

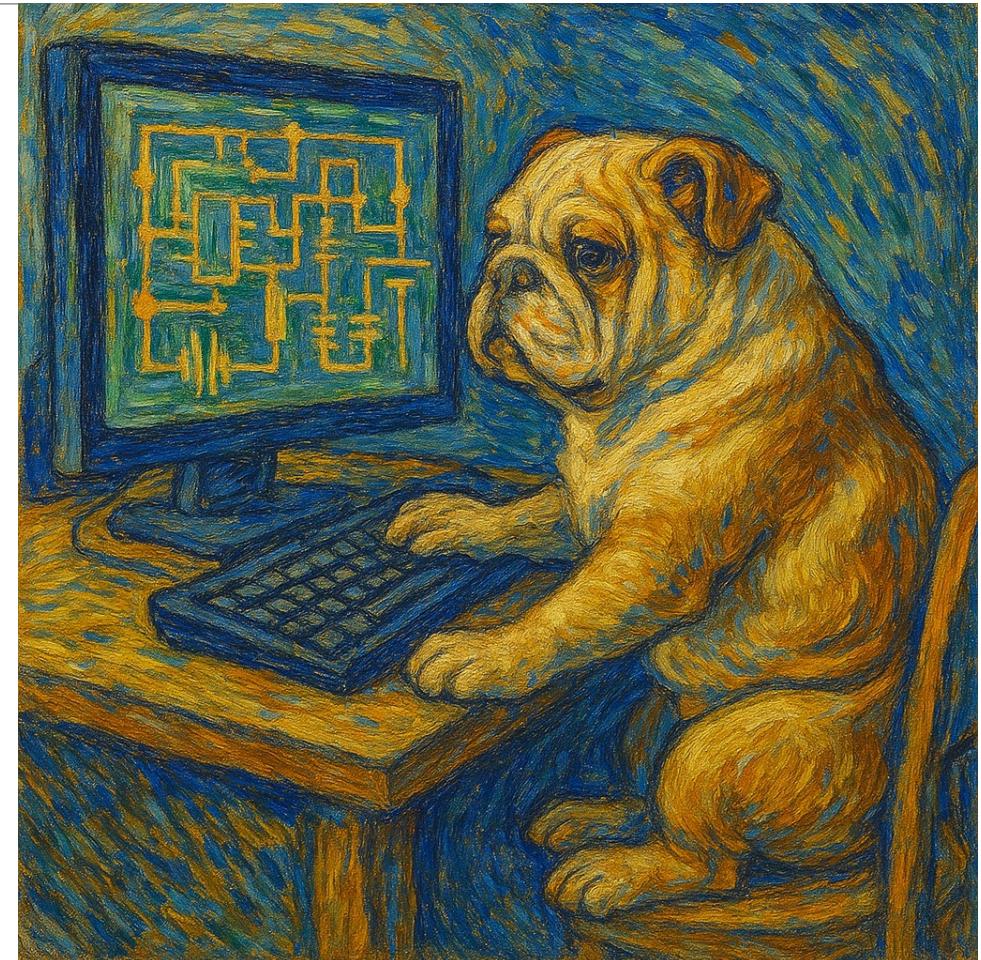
Hands-on PCB Design and Electronic Prototyping with Autodesk Fusion

SUMMER ENRICHMENT- LYLES COLLEGE OF ENGINEERING

REVISION 1.0

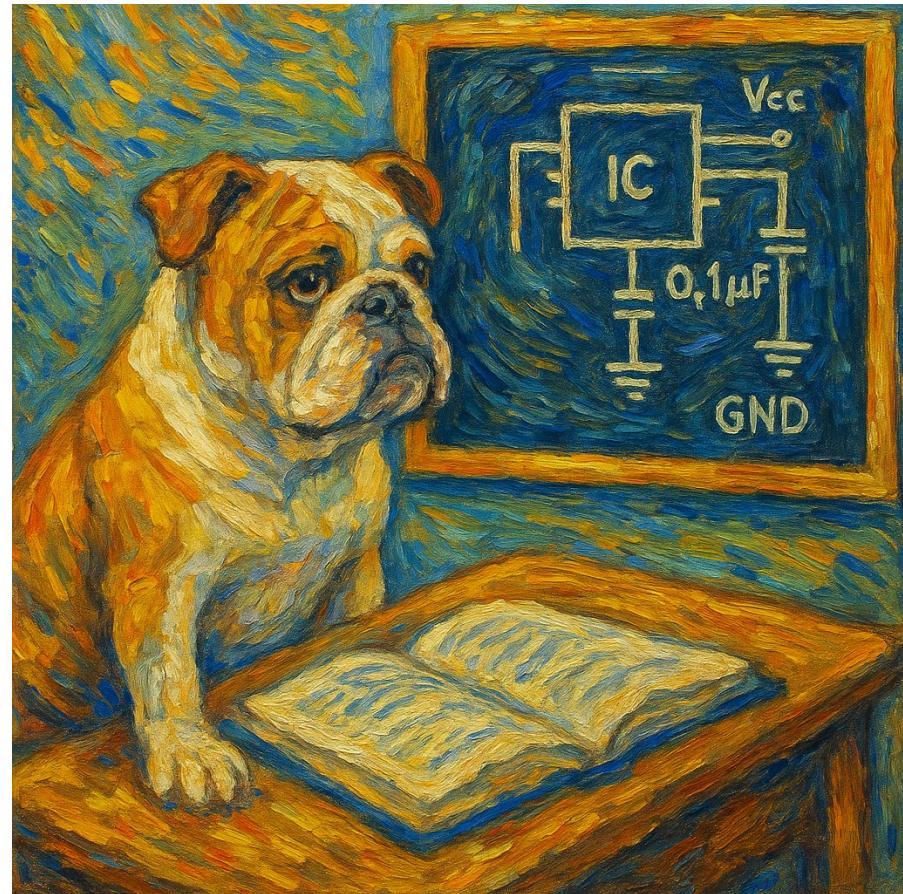
Day 2: Schematic Capture Basics

-
- Review Day 1 Material
 - Review the PCB Design Process
 - Schematic Capture Essentials
 - Creating and saving a new project and schematic
 - Component libraries and management
 - Placing and wiring components
 - Hands-on Activity
 - Creating a basic schematic circuit



Day 1 Review

- Notebook LM Audio Review
- GitHub location for course files
 - https://github.com/RogerMoore2/PCB_Workshop/tree/main



PCB Design Process Review

- PCB Design Process Steps
 - Define PCB requirements
 - Design Electronic Circuit
 - **Schematic Capture**
 - Capture Schematic
 - Generate Netlist
 - Using Libraries to Select Standard Components
 - ERC (Electrical Rules Check)



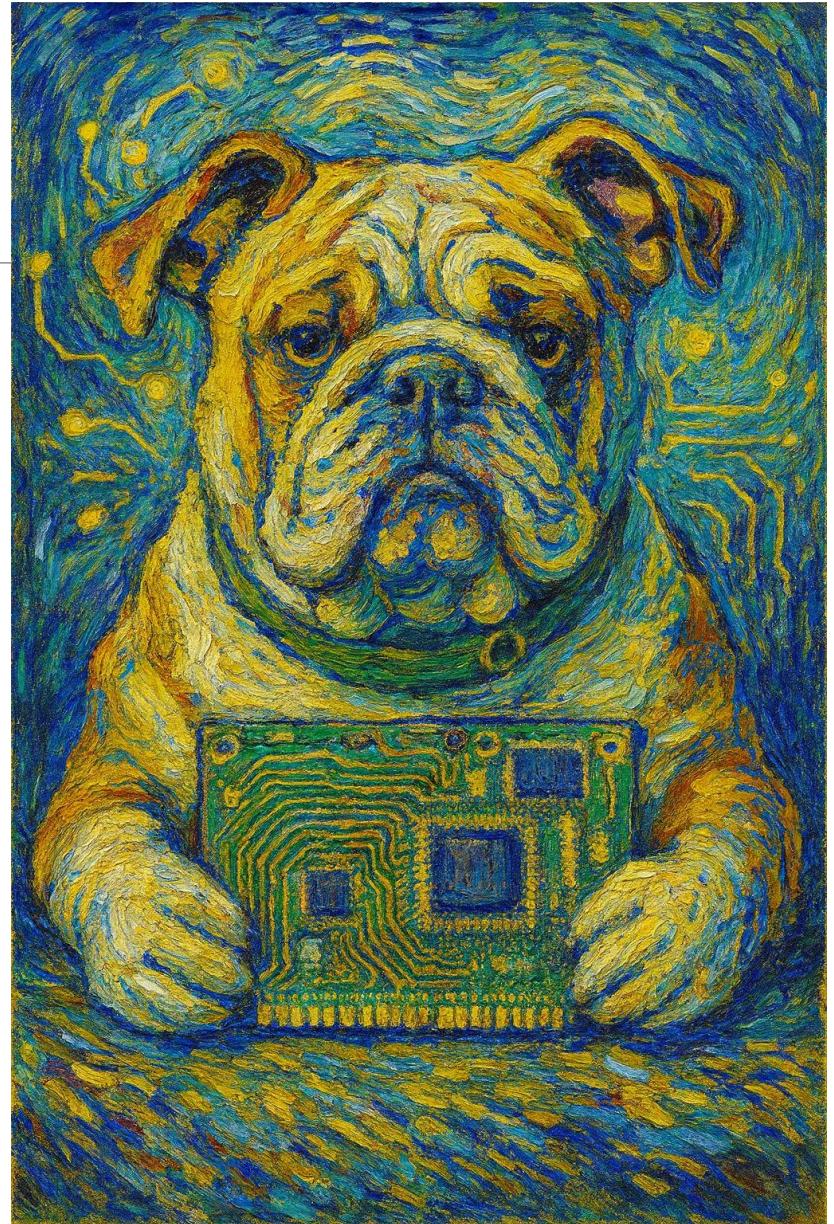
PCB Design Process Review

- Verify Circuit Functionality
 - Simulate
 - Prototype (Solderless Breadboard, DeadBug, etc.)
 - Troubleshoot Prototype issues
- Layout PCB
 - Layout PCB
 - FloorPlanning
 - Ratsnest
 - Design Rule Check

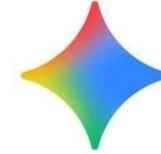


PCB Design Process Review

- Generate Manufacturing and Assembly Files
 - Gerber Files
 - Drill Files
 - Bill of Materials
- Send Files to Manufacturer for PCB Fabrication
- Create PCB Assembly



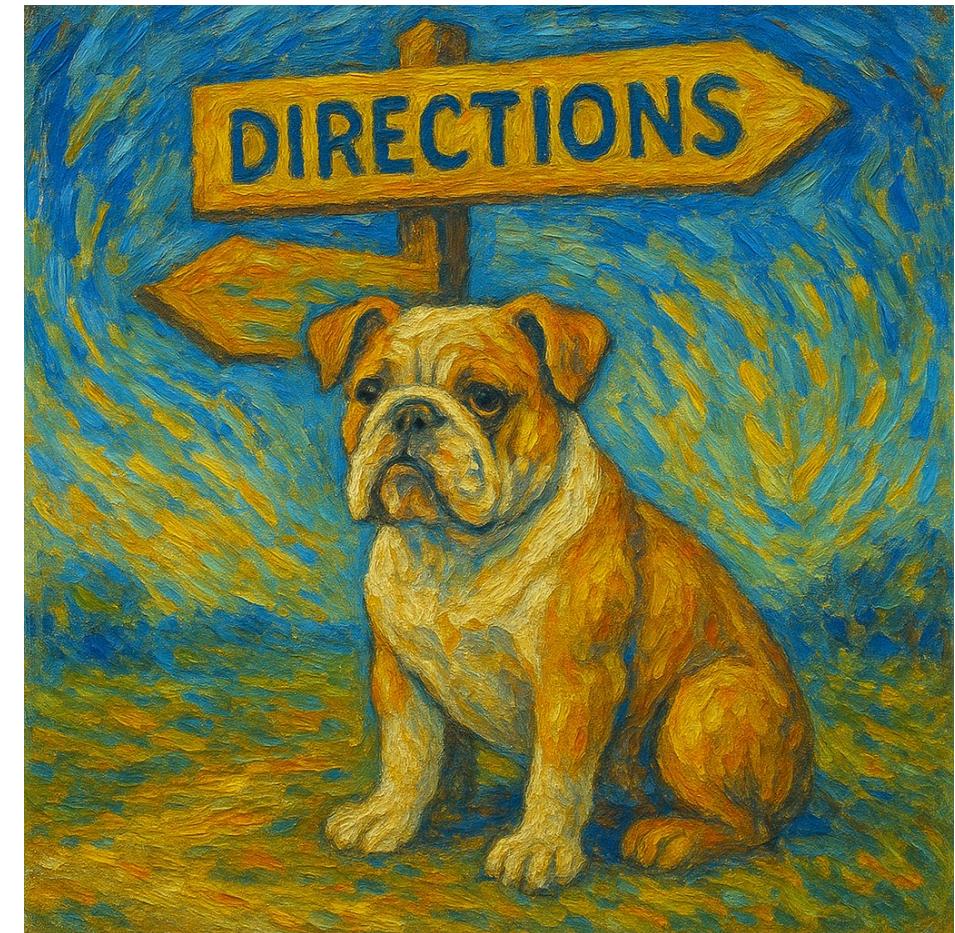
AI Minute



- Please type the following prompts into the AI Tool of your choice, Chat GPT, Gemini, Grok, Copilot, etc...
- What is a decoupling capacitor and why should I include them in my electronics designs?
- What are the methods of determining the value of a decoupling capacitor, what are the advantages and disadvantages of each, and which method is best?
- What type of capacitor should I use for a decoupling capacitor, e.g. dielectric, and where should it be located?

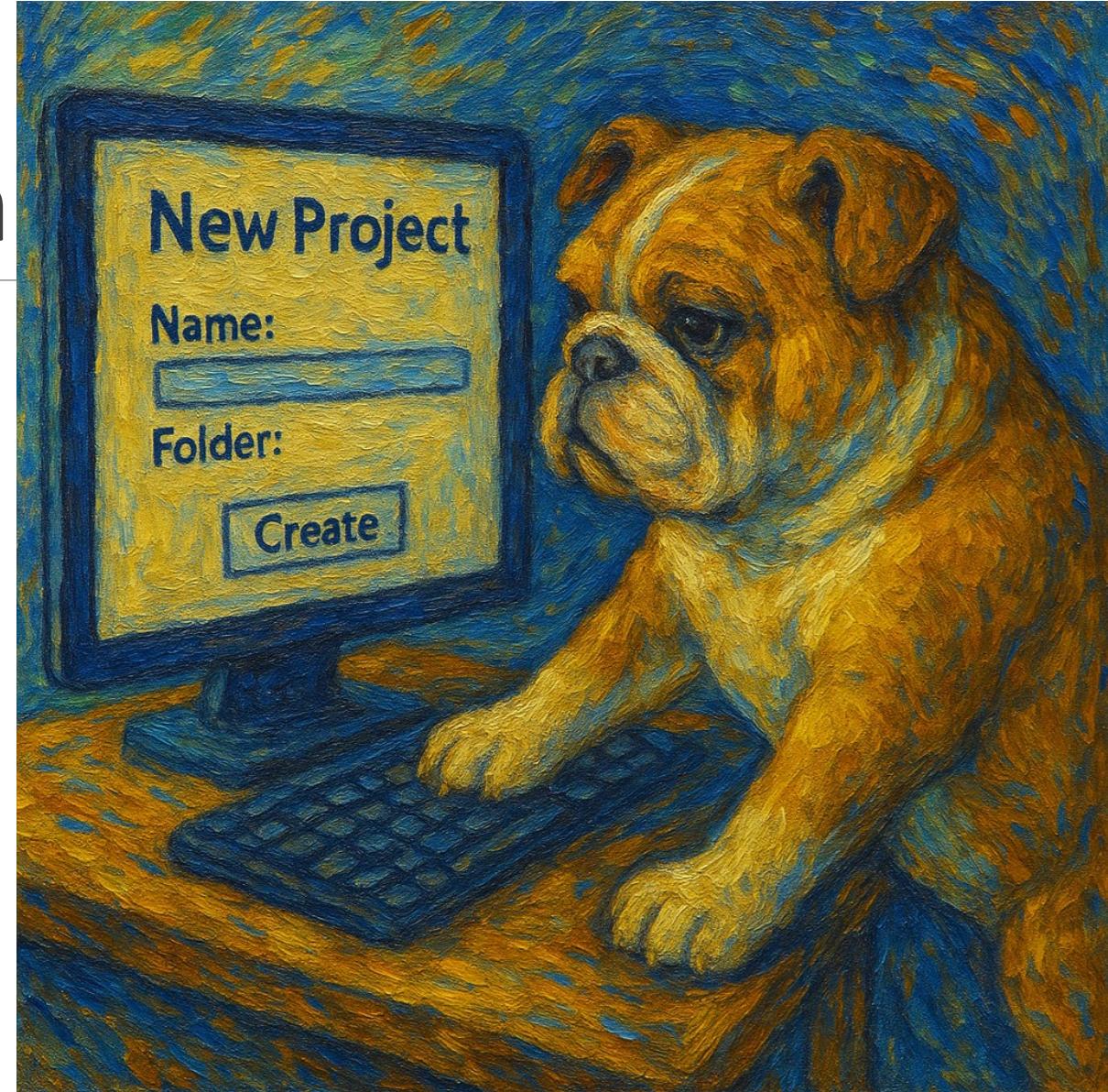
The Fusion Schematic Environment

- Schematic Navigation



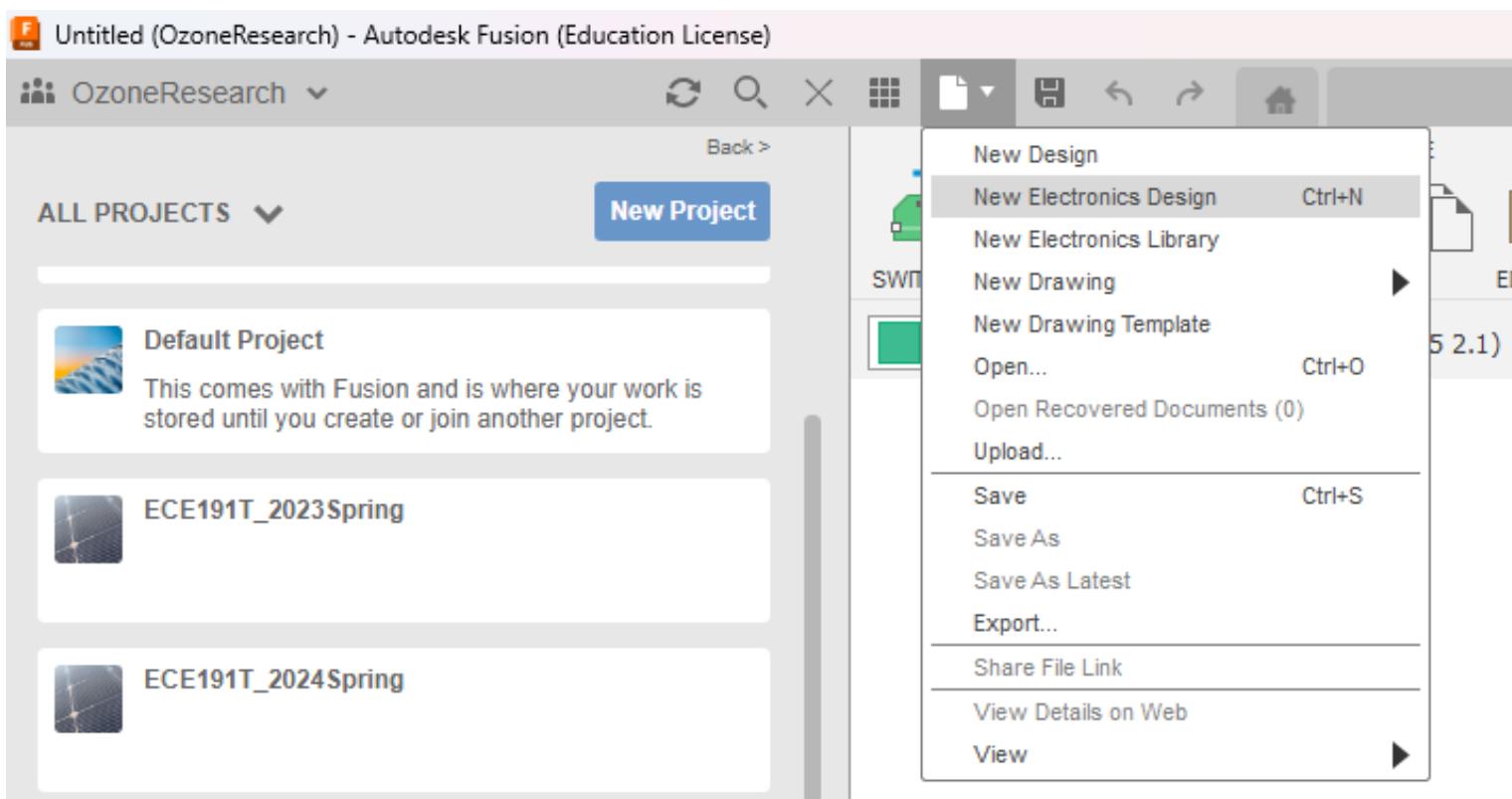
New Project In Fusion

- Let's Create a New Project in Fusion



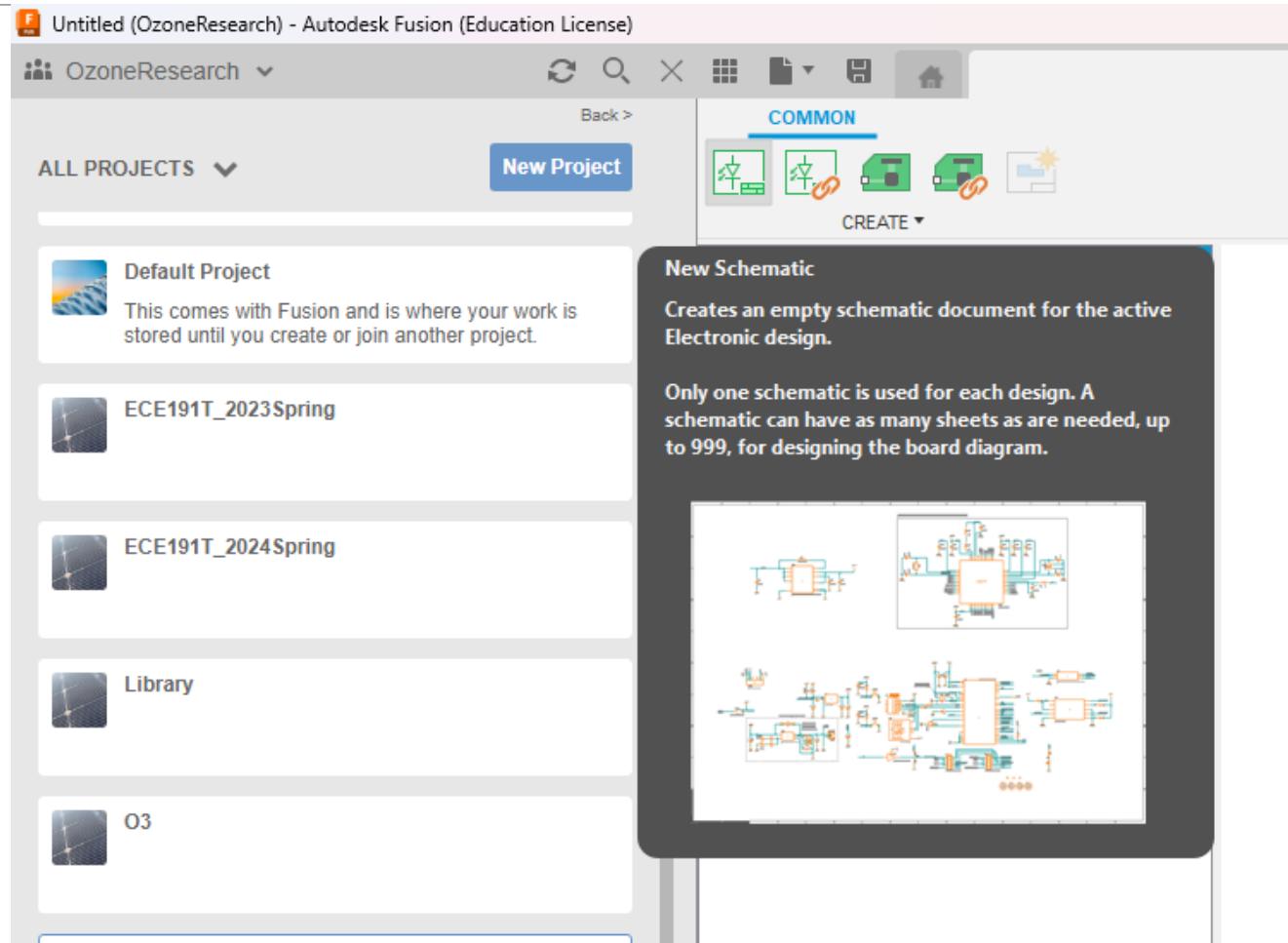
Create a New Project In Fusion

- Click on the paper icon in the ribbon
- Select “New Electronics Design” or type “Control+N”



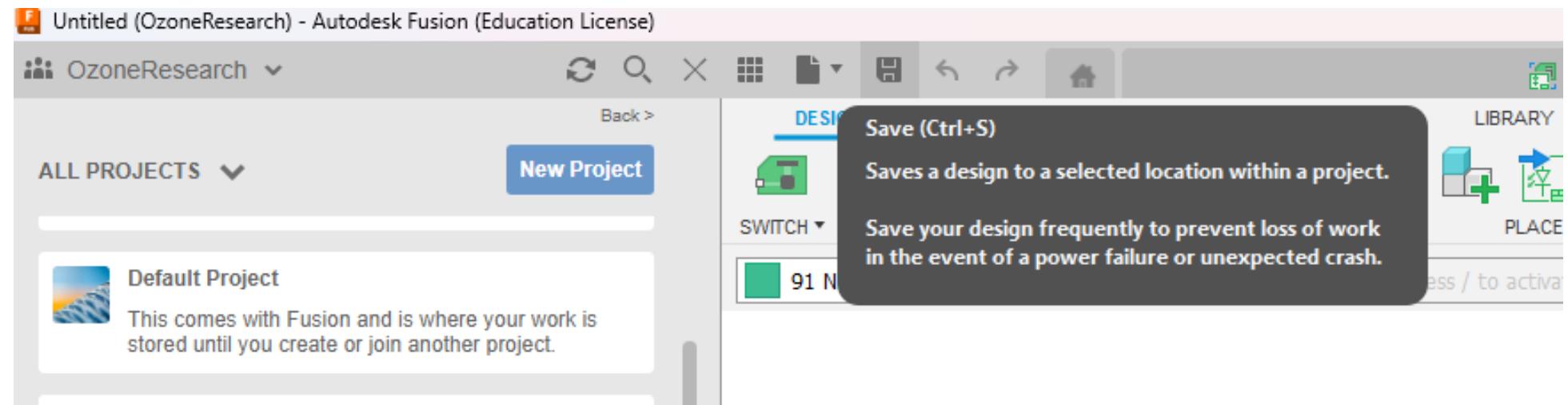
Create a New Schematic

- Click on the schematic icon under “Common”



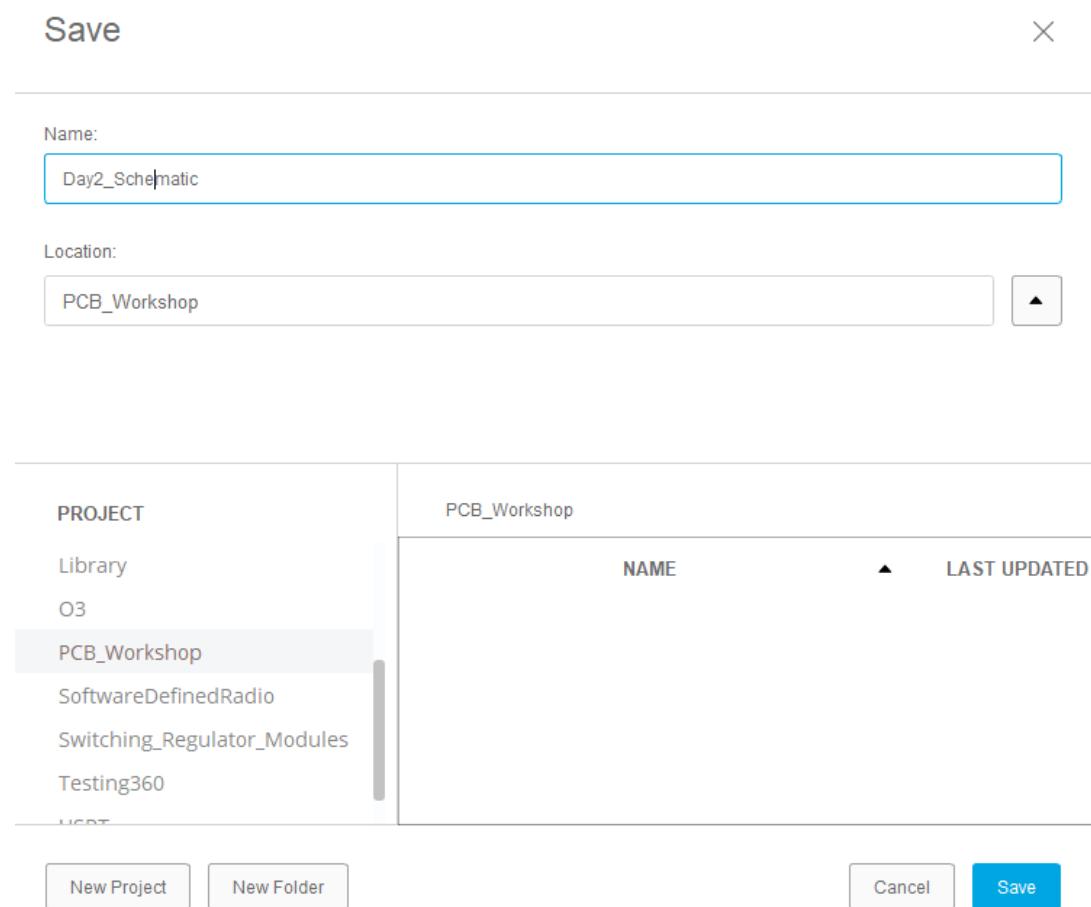
Save the Schematic with a New Name

- Click on the disk icon in the ribbon
- Enter a name



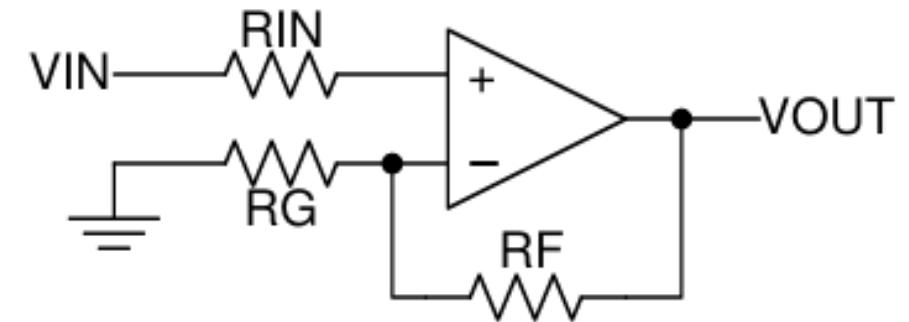
Save the Schematic with a New Name

- Enter a name
- Verify the location folder is correct
- Click the “Save” button



Create a Schematic of a Circuit

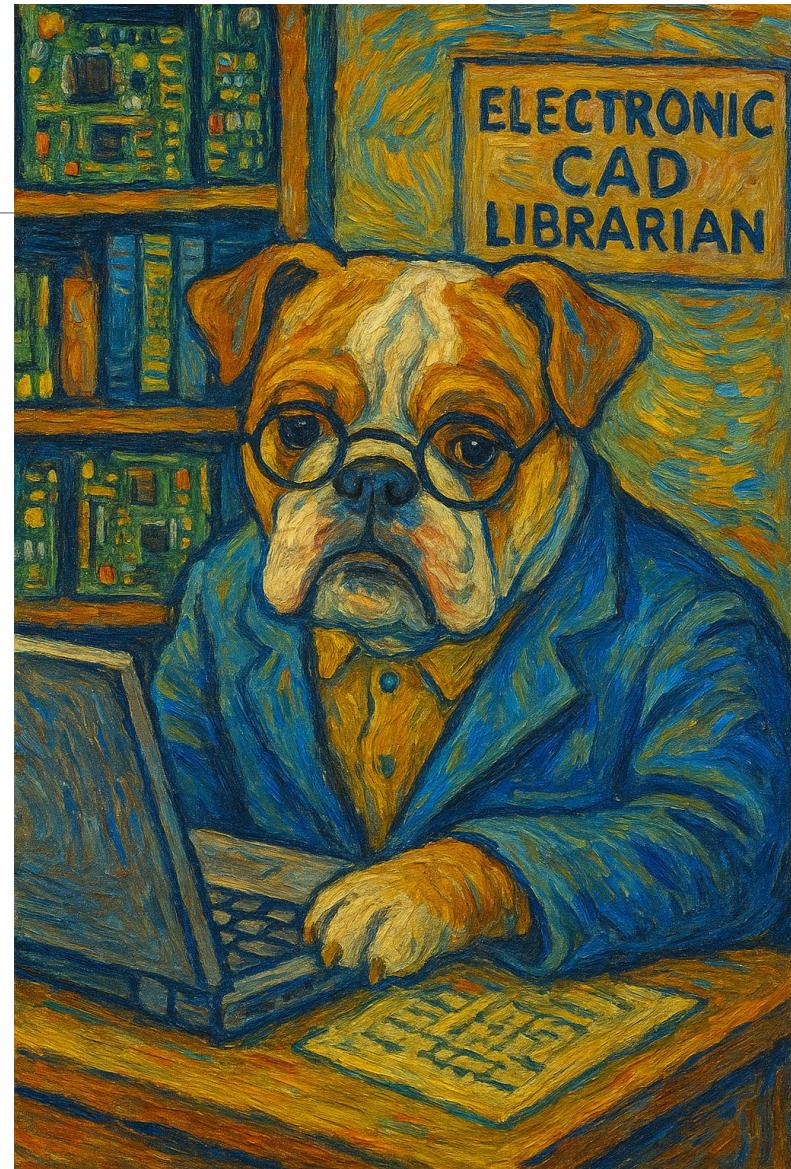
- Op Amp Non-Inverting Amplifier
- Basic Schematic Included on this Slide
- Use a LM358 Op Amp
- Add a 4-pin header
- Add a +5V supply
- Add GND symbols to ground connections
- Add a ceramic decoupling capacitor between +5V and GND
- Configure the unused amplifier for non-inverting unity gain and add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



Place Holder

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



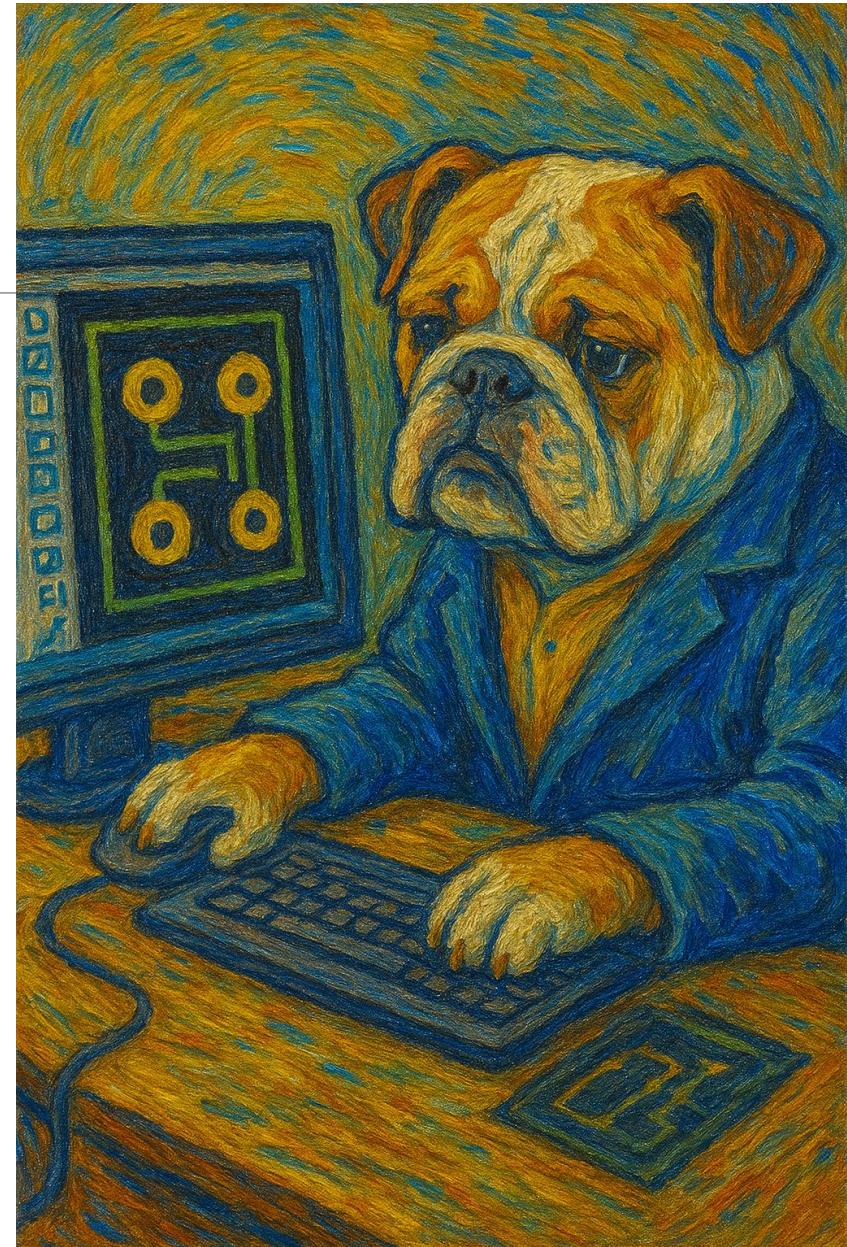
Fusion CAD Libraries

- Managed Libraries

- Maintained by AutoDesk or Partners
 - Staff engineers are tasked with maintaining component libraries

- User Libraries

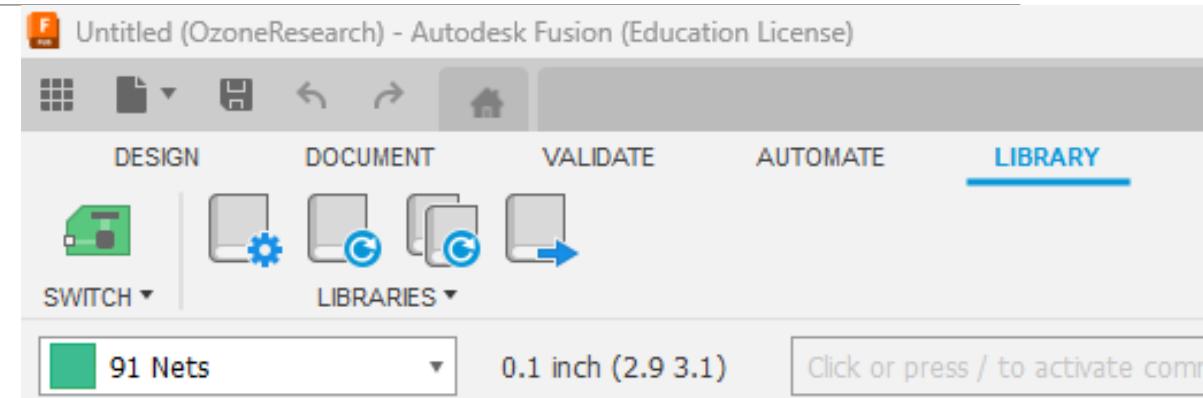
- Users can create their own libraries
 - Maintained by the user
 - New components can be added by the user
 - Useful for custom or unsupported components



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?**

The screenshot shows the AutoDesk Fusion Electrical Library Manager window. The title bar says "Library Manager". Below it, there are three tabs: "Hub Libraries", "Public Libraries" (which is selected and highlighted in blue), and "Private Libraries".
On the left, there is a "Filters" sidebar with the following sections:

- Status:** Includes "Active" (unchecked) and "Inactive" (unchecked) checkboxes.
- Used in:** Includes "In this design" checkbox.
- Updates:** Includes "Updates available" checkbox.
- Source:** Includes "Fusion Electronics" and "3rd Party Libraries" checkboxes, with "All Publishers" dropdown below.

The main area displays a table titled "1055 Results" with the following columns:

Library	Folder Name	Version	Active
arduino_due_shield	User Submitted Libr...	1	(checkbox)
arduino_jw	User Submitted Libr...	1	(checkbox)
ArduinoNanoV30	User Submitted Libr...	1	(checkbox)
ase	Eagle Pcb	1	(checkbox)
ast3tq_10	User Submitted Libr...	1	(checkbox)
atmel	Eagle Pcb	5	(checkbox)
atmel-at30ts75x	User Submitted Libr...	1	(checkbox)
atmel-sama5d3	User Submitted Libr...	1	(checkbox)
ATSAME7ON21	User Submitted Libr...	2	(checkbox)
ATSAME7Q21	User Submitted Libr...	2	(checkbox)
audio amplifier example	Eagle Pcb	4	(checkbox)
Audio Connectors	Hetal @pcblayout	1	(checkbox)
Audio-Devices	Fusion Electronics	9	(checkbox)
austriamicrosystems	Eagle Pcb	2	(checkbox)
avago	Eagle Pcb	3	(checkbox)
avago-hlcp-j100	User Submitted Libr...	1	(checkbox)

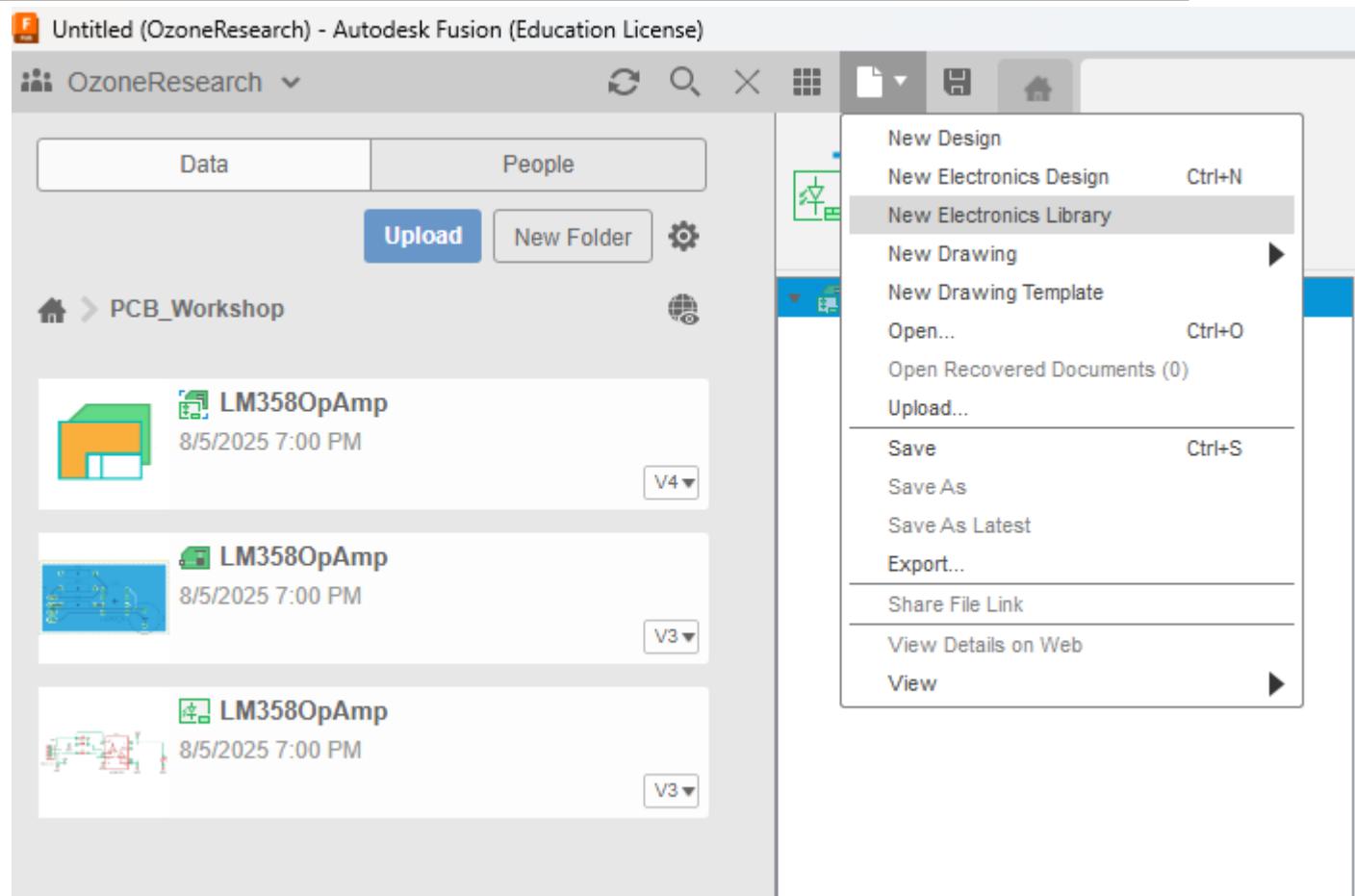
Create a Library

- 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

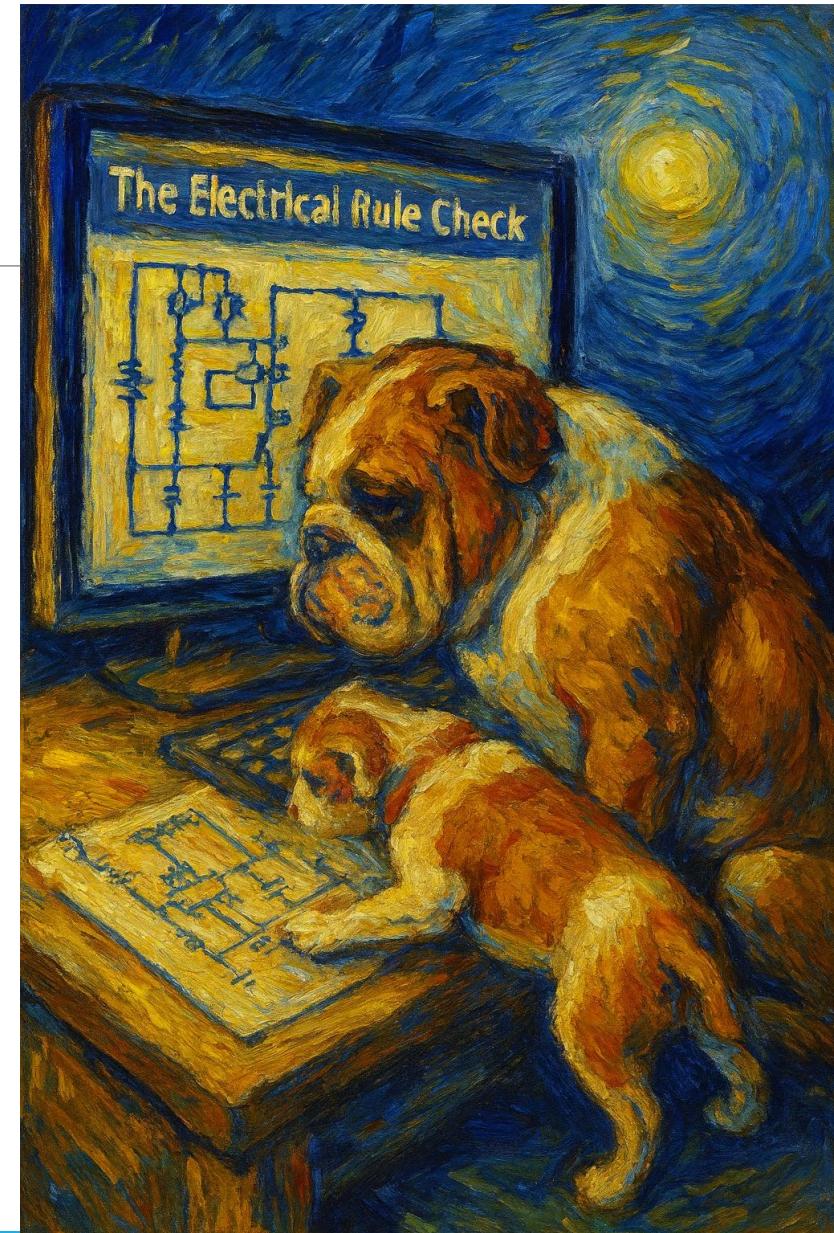
- Create a new Electronics Library

- From the file icon, select “New Electronics Library”
- Click the “Save” icon and give the library a name



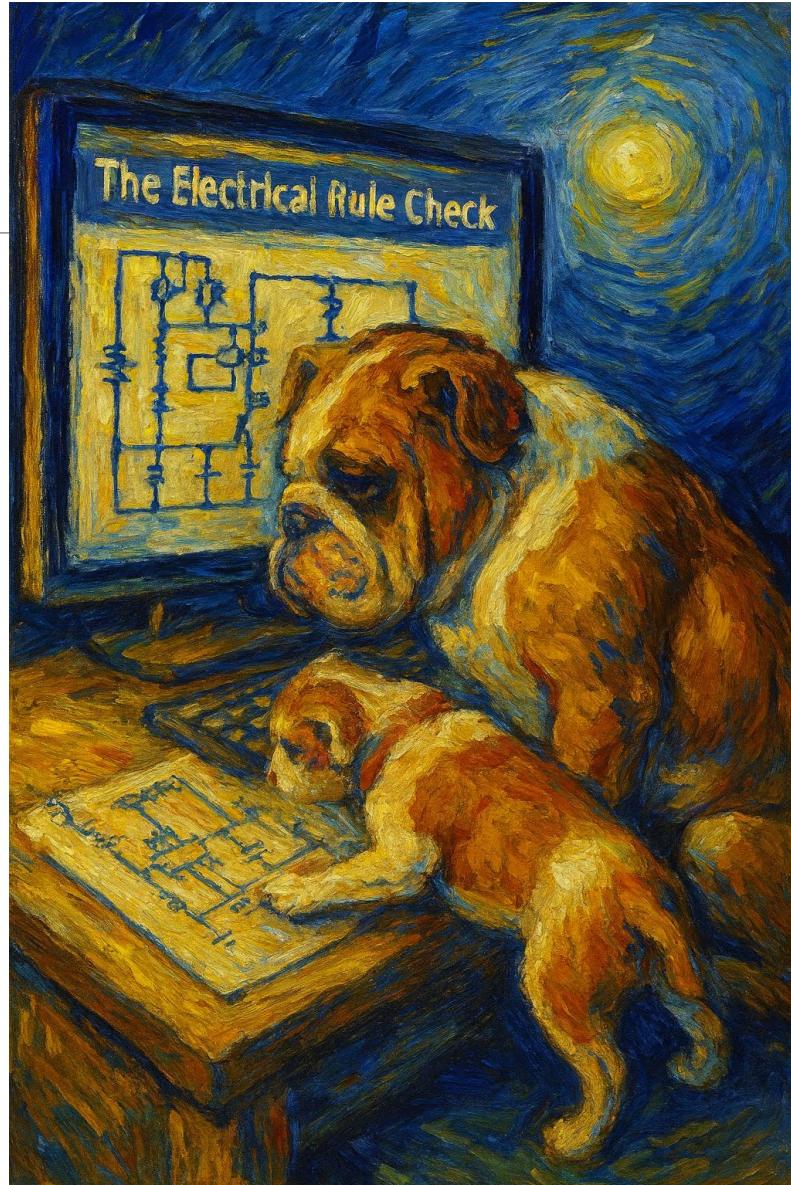
Schematic ERC Check

- **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.



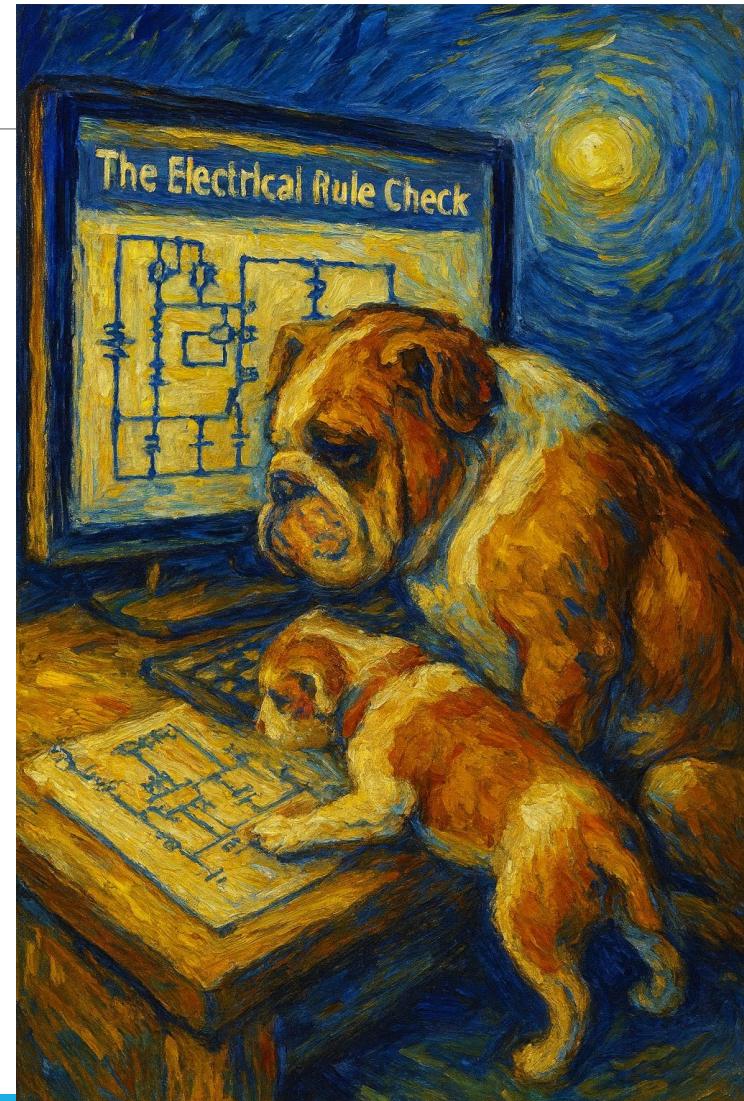
Schematic ERC Check

- 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.



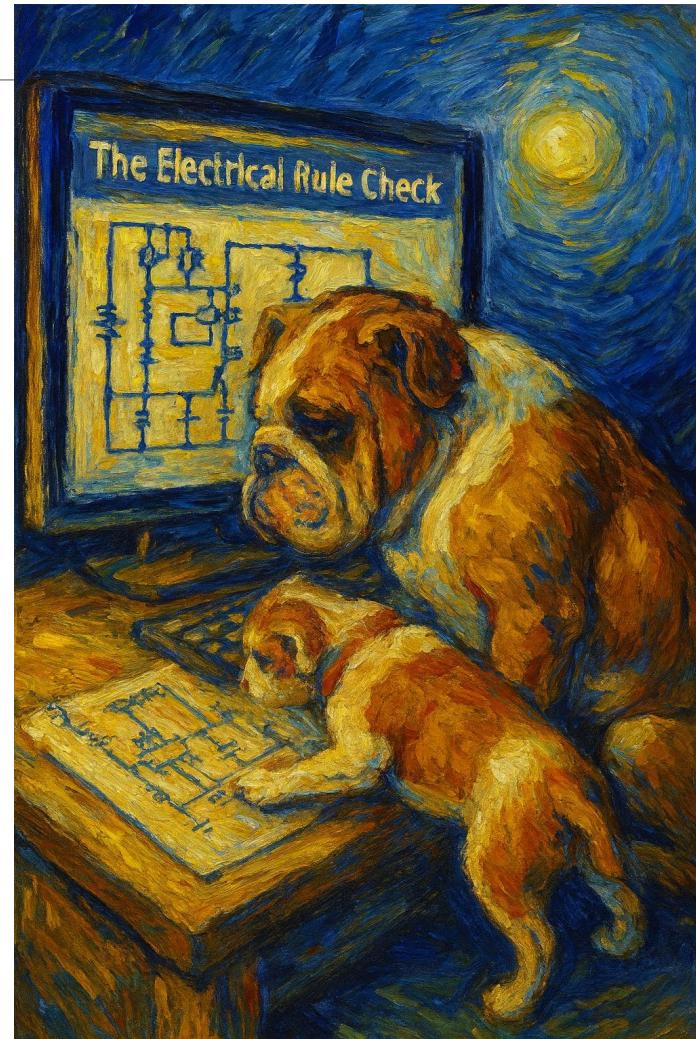
Schematic ERC Check

- 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.



Questions?

