

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

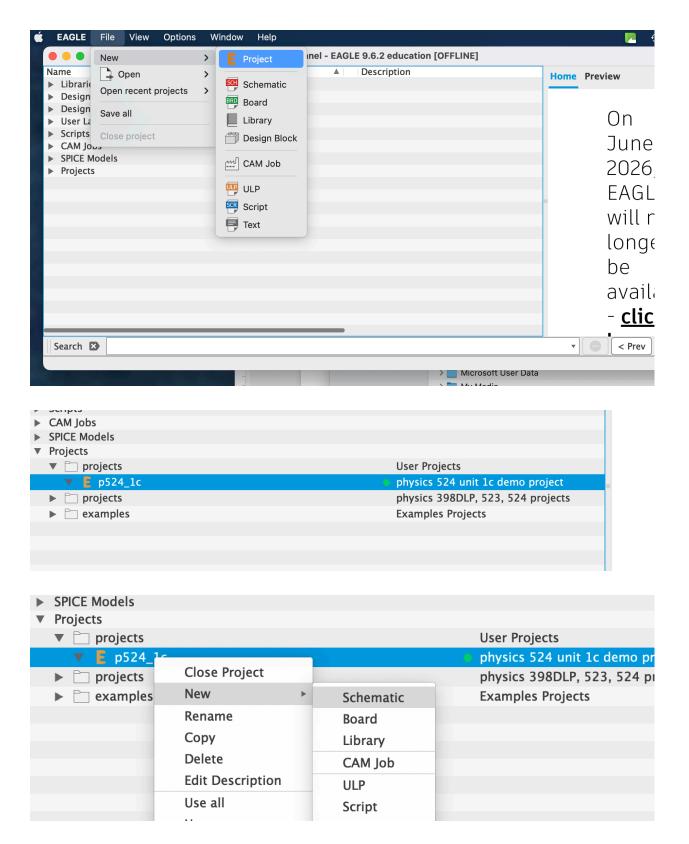
Visit <u>https://www.autodesk.com/education/free-software/eagle</u> and sign in. Follow the steps to download EAGLE, Autodesk's schematic capture and printed circuit board layout tool. There are good tutorials on how to install, then use EAGLE on the sparkfun.com web site if you'd like additional information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



I Schematic - /Users/g-gollin/Documents/EAGLE/projects/p524_1c/untitled.sch - EAGLE 9.6.2 education [OFFLINE]
$\mathscr{O}_{\text{LESSEN}} \equiv \texttt{SCB}_{\text{TE}} \models \texttt{L} \models \texttt{L} \models \texttt{L} = \texttt{LSS} 1/1 \bullet \texttt{M} \texttt{SCN} \models \texttt{L} \models \texttt{Q} \bullet $
一     授     ,      冊     、     工     Layer:     □     91 Nets     ○     □    □   □    □    □    □    □   □    □    □    □    □
(i) (i) ∩ ∩ (0.6 − 0.4)
Group objects
Move selected stuff
Copy
Change
Add part
→ (AB)
R2 R2 IDK IDK
Grid origin (0,0)
Add text
Left-click & drag to define group (or left-click to start defining a group polygon)

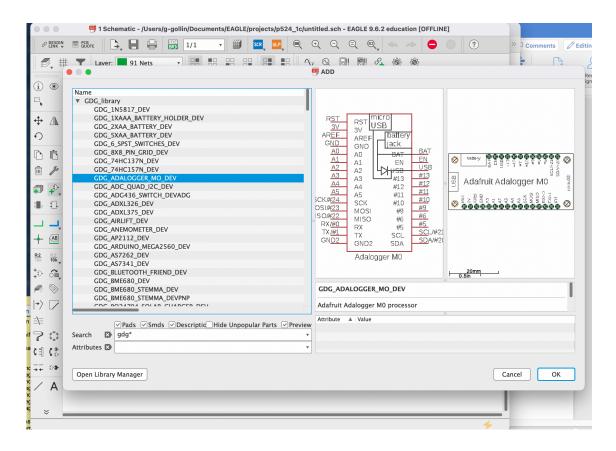
You'll get an empty window that looks something like this:

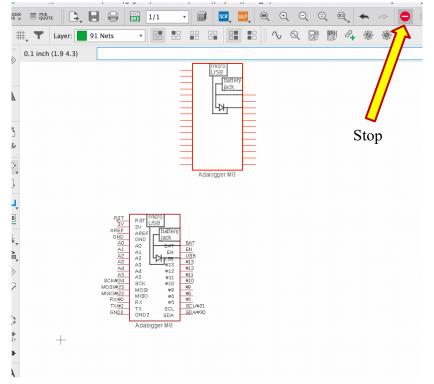
🗧 😑 🗧 🖾 🗐 1 Schematic - /Use	s/g-gollin/Documents/EAGLE/projects/p524_1c/untitled.sch - EAGLE 9.6.2 education [OFFLINE]
🖉 🚛 🗮 Layer: 🚺 91 Nets	
-0.6 -0.4)	
🔍 🔿 🗌 Informati	
° ↔ /\\ <u>+</u>  ↔	Zoom controls
	Active lavor Stop
r i	Active layer Stop
discar	
<b>₽ + .</b>	Switch between schematic and board views
<b>_</b>	
$\frac{R2}{10k} \frac{R2}{10k}$	
‡0- ã 	
$  \rightarrow \rangle [  ]$	
- A	
× ×	
Left-click & drag to define group (or left-	lick to start defining a group polygon) $\checkmark$

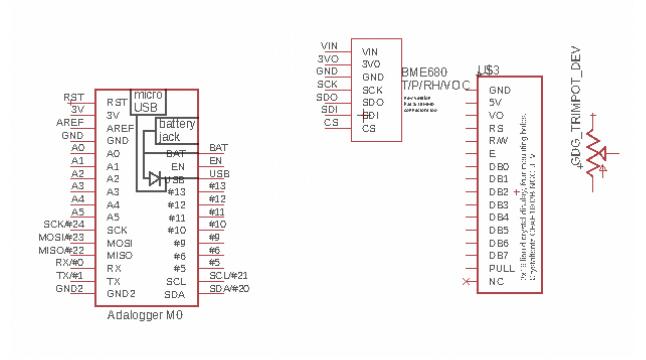
More about the window...

#### Add parts to the circuit

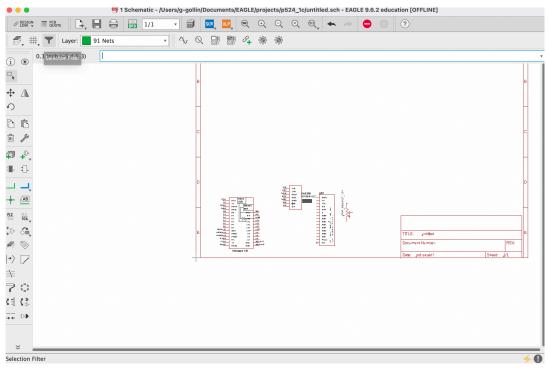
We want to use an Adalogger, a BME680, an LCD, and a 10k adjustable resistor (for setting LCD contrast). Click on the "add part" tile and look for GDG\_M0\_ADALOGGER\_DEV in the parts menu, then click OK. Use "stop" after adding a single Adalogger to the schematic. Add the other parts in similar fashion.



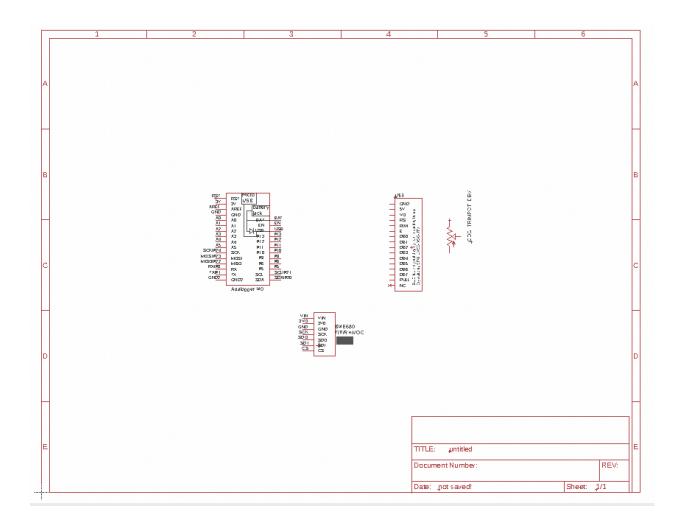




Now let's put a frame around the entire schematic, with the lower eft corner at the origin. In the add parts menu search for \*frame\* and select FRAME\_A\_L.



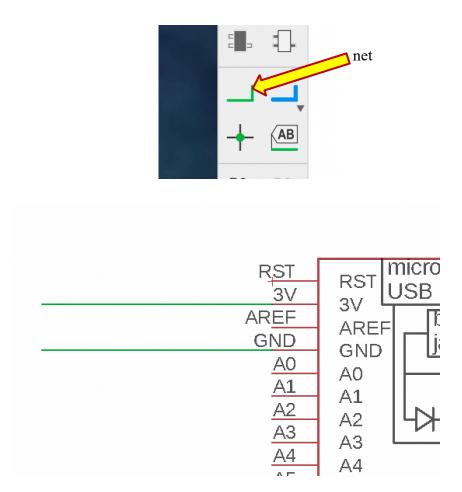
Use one of the Zoom controls to fill the window with everything we have placed so far. Then select the move stuff tile, click on the origins of various parts, and move them so that they are better centered in the frame. Oh—it's a good idea to do a "save all" from time to time.



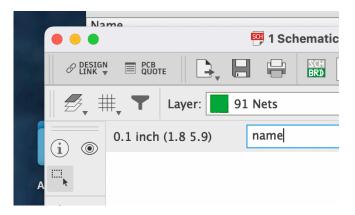
Now connect everything up! Let's wire the LCD and BME680 this way:

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	Е				
5	DB4				
6	DB5				
9	DB6				
10	DB7				

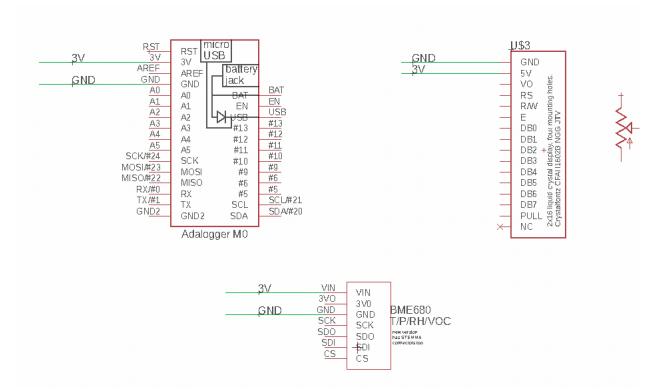
First use the net tile to draw nets from the Adalogger GND and 3V pins, bur don't bother to connect them to anything.



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.

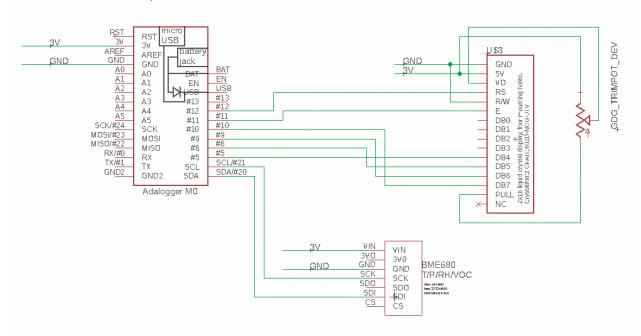


Do the same for the BME680 and LCD pins that are supposed to be connected to GND and 3V:

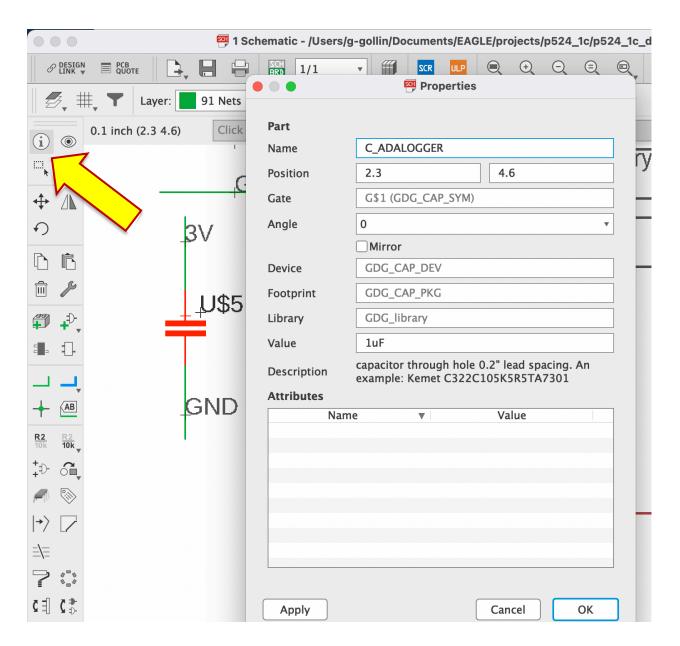


The schematic tool will know during PCB design that these pins need to be connected. It's entirely your choice—whether to label nets or to run the connections from pin to pin—and you should develop your schematic with an eye towards readability and clarity.

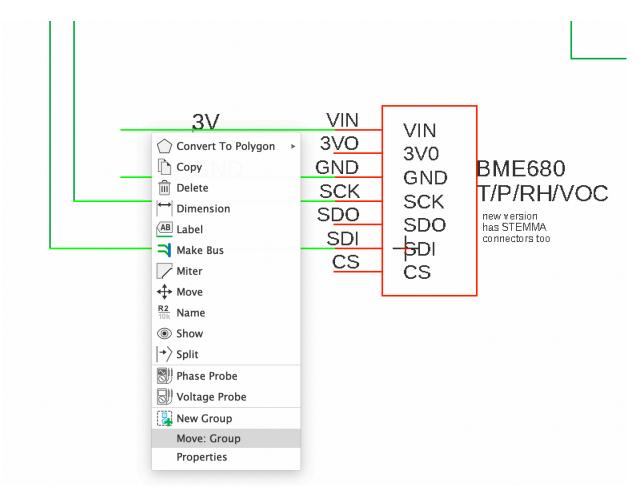
Here's how my version of the circuit looks.



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



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.



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.

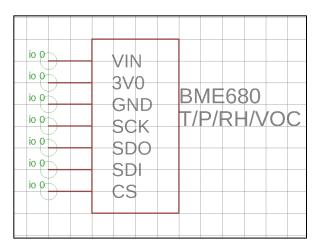
Did any of you play with Colorforms when you were little? It's a creative toy which includes a variety of thin vinyl shapes—circles, rectangles, and the like—that will adhere to the smooth, black surface that a child uses as a drafting environment.

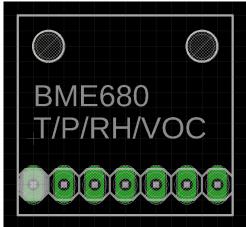


A schematic capture tool works in similar fashion: the designer selects parts from a library of components, places them on the design surface, and draws the interconnections between component pins.

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.

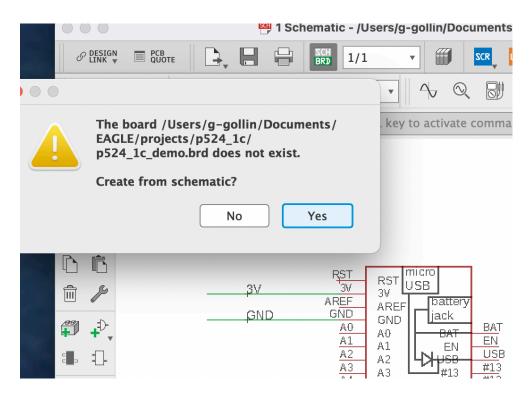
An example: here are the symbol and package (or footprint) for a BME680.



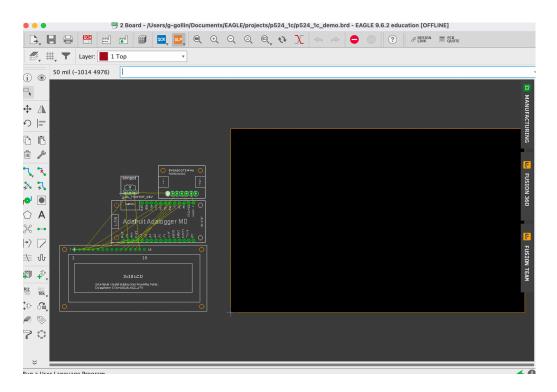


#### Board layout tool

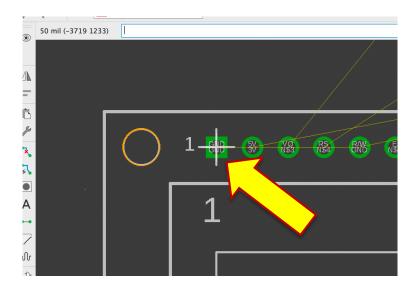
The SCH/BRD tile will let you switch from the schematic tool to the PCB tool.



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.

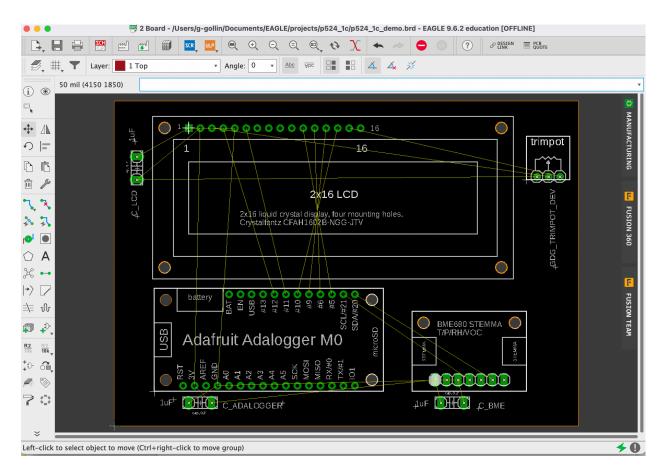


Take a look at the layers menu (see below). "Top" and "bottom" refer to the layers that'll eventually receive copper traces.

	ten/	👜 2 Board - /Users/g-gollin/Docu	ments/E
	•	🕮 Visible Layers	2 0
<b>_</b>			
	Filter: All La	yers 🔹	ate c
(i) 💿		IN	
Щ.	#	Name ×	
k	<ul><li>16</li></ul>	Top Bottom	
↔ /\	<ul> <li>10</li> <li>17</li> </ul>	Pads	
•	<ul> <li>17</li> <li>18</li> </ul>	Vias	
の  =	<ul> <li>10</li> <li>19</li> </ul>	Unrouted	
	<ul><li>20</li></ul>	Dimension	
D B	<ul> <li>20</li> <li>21</li> </ul>	tPlace	
		bPlace	
		tOrigins	
• •	24	bOrigins	
<b>N N</b>	25	tNames	
<i>4</i> , <i>1</i> ,	26	bNames	
		tValues	
		bValues	
$\triangle A$	-	tStop	
<u> </u>	30		
ж 🚥		tCream	
	-	bCream	
→ /	33	tFinish	
i)≣ ∿	34		
	35		
<b>₽</b> + <sup>0</sup> ,	36		
R2 10k 10k	New Layer	Show Layers Hide Layers	
‡D <b>∂</b>			
<i>🛋</i> 💿	Layer Sets	New Set     Remove Set	
7 :			
E 000		Cancel OK	
*			
~ =			
creen shot clip	Cmd-ctrl-s		
	and our c		

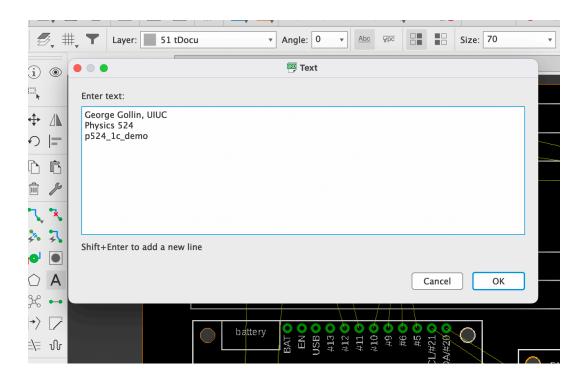
Unit 1c

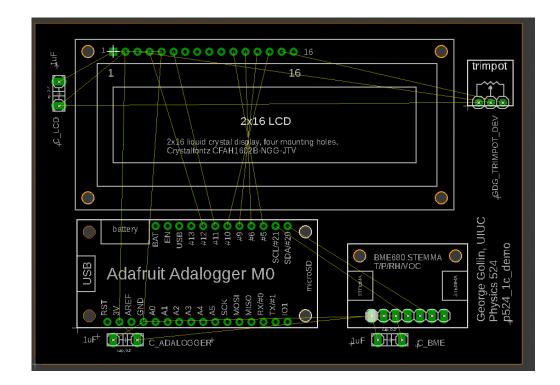
Move all parts into the dark area that is outlined in orange. You can rotate them as suits your aesthetic sense. Then shrink the orange-outlined area using the "move" tile to allow you to move the boundaries.



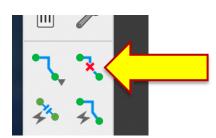
Add some text to the tdocu layer:

Unit 1c

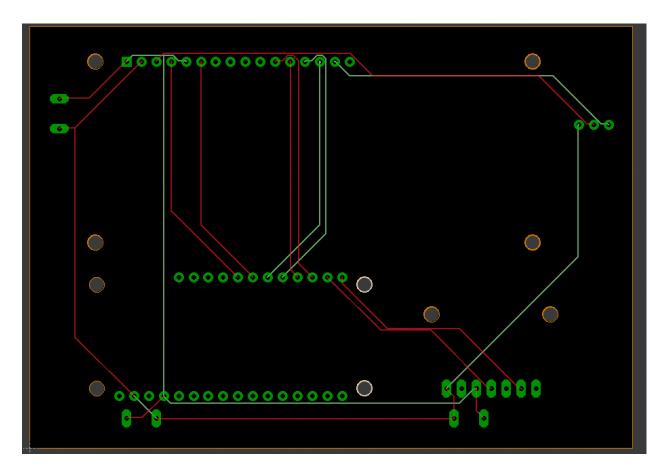




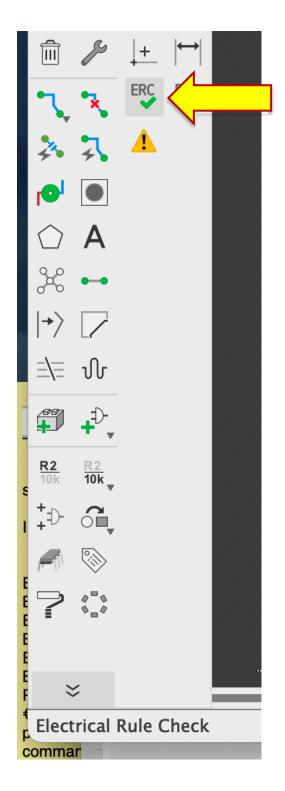
Now run the autorouter:



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

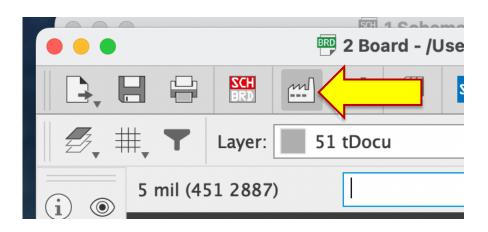


When you're done, run the ERC ("electrical rules checker") and fix any problems that are found. After that, run the DRC (design rules checker). Again, fix problems that are found.



# **Production/fabrication files**

Here's the CAM processor tile:



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

Ħ			🕮 CAM Pro	ocessor	H n	
u	template_2_layer.cam	Expor	t as ZIP 🕑	Export to Proje	ect Directory	Units: N
k or pro	Output Files	Gerber Fil	e			
	▼ Gerber		Cillian	. T	nu de la companya de	
	Top Copper	Name:	Silkscreer	птор	Function: Leger	ia
	Bottom Copper					
$\bigcirc$	Profile	Layer ty	pe: Top	▼ Ne	gative image	
	Soldermask Top			- • •	🕮 Layers	
	Soldermask Bottom	Layers				
-//	Solderpaste Top	#	Lauran	#	Name	
	Solderpaste Bottom	21	Layer tPlace	1	Тор	
	Silkscreen Top	21	tNames	2	Route2	
	Silkscreen Bottom	23	tivames	3	Route3	
	▼ Drill	3		4	Route4	
	Auto Drill			5	Route5	
	Assembly			6	Route6	
	Bill of Material			7	Route0	
	Pick and Place			8	Route8	518.975 114
	Drawings			9	Route9	
	Legacy			10	Route10	200
$\frown$		<b>5</b> ,		11	Route10	
$\cup$		2/		12	Route11	
				13	Route12	C
_				14	Route15	
()		Output		15	Route15	
	_ ÷ _ − ↑ _ ↓	Gerber	filename:	% 16	Rottom	хр
			L		Preset_Botto	
	EAGLE default 2 layer CAM job.	Deschued	file path:	C	Treset_botte	
		Resolved	me path:	C		
					Cancel	ОК
		▶ Advance	d			
$\cap$	Edit Description					

Soldermask Top Soldermask Bottom		Layers	
Solderpaste Top		#	Lover
Solderpaste Bottom		#	Layer
Silkscreen Top		21	tPlace
Silkscreen Bottom		25	tNames
▼ Drill	£03-	27	
Auto Drill	~~~	51	tDocu
Assembly			
Bill of Material			
Pick and Place			
Drawings			
Legacy			
		27	

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.

