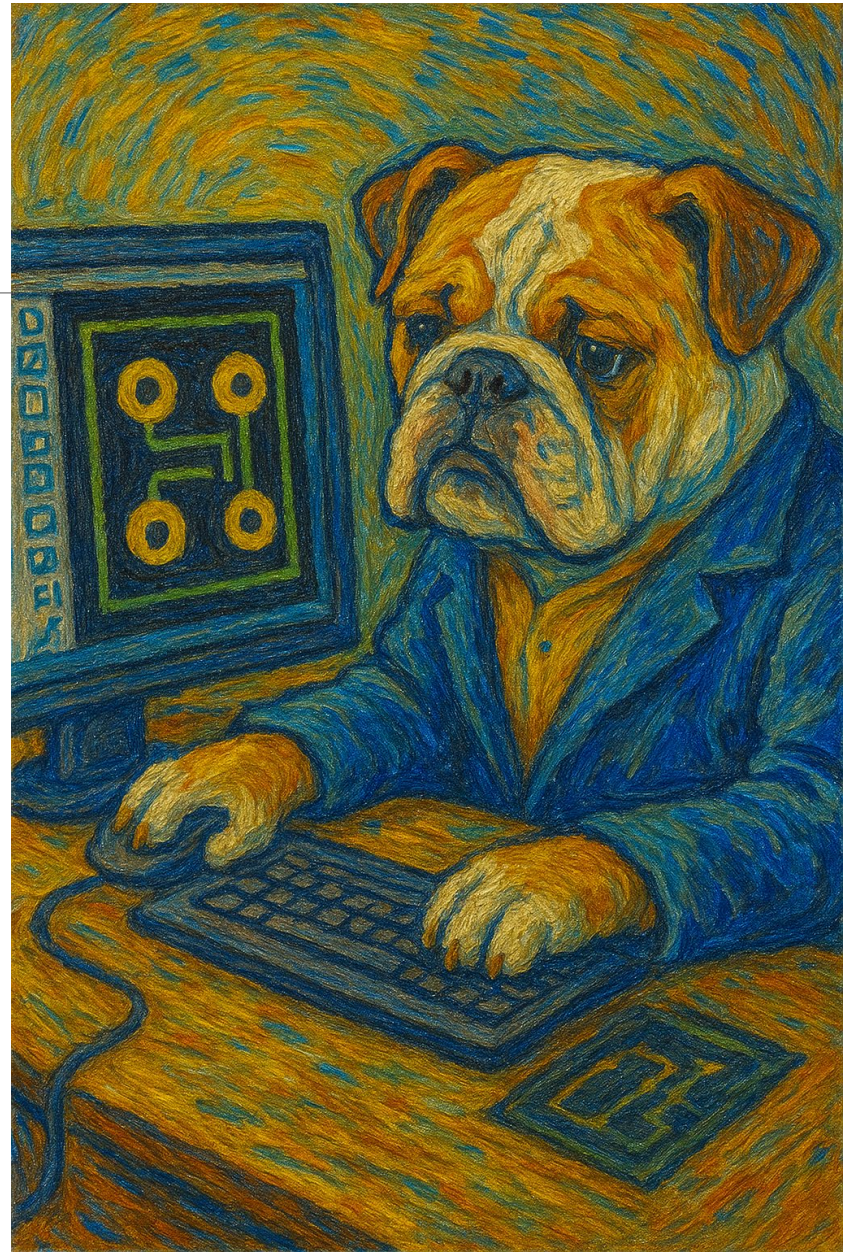
Electronic CAD Libraries

- CAD Library Management
  - Smaller Companies
    - Staff engineers are tasked with maintaining component libraries
  - Larger Companies
    - Typically have CAD Librarians
      - Maintain Libraries
      - Create New Components



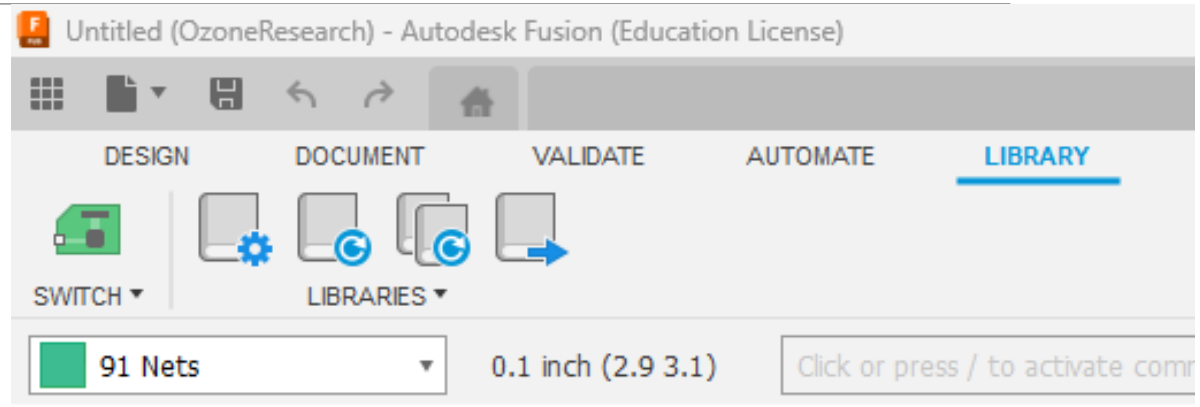
Fusion CAD Libraries

- Managed Libraries
  - Maintained by AutoDesk or Partners
    - Staff engineers are tasked with maintaining component libraries
- User Libraries
  - Users can create their own libraries
    - Maintained by the user
    - New components can be added by the user
    - Useful for custom or unsupported components



The Library Menu

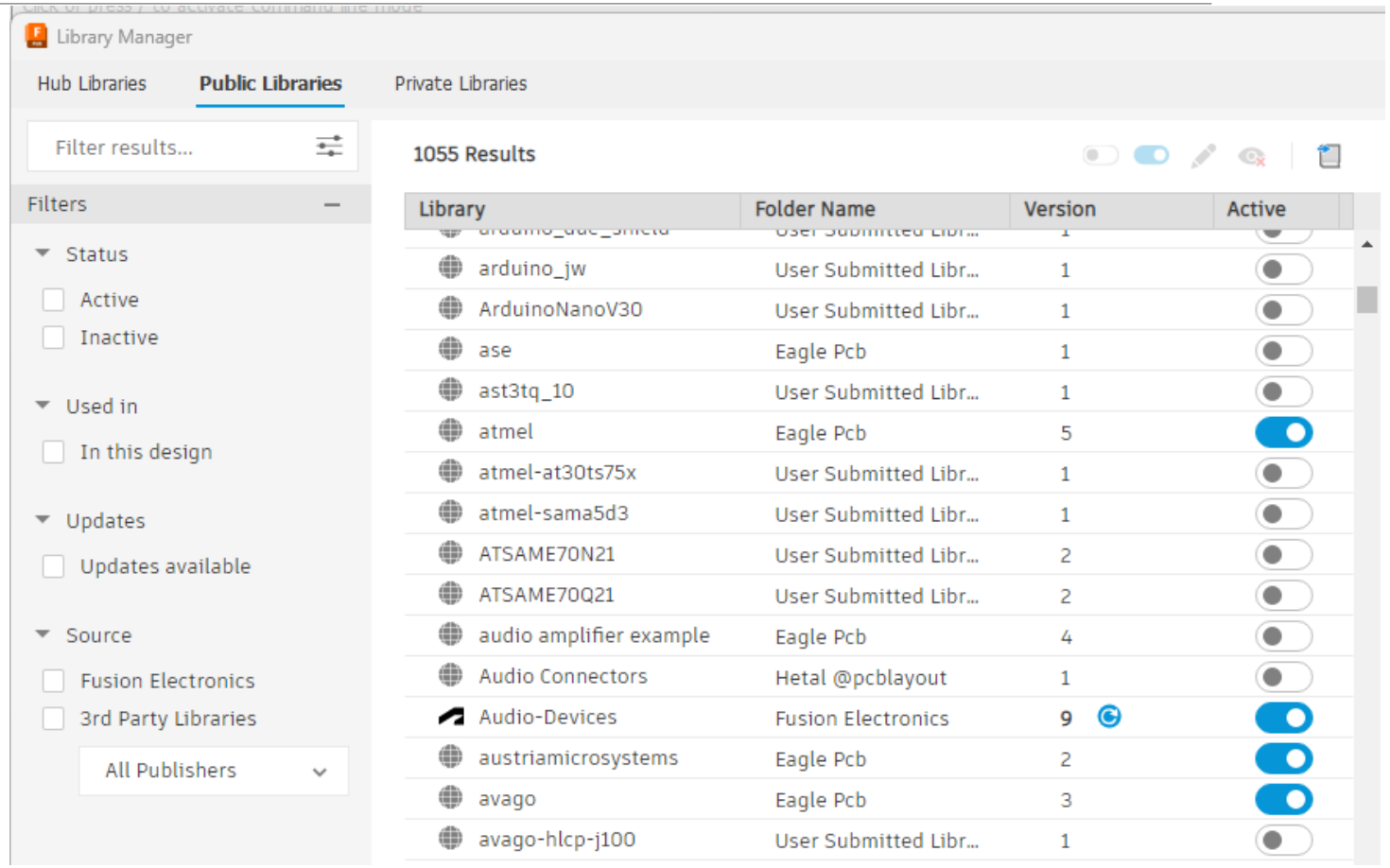
- From the Schematic tool
  - Click on the “Library” tab
    - Four Icons
      - Open Library Manager
      - Update Design from Library
      - Update Design from all Libraries Used In Design
      - Export Libraries



The Library Manager

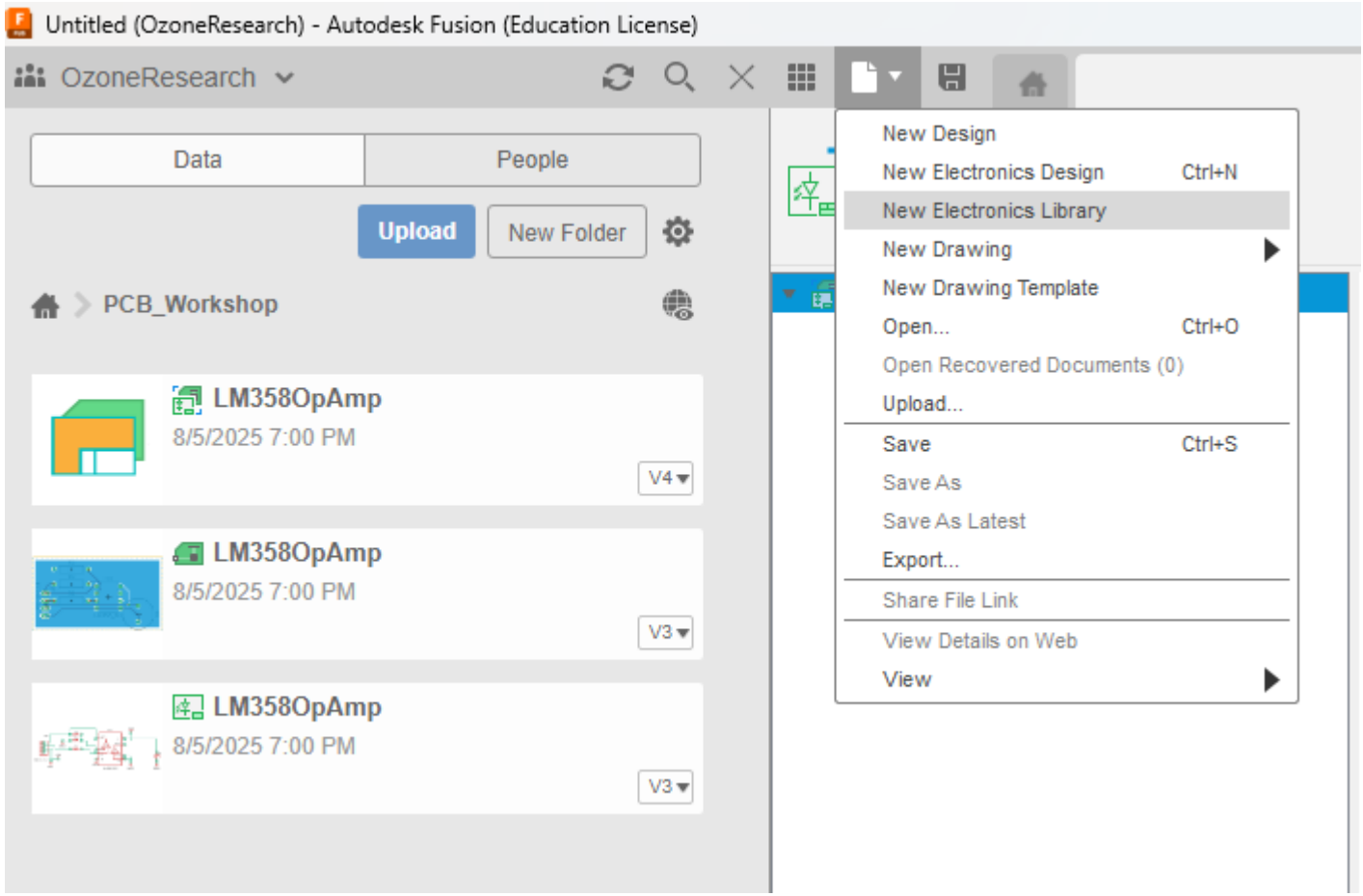
- Library Manager Functions Include
  - Edit Libraries
  - Select Libraries

- AI Minute:
  - Prompt
  - **What are the differences between Hub Libraries, Public Libraries, and Private Libraries in AutoDesk Fusion Electrical?**



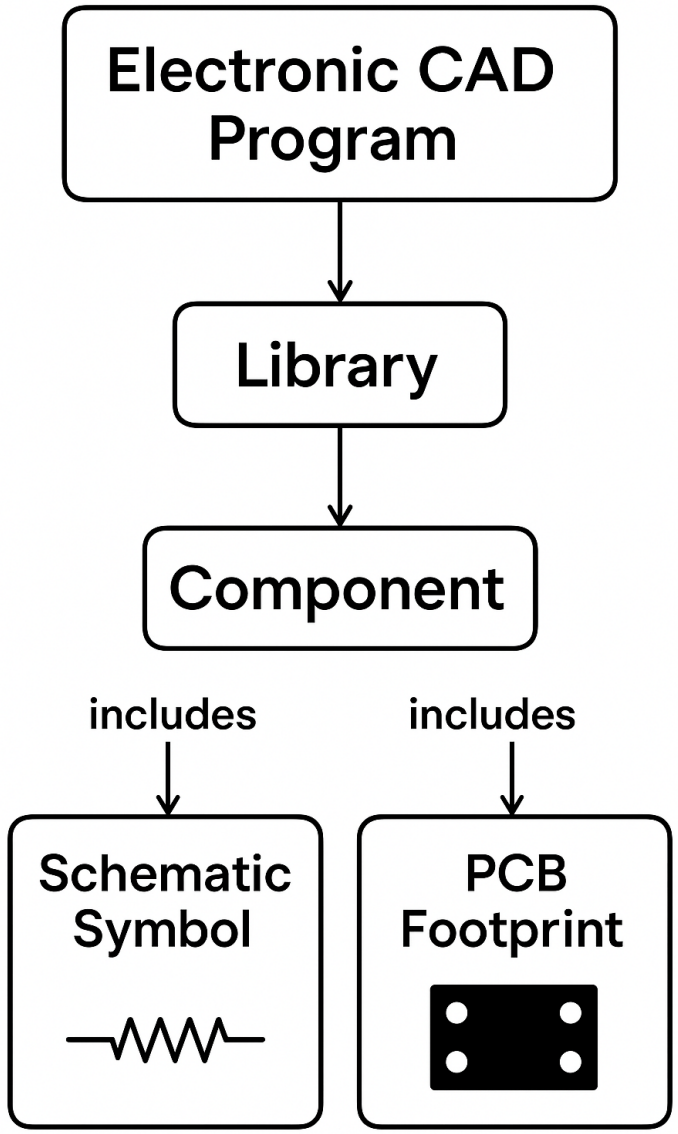
Create a Library

- Workspaces
  - Make sure you have created a workspace
    - If not, from the home location in the Data Panel
      - Click on the “New Project” Button”
      - Name the workspace
        - Suggested name PCB_Workshop
- Create a new Electronics Library
  - From the file icon,select “New Electronics Library”
  - Click the “Save” icon and give the library a name



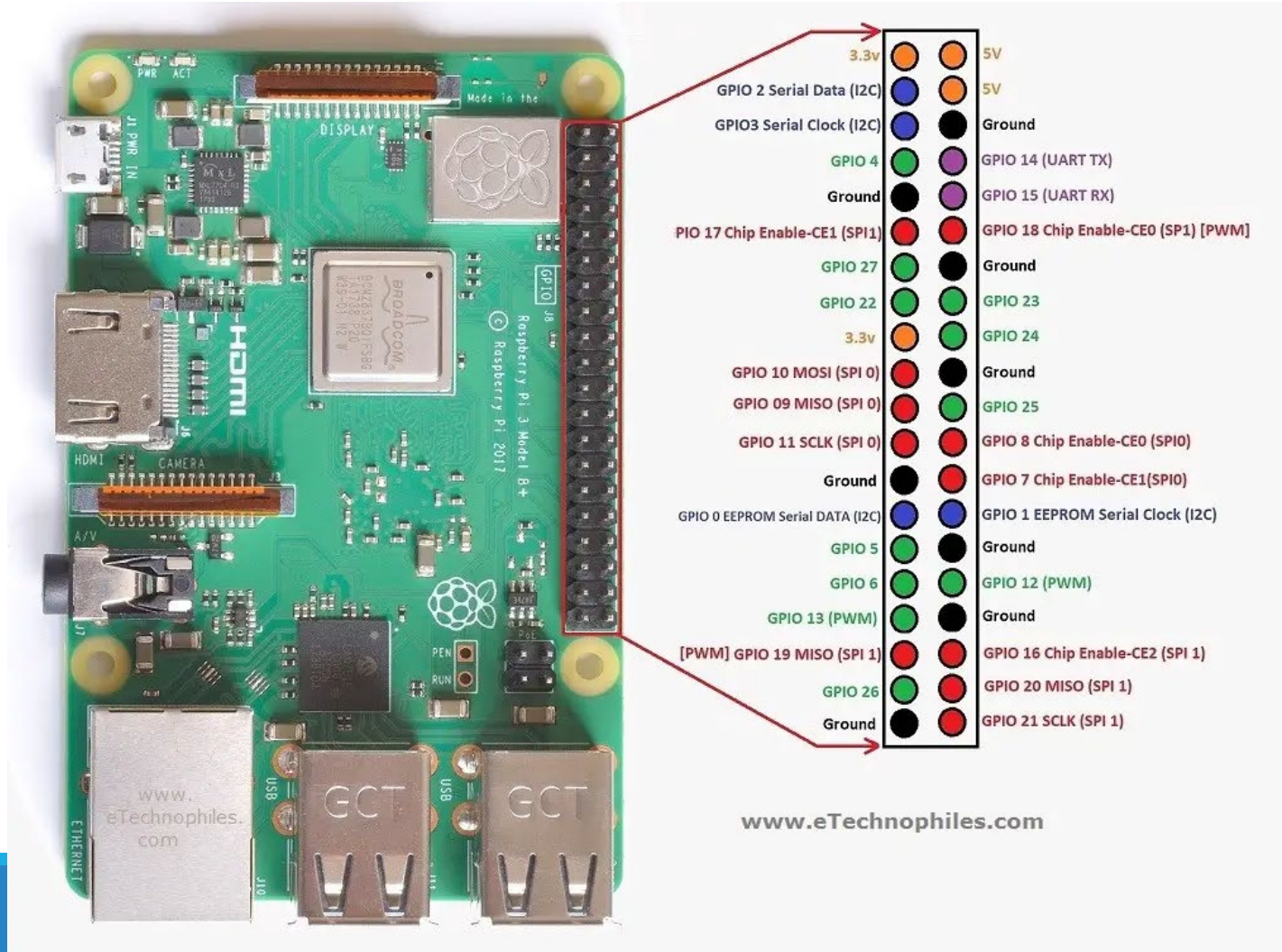
CAD Library Elements

- Libraries
  - Libraries contain components
- Components
  - Components include:
    - A Schematic Symbol
    - A PCB Footprint
    - A 3D Model



Raspberry Pi Pinout- Create Library Component

- Component Creation
  - Create Schematic Symbol
  - Create PCB Footprint
  - Combine Footprint and Symbol into a Component



Questions?