http://paintingandframe.com/prints/pierre_auguste_renoir_vase_of_roses-797.html

Physics 524

Survey of Instrumentation and Laboratory Techniques

2024

University of Illinois at Urbana-Champaign

Unit 1c: Schematic Capture

Goals for this unit 2
EAGLE PCB startup 2
Add parts to the circuit 5
Parts in the library 12
Board layout tool 14
Production/fabrication files 19

Goals for this unit

- Learn how to do (electronic circuit) schematic capture
- Learn to do PCB layout

EAGLE PCB startup

Visit https://www.autodesk.com/education/free-software/eagle and sign in. Follow the steps to download EAGLE, Autodesk's schematic capture and printed circuit board layout tool. There are good tutorials on how to install, then use EAGLE on the sparkfun.com web site if you'd like additional information to what I present below.

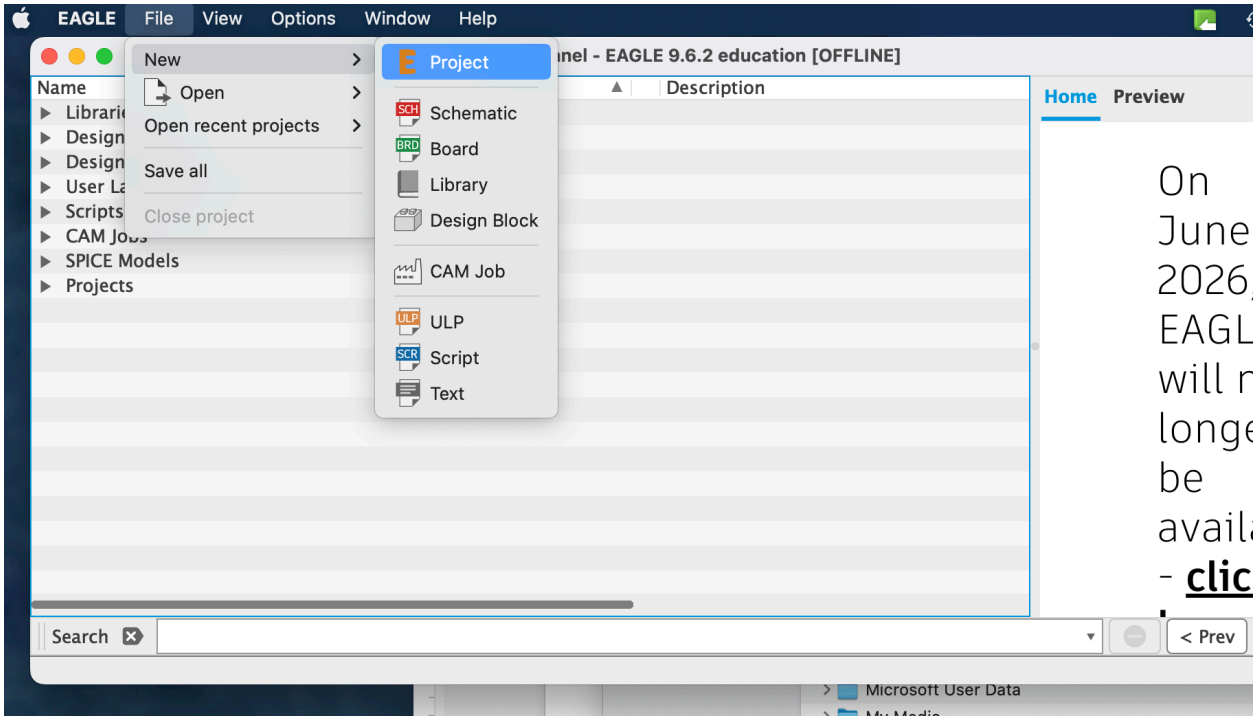
We'll do the following this unit:

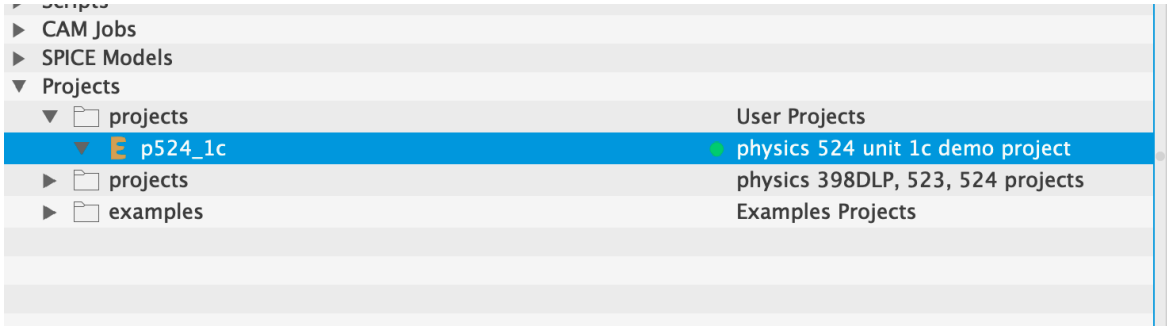
- Install EAGLE and load the library GDG_library.lbr into the folder external_lbrs in your PCB working area;
- Create the schematic for a simple circuit in which an Adalogger M0 reads a BME680 T/P/RH/VOC sensor and displays data to an LCD;
- Design the printed circuit board for your circuit and create the fabrication files necessary to have it built by the JLCPCB fabricator.

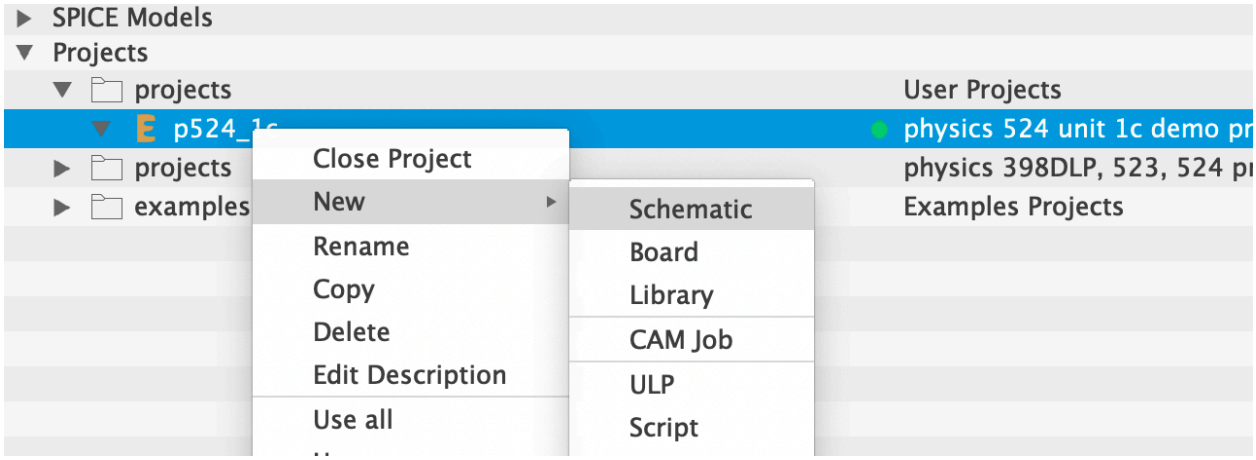
I'll have you follow along on your laptops as I do these things on mine.

Please fire up EAGLE and open a new project, then open a new schematic in the project. See screen shots below.

 ©George Gollin, 2024

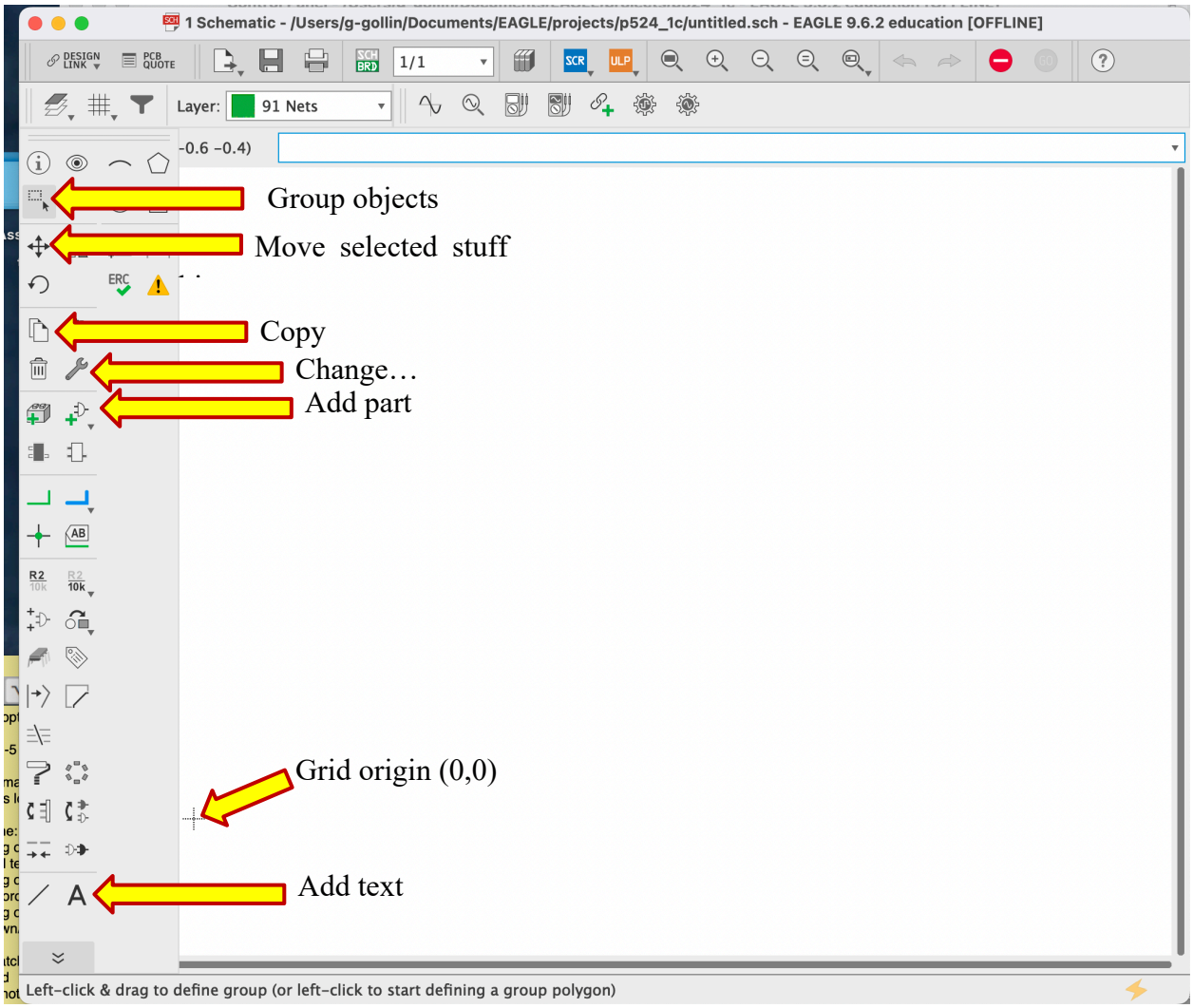






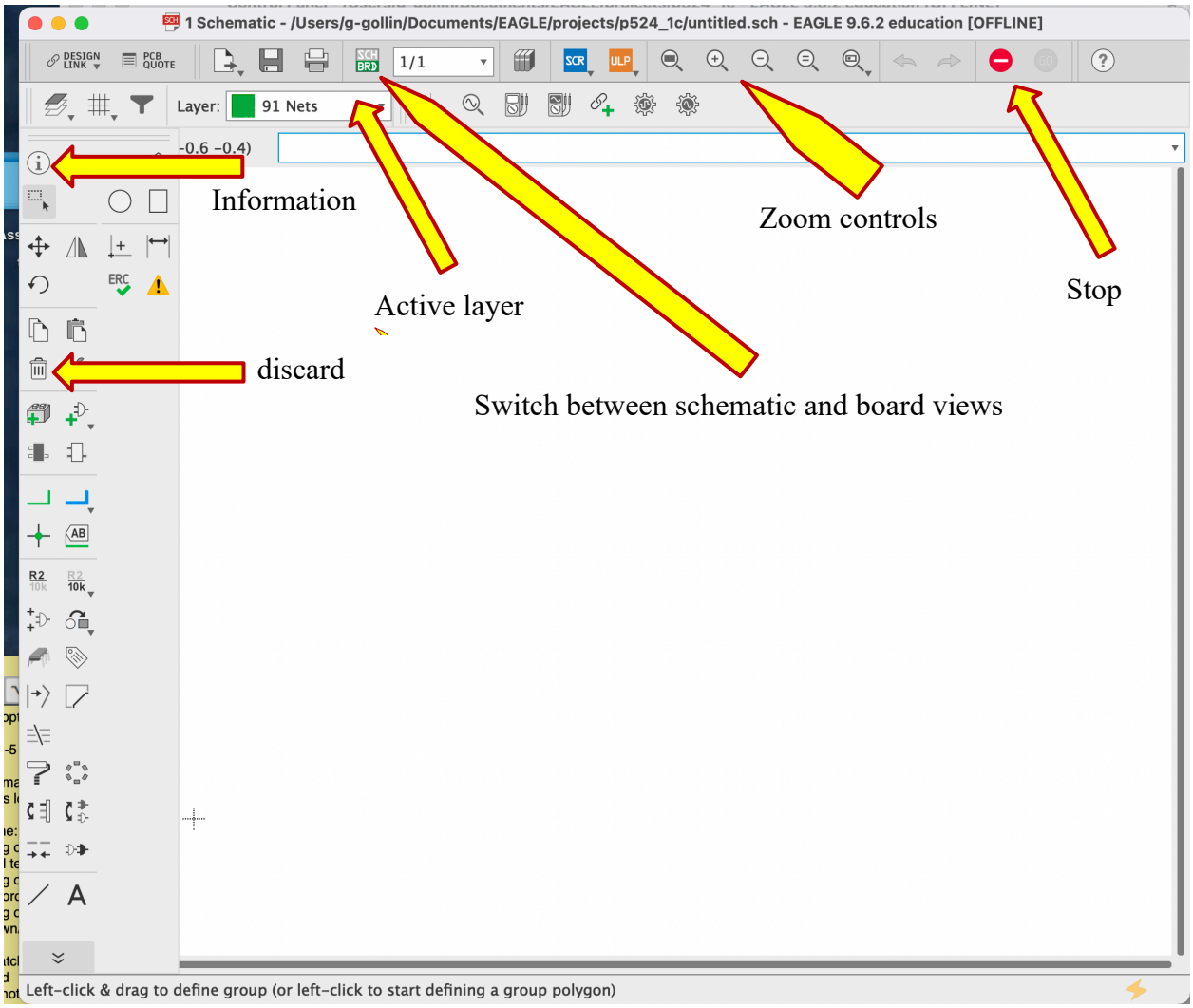
 ©George Gollin, 2024

You'll get an empty window that looks something like this:



 ©George Gollin, 2024

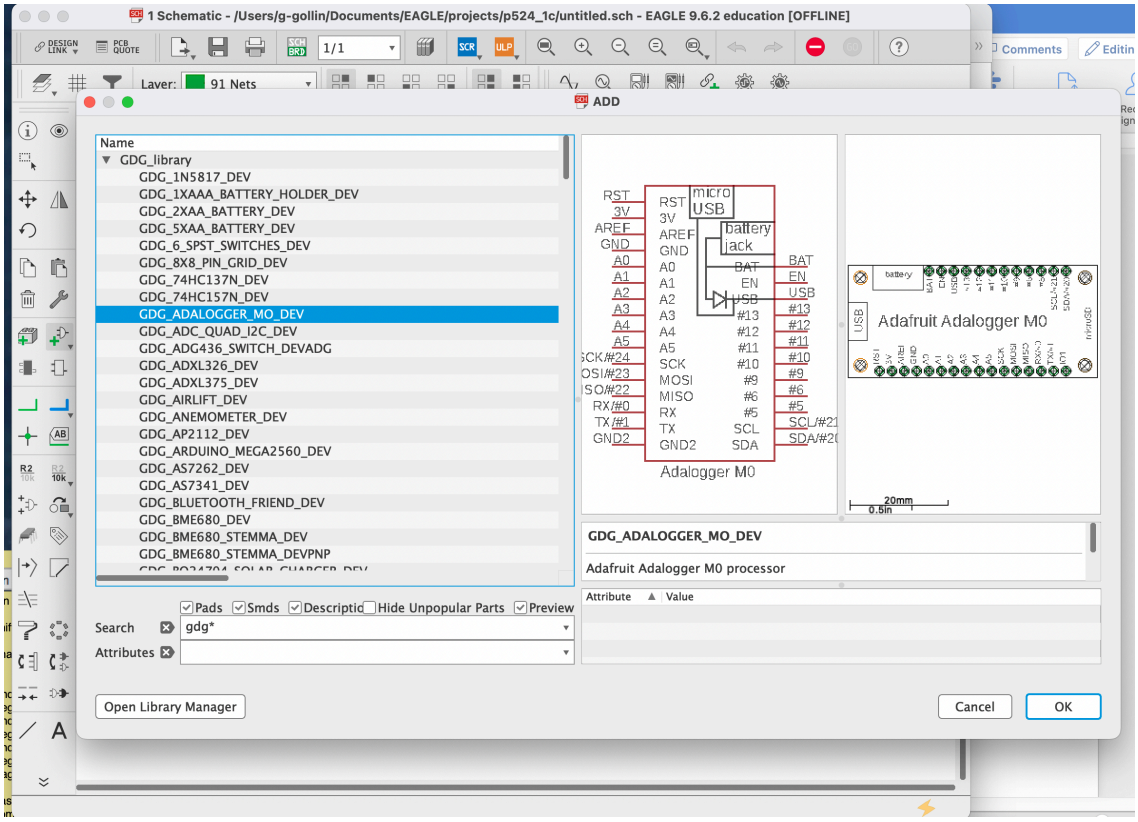
More about the window…

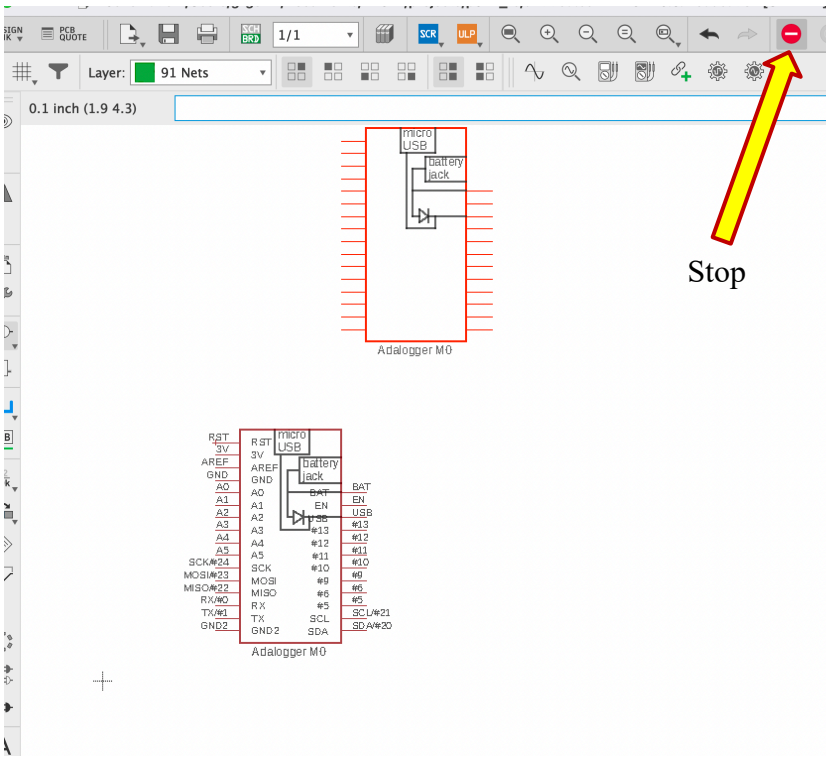


Add parts to the circuit

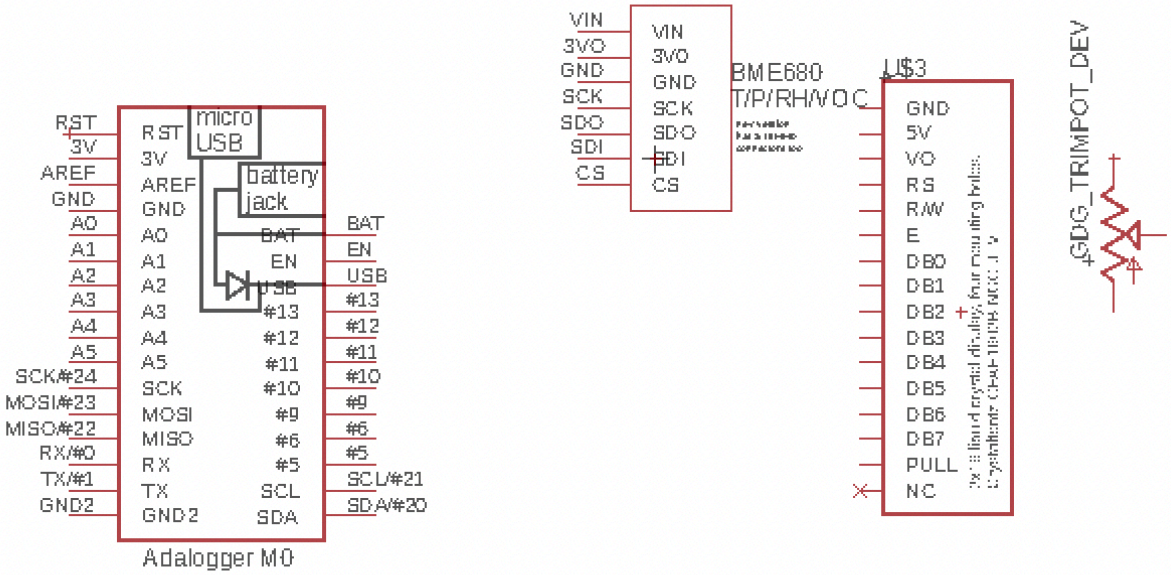
We want to use an Adalogger, a BME680, an LCD, and a 10k adjustable resistor (for setting LCD contrast). Click on the "add part" tile and look for GDG_M0_ADALOGGER_DEV in the parts menu, then click OK. Use "stop" after adding a single Adalogger to the schematic. Add the other parts in similar fashion.

 ©George Gollin, 2024

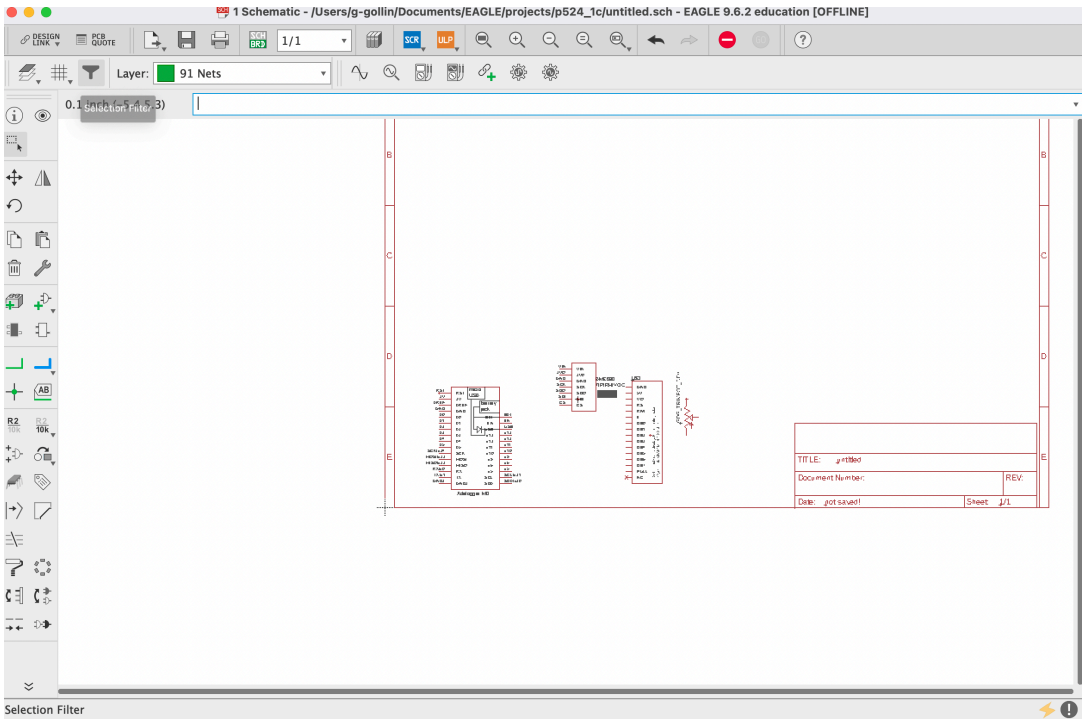




 ©George Gollin, 2024

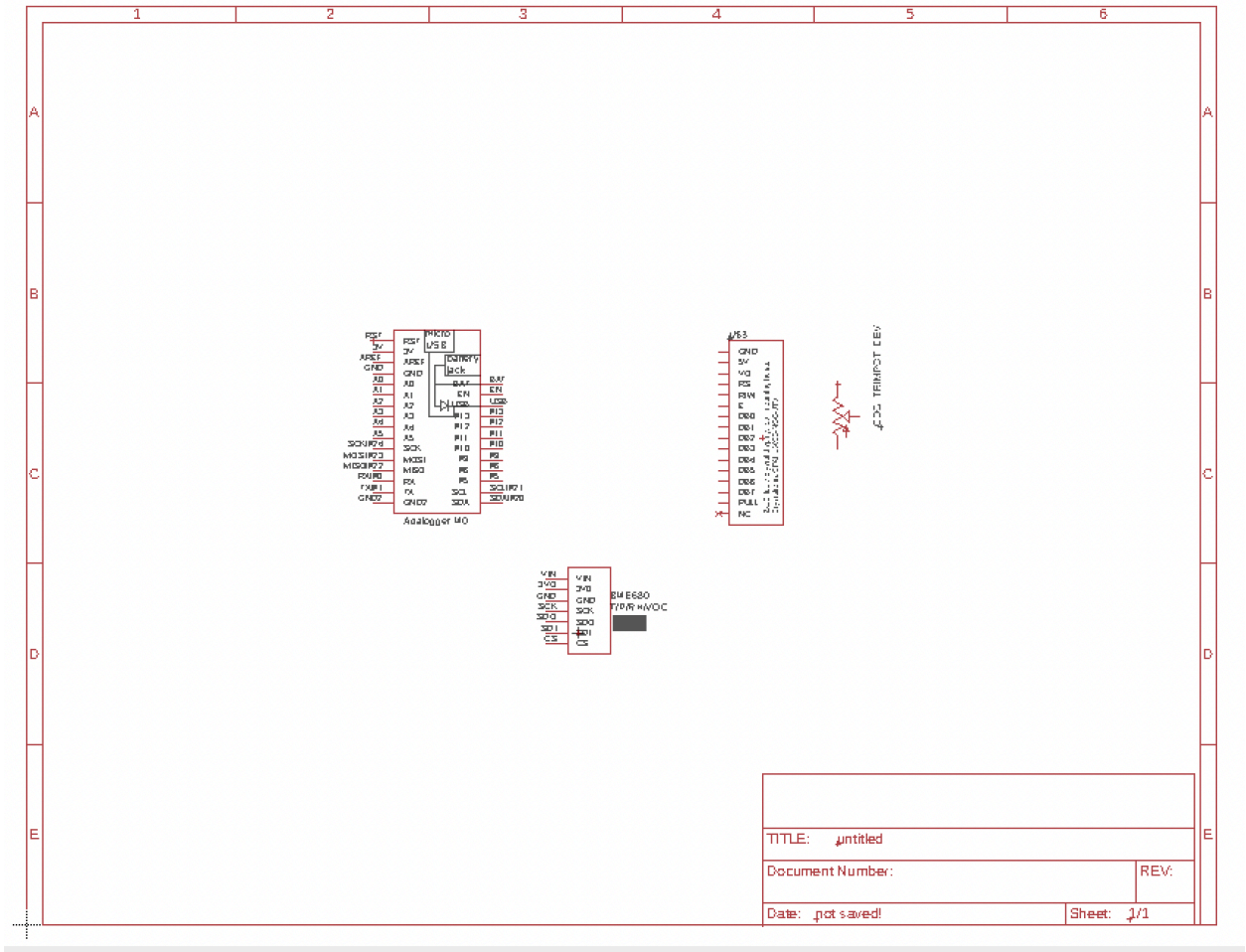


Now let's put a frame around the entire schematic, with the lower eft corner at the origin. In the add parts menu search for *frame* and select FRAME_A_L.



Use one of the Zoom controls to fill the window with everything we have placed so far. Then select the move stuff tile, click on the origins of various parts, and move them so that they are better centered in the frame. Oh—it's a good idea to do a "save all" from time to time.

 ©George Gollin, 2024

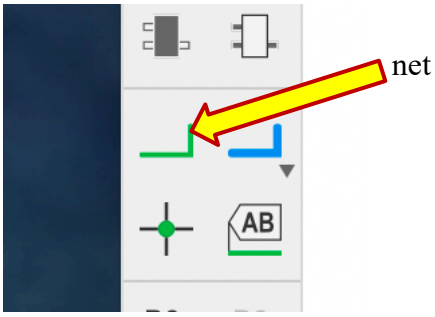


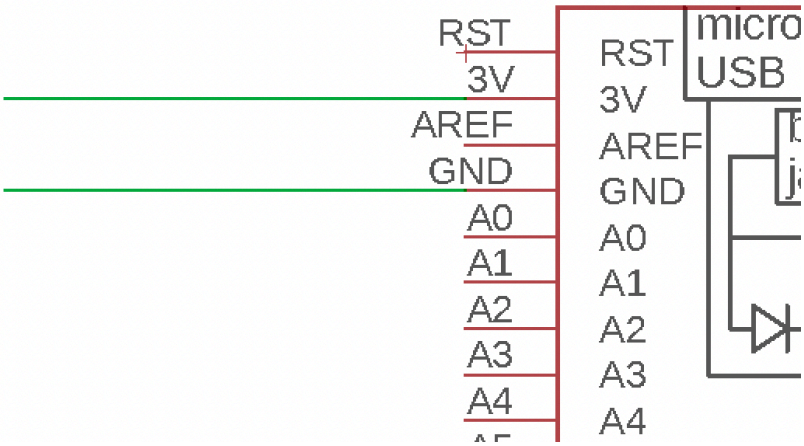
Now connect everything up! Let's wire the LCD and BME680 this way:

Adalogger pin	LCD pin	Adalogger pin	BME680 pin	LCD pin	10k trimpot
GND	GND	3V	VIN	5V	Left pin
3V	5V	GND	GND	PULL	Right pin
12	RS	SCL/#21	SCK	VO	Center pin
GND	R/W	SDA/#20	SDI		
11	E				
5	DB4				
6	DB5				
9	DB6				
10	DB7				

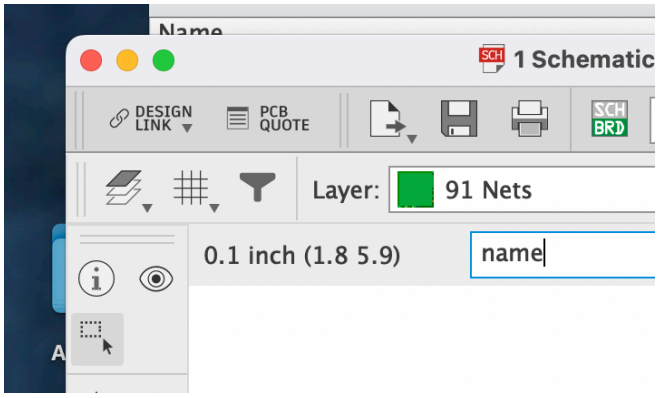
First use the net tile to draw nets from the Adalogger GND and 3V pins, bur don't bother to connect them to anything.

 ©George Gollin, 2024



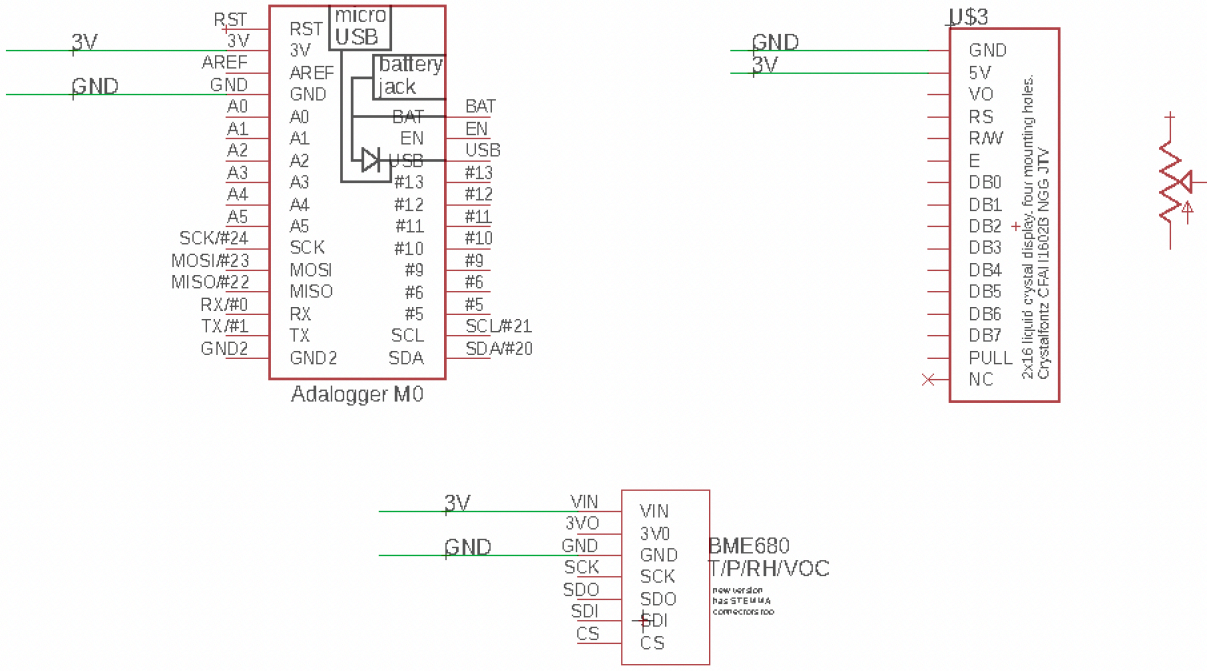


Now type “name” into the command entry field and click on the net lines you’ve drawn, one at a time, renaming them as GND and 3V.



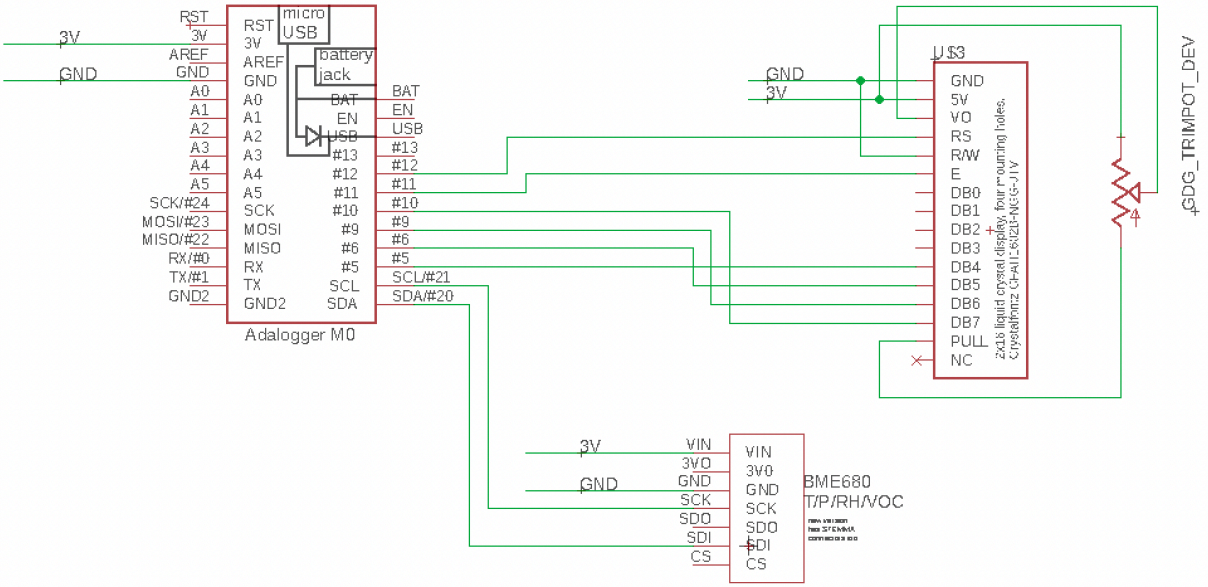
Do the same for the BME680 and LCD pins that are supposed to be connected to GND and 3V:

 ©George Gollin, 2024



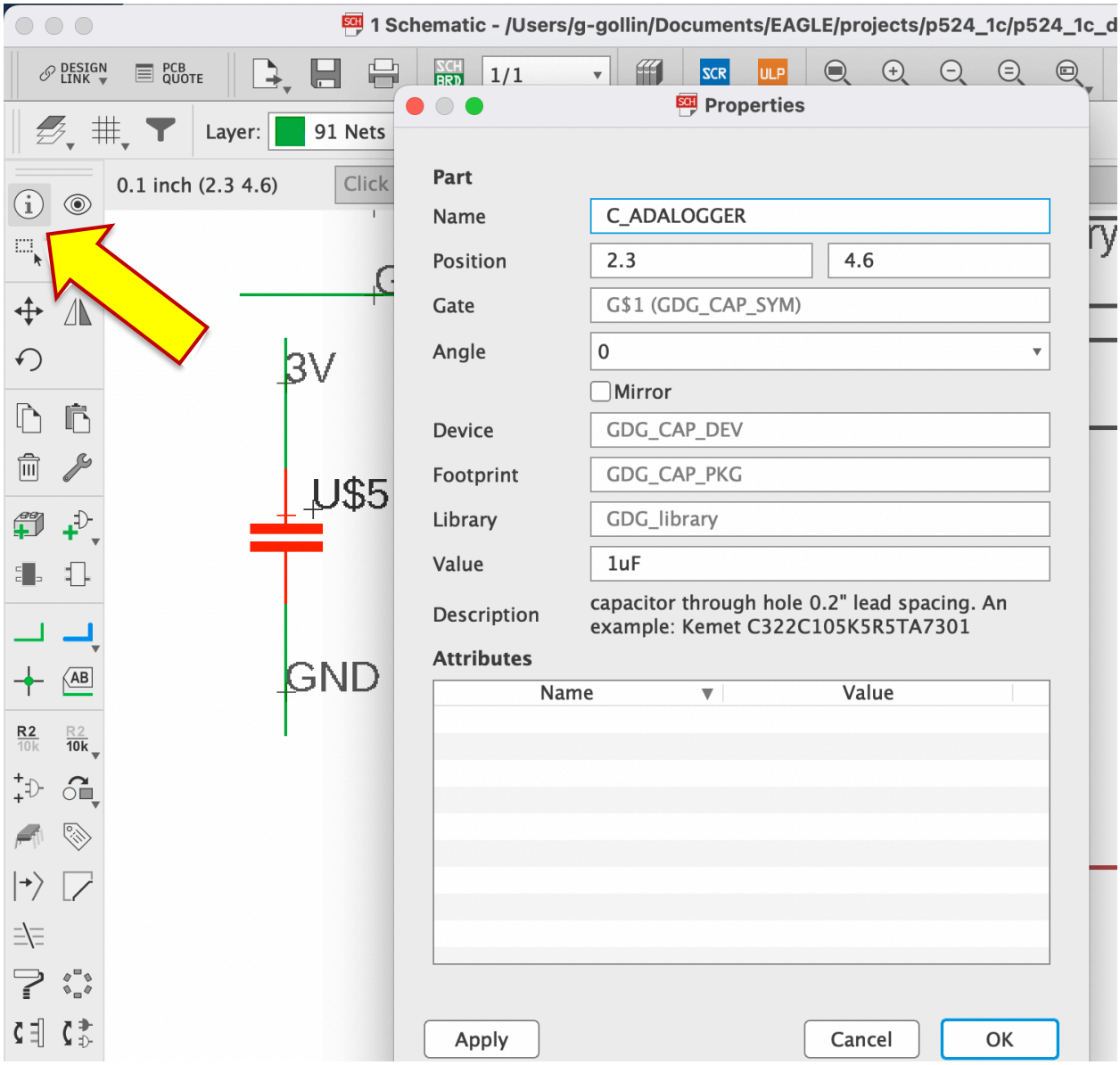
The schematic tool will know during PCB design that these pins need to be connected. It's entirely your choice—whether to label nets or to run the connections from pin to pin—and you should develop your schematic with an eye towards readability and clarity.

Here's how my version of the circuit looks.



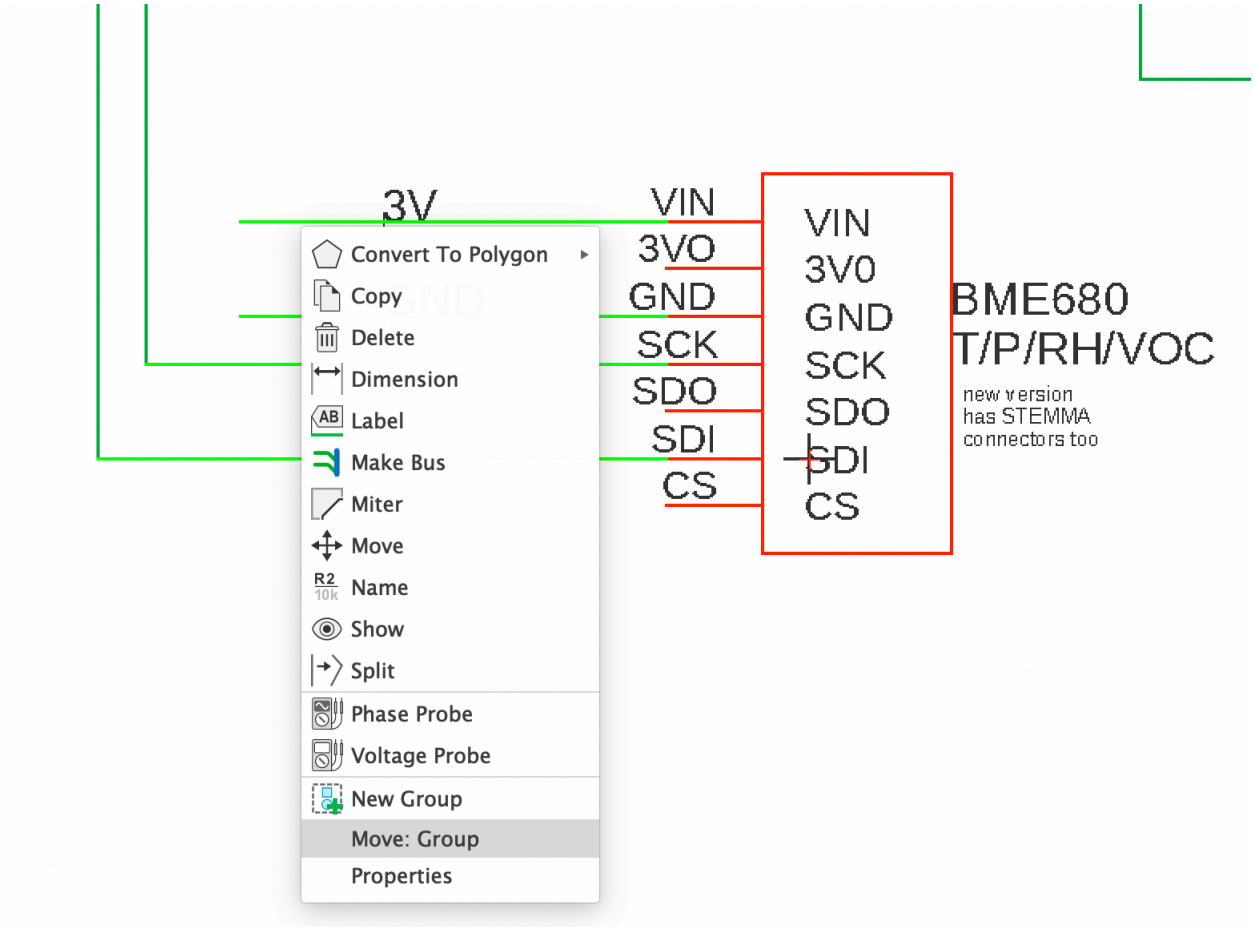
Now add a few 1μF capacitors (one per device) to the circuit. Use the information window to set an appropriate name as well as the capacitance value.

 ©George Gollin, 2024



Here's something useful: you can use the "group objects" tile to select multiple objects. If you then click, for example, the "move objects" tile, you can right click, then select "move group" to move all the selected objects.

 ©George Gollin, 2024



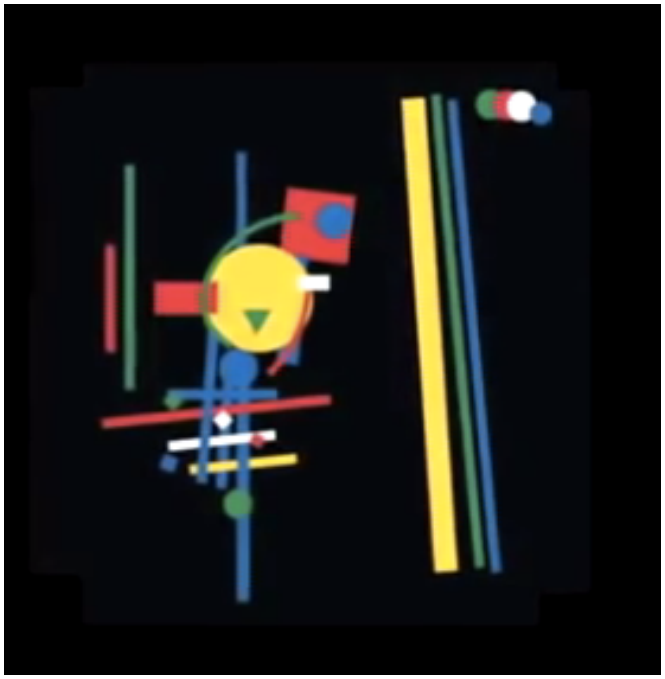
Other group operations can include copy group, delete group, and others.

Parts in the library

The representation of a part includes both a symbol that can be dropped into a schematic and a footprint that describes how that part appear on the PCB that you'll fabricate.

Did any of you play with Colorforms when you were little? It's a creative toy which includes a variety of thin vinyl shapes—circles, rectangles, and the like—that will adhere to the smooth, black surface that a child uses as a drafting environment.

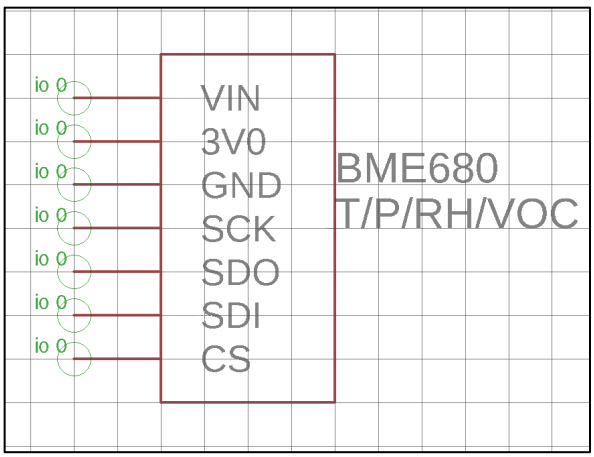
 ©George Gollin, 2024

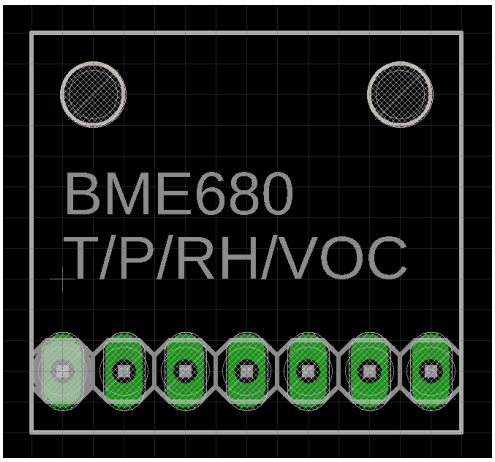


A schematic capture tool works in similar fashion: the designer selects parts from a library of components, places them on the design surface, and draws the interconnections between component pins.

The data characterizing each part in the library include two distinct representations: one specifying just the pins to which connections are made, and another that incudes geometric information. We use the first to define the topology of the circuit under construction, while we employ the other to decide where we can actually place components on a circuit board.

An example: here are the symbol and package (or footprint) for a BME680.

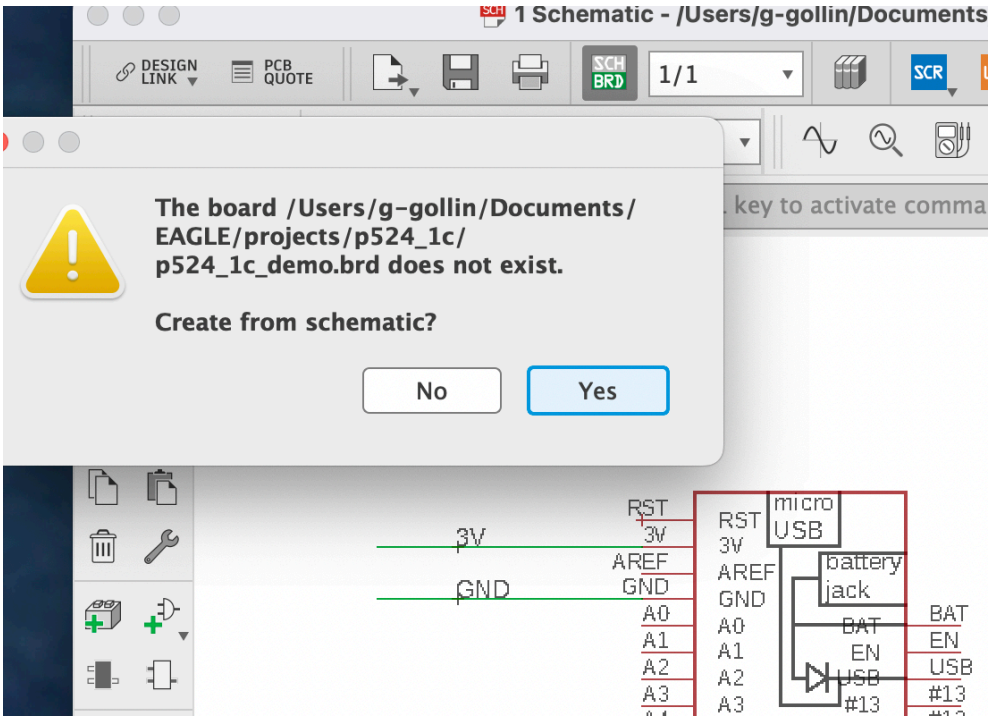




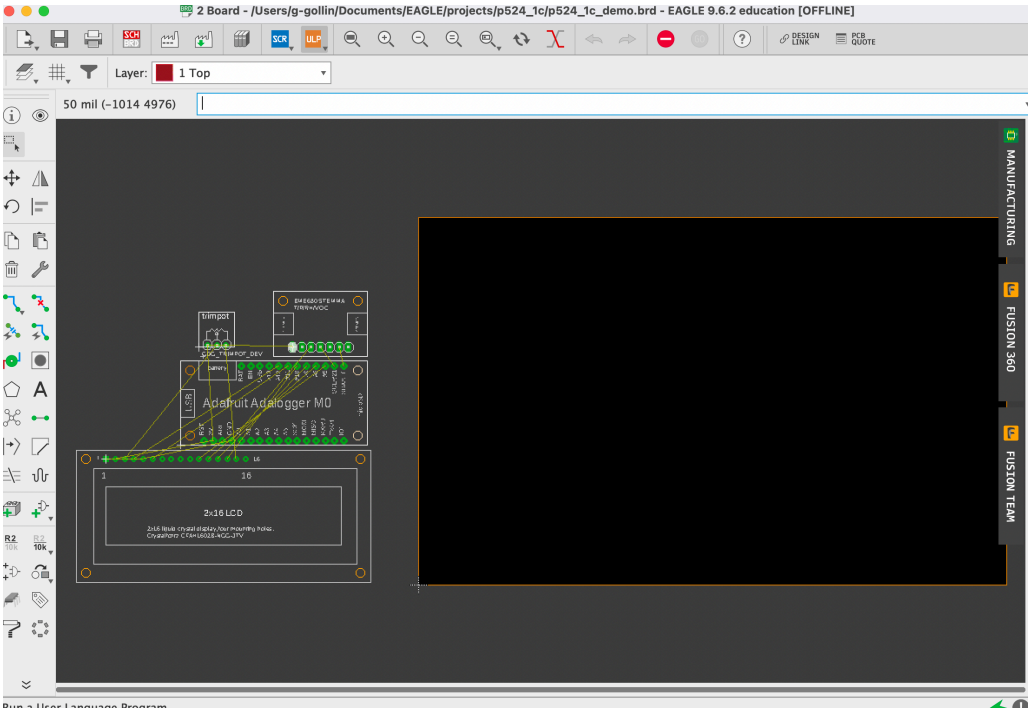
 ©George Gollin, 2024

Board layout tool

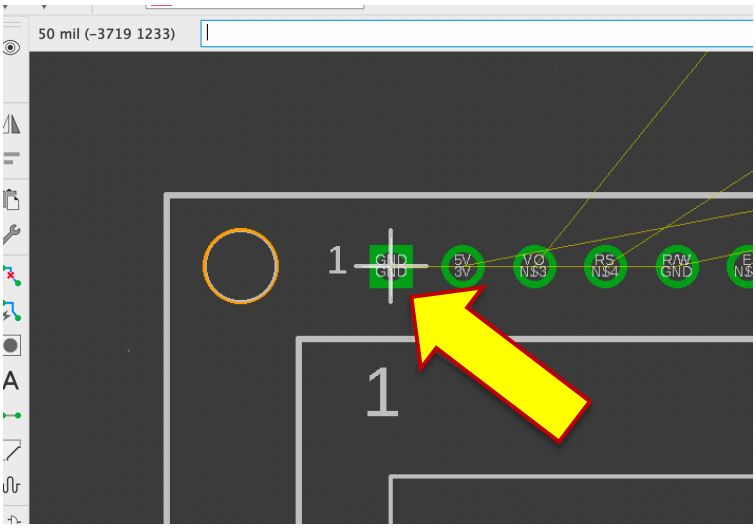
The SCH/BRD tile will let you switch from the schematic tool to the PCB tool.



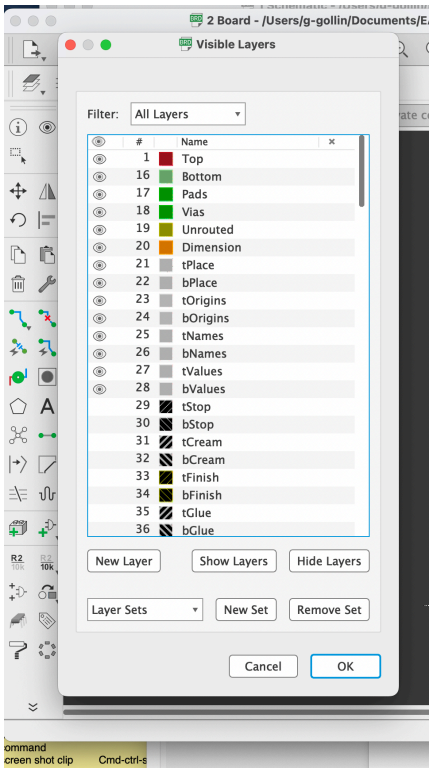
Here's what pops up.



Take note that each part will include an origin, which is the feature you use to select, move, delete, copy,… the part.

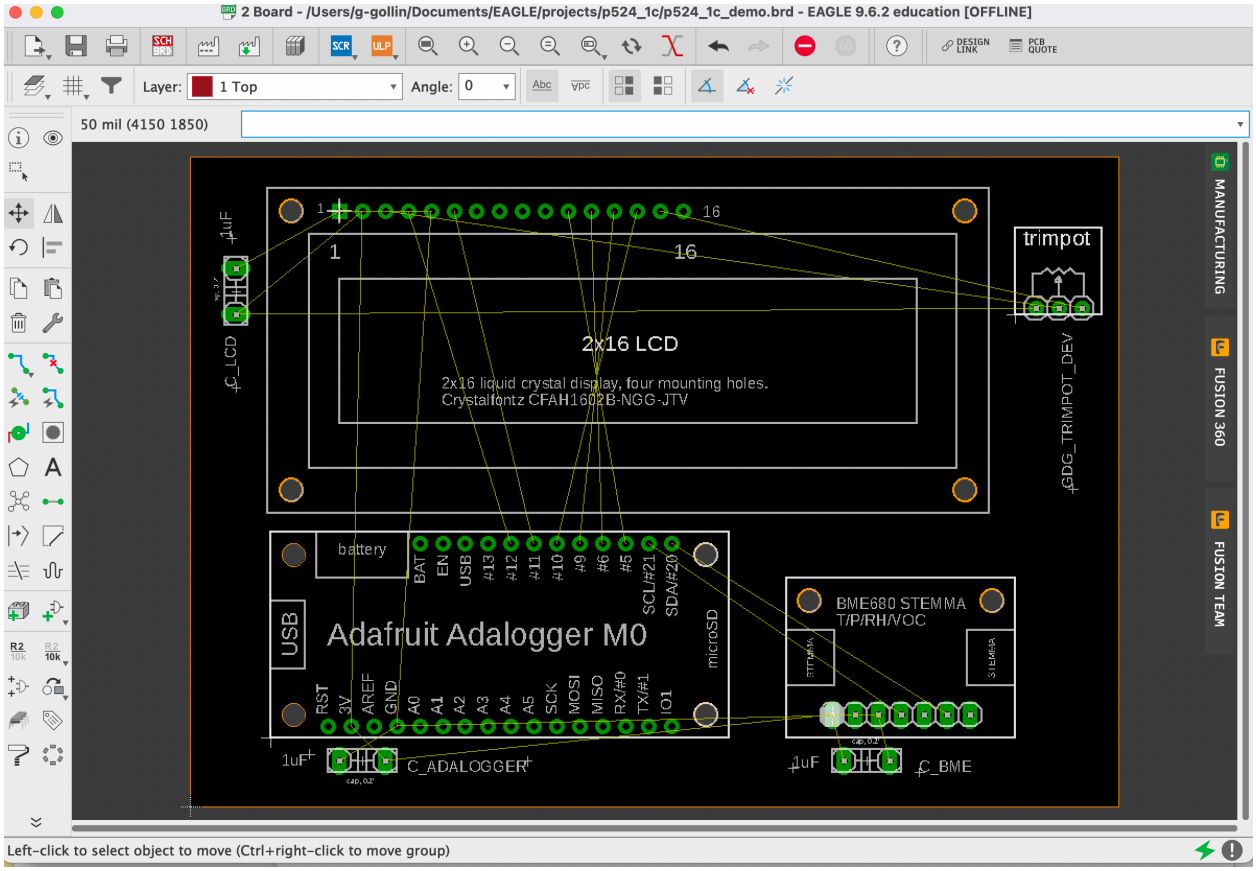


Take a look at the layers menu (see below). “Top” and “bottom” refer to the layers that’ll eventually receive copper traces.



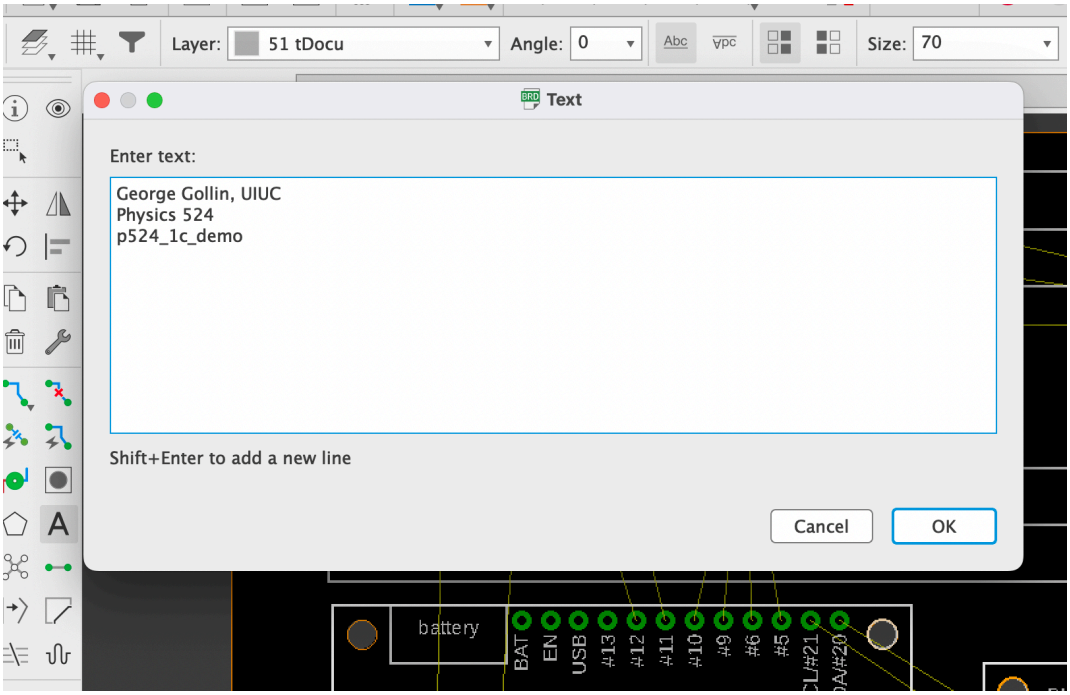
 ©George Gollin, 2024

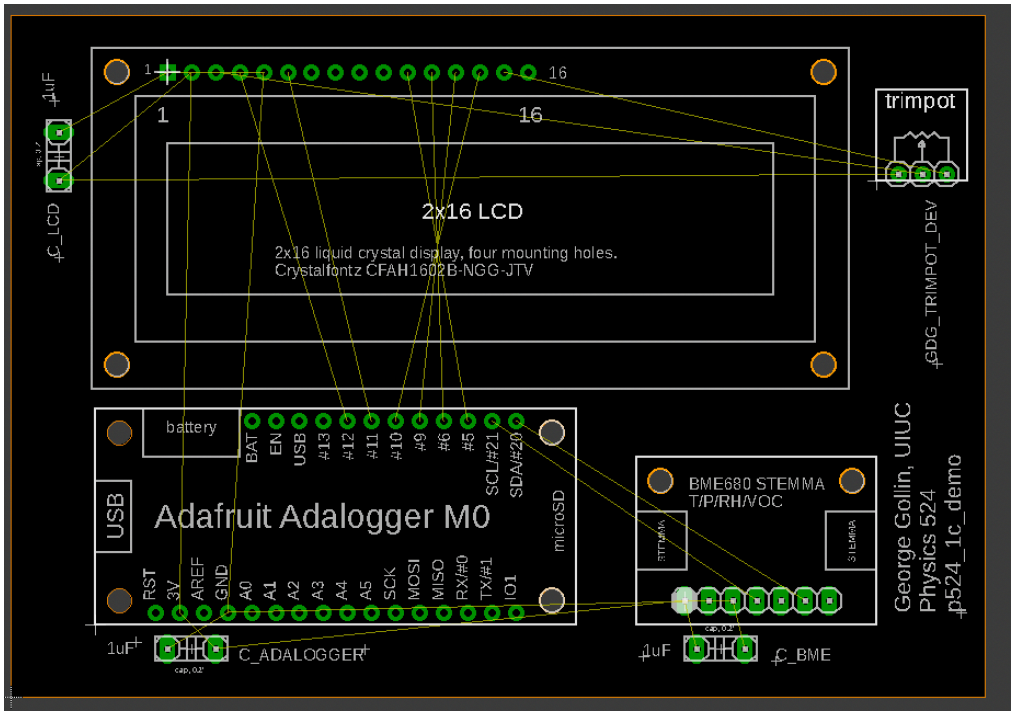
Move all parts into the dark area that is outlined in orange. You can rotate them as suits your aesthetic sense. Then shrink the orange-outlined area using the “move” tile to allow you to move the boundaries.



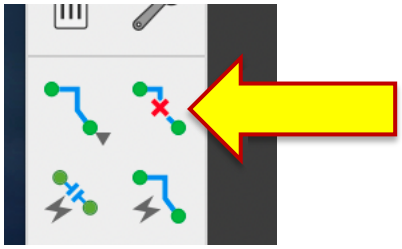
Add some text to the tdocu layer:

 ©George Gollin, 2024

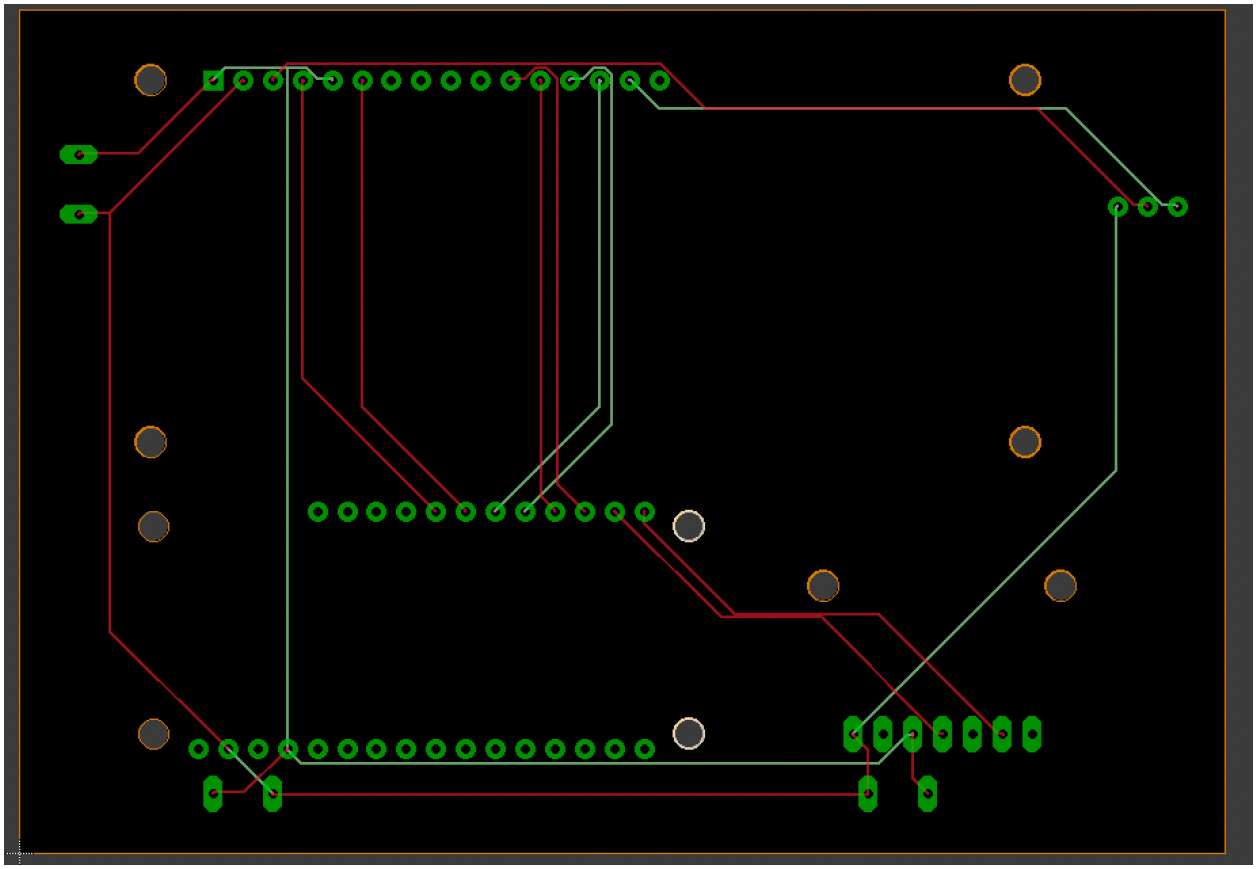




Now run the autorouter:

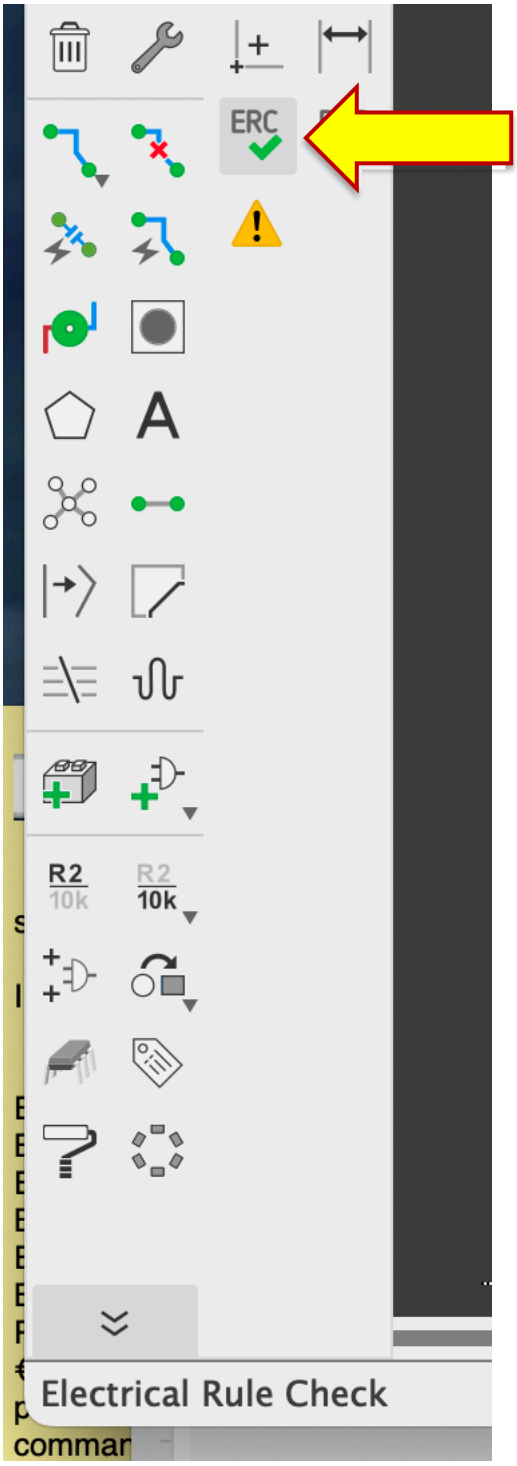


Here's what I get. I've turned off most layers.



When you're done, run the ERC ("electrical rules checker") and fix any problems that are found. After that, run the DRC (design rules checker). Again, fix problems that are found.

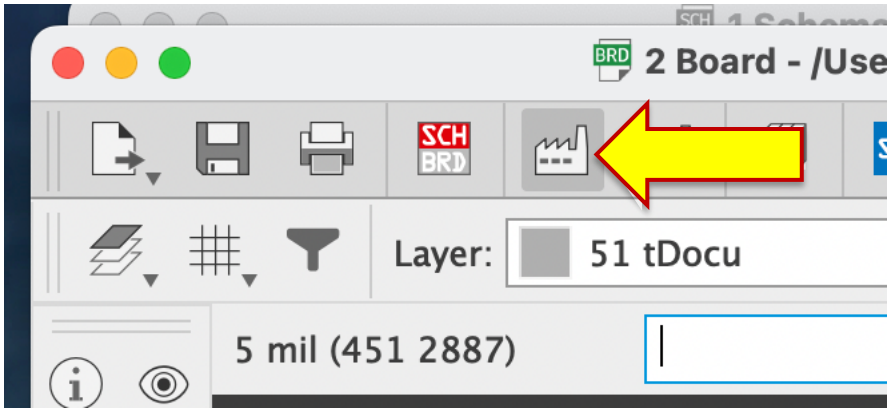
 ©George Gollin, 2024



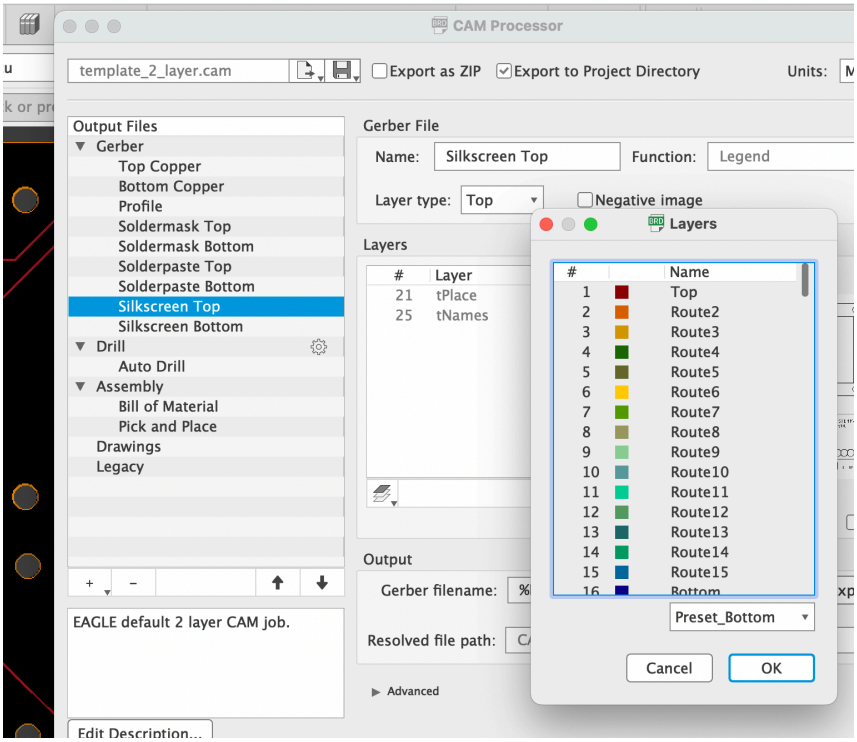
Production/fabrication files

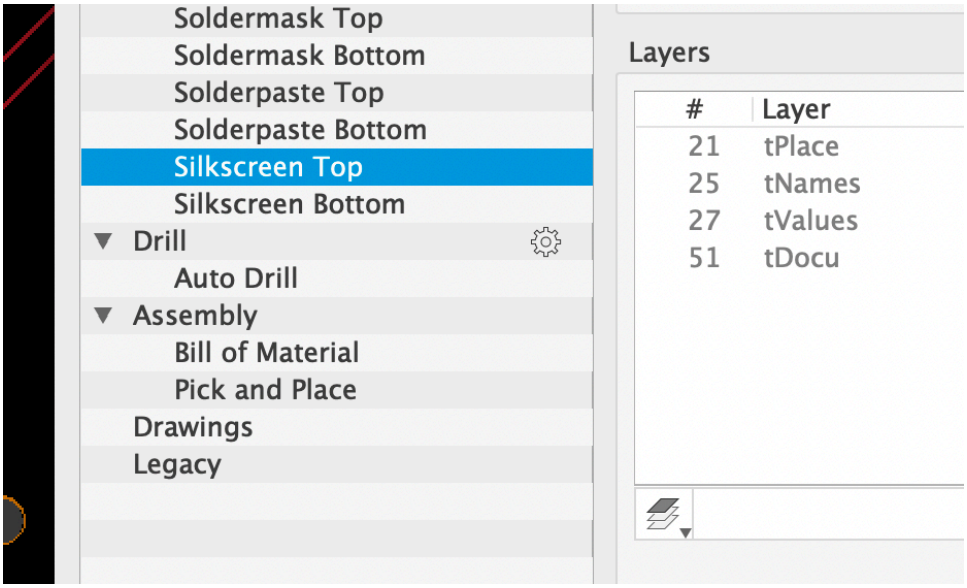
Here's the CAM processor tile:

 ©George Gollin, 2024



Click on it, then add tValues and tDocu to the top silkscreen layer:





Also add the bValues and bDocu layers to the Silkscreen Bottom layer, then click "Process Job."

You'll find that there is now a folder named CAMOutputs in your project folder.

That's it! This file can be zipped, then sent off to a fabricator like JLCPCB.

This week's homework assignment (due at the first class meeting next week)

Please design a circuit featuring an Adalogger, an LCD, a BME680, and a DS3231 real time clock. Lay out the PCB for your circuit, add some identifying text to it, and create the CAM files needed to fabricate it.

We will look at your design at the start of class.

a

 ©George Gollin, 2024