**Title:** Fall 2025 Camp, Printed Circuit Board Design

**Please use 319K ed discussion for questions, PCB Camp category**
**Instructor:** Jonathan Valvano valvano@mail.utexas.edu

**Class:** watch YouTube videos and ask questions in zoom office hours
Youtube video playlist

**Office Hours:** See ECE319K Syllabus Page on Canvas for zoom link

The plan for fall 2025 is to run camp asynchronously during the week of 10/20 to 10/24. You will finish assignment 1 alone, form groups 2-5 students, and then **finish assignment 2 by 10/27/2025**, get PCBs back before doing Labs 8 and 9. If you want me to buy the PCB, assignment 2 must be in groups of 2-5. Don't finish assignment 2, and then look for a group; that is no fun for the others. If you want to work alone, you can simply buy the PCB yourself (I will still review and answer questions)

**OH:** see my door, EER5.820 (next to Dr. Y), ask for questions

**How to find partners:**
-A good way to find PCB partners is in lab, before class, or after class.

**Disclosure:** There will be no student cost, no face to face student contact, and no grades. This camp is completely optional. It will not affect your regular ECE319K/ECE319H grade.

**Description:** Printed circuit board (PCB) :design process; tools for PCB layout and fabrication; standardization; design for testability. Make a PCB similar this this

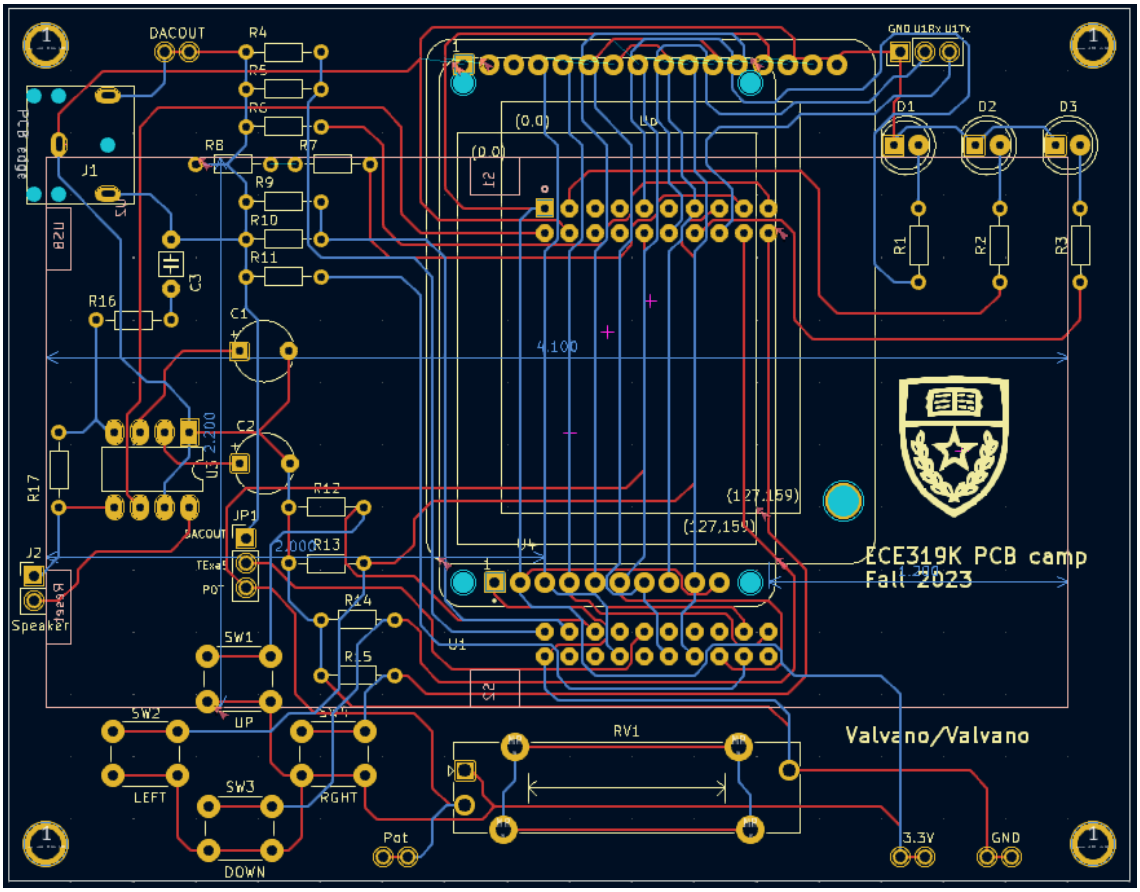



*Figure 1. PCB in KiCad showing a typical ECE319K project (from Fall 2023).*

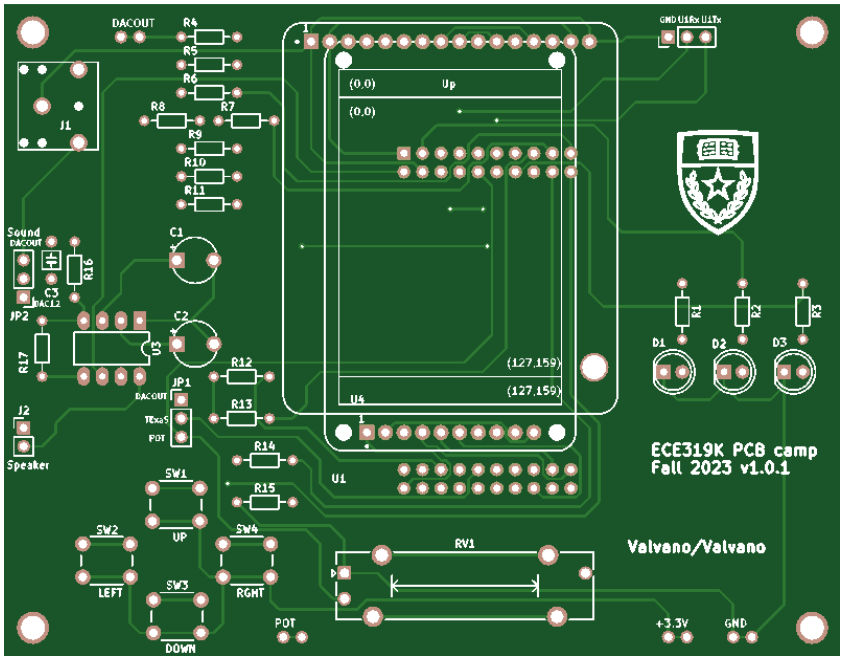



*Figure 2. PCB manufactured by JLCPCB (from Fall 2023).*

**Overview:** This hands-on course will walk through the process of PCB design, layout, construction, and test. Each team of two to five students will layout an ECE319K/ECE319H PCB that contains all the circuits required to complete Labs 5, 6, 7, 8, and 9. The circuit design and software will be presented as part of regular ECE319K/ECE319H, but in this camp, students will manufacture the interconnections as a printed circuit board. Students will translate the electrical circuit into a PCB design, where most of their parts are selected from a starter file containing the parts already in your ECE319K/ECE319H lab kit. Students will learn to consider issues like drill holes, high current routing, standard values, tolerances, and testing. Students draw the electrical circuits in schematic form as the first assignment, performed individually. If you complete assignment 1, teams of 2 to 5 will be formed and the team will create one PCB design. Teams of 2 to 5 students who complete their PCB layout by **Monday 10/27/2025 8pm** will have their board manufactured, making five copies (Valvano will pay for it). There are two 20-pin headers and some hardware that will be distributed with the PCBs. Since all students will create similar circuits, students will evaluate each other's layout. It takes about 2 weeks to get PCB boards produced, so the PCBs should be back in time for you to use it in Labs 8, and 9. To use the PCB for ECE319K/ECE319H you will need to solder the kit components onto the PCB.

**Signup:** (no actual commitment)
https://docs.google.com/spreadsheets/d/1JlcZKAabRoVn5QLbOWWUosIQ9vHJUVuZPtwNsVPdiMY/edit?usp=sharing

**Equipment:** Students must have a laptop that can run KiCad.

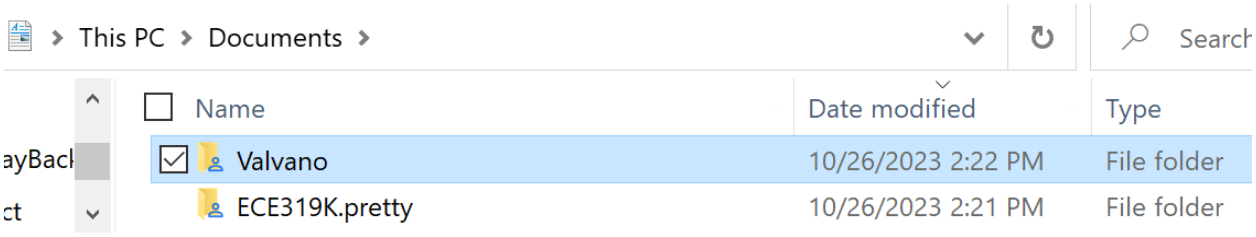
**Pre-assignment 1) Install KiCad:**
Install latest version of KiCad, 9.0 or later, (do not spend any money); it was tested with v9.04; it should work with 9.0.5 (current version, 10/2025)
Download and install **https://www.kicad.org/**

***Warning*: do not open files in KiCad 8**

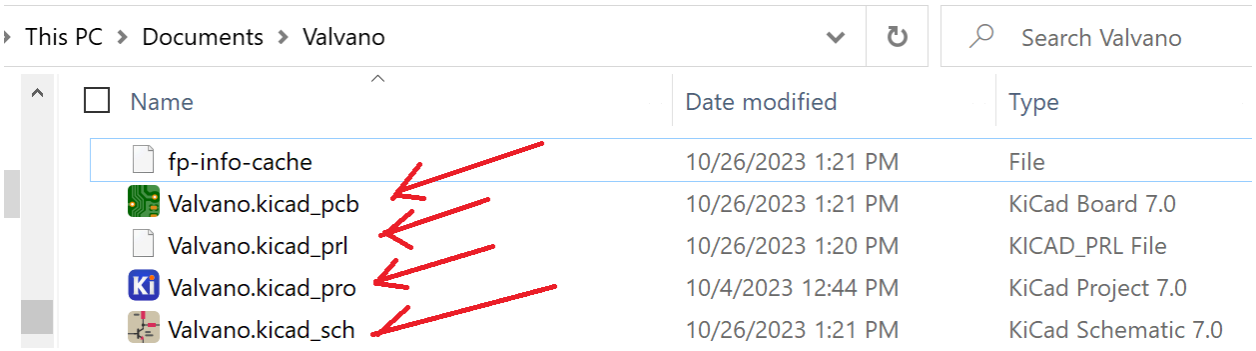
**Bookmark this site, which has tutorials and videos:**
https://www.kicad.org/help/learning-resources/

**Download this helpful hotkey definitions**
https://drive.google.com/file/d/18eobHFZlUDPsCQhepW_lLWPqnnkM5umo/view?usp=sharing
*Learn these hotkeys to greatly improve efficiency*

**Pre-assignment 2) Download the PCB camp Starter files**
1) Download the entire **Starter** folder from
https://drive.google.com/drive/folders/1Cxj-Af-QSPSRLyT7VdW92KhDO7RoeUtv?usp=sharing
renaming the folder to your EID
2) Copy the **ECE319K.pretty** folder onto your computer (these are footprints).
https://drive.google.com/drive/folders/1J925VqsIuFCgUggCJW_CLpIT0CpJX5xF?usp=sharing

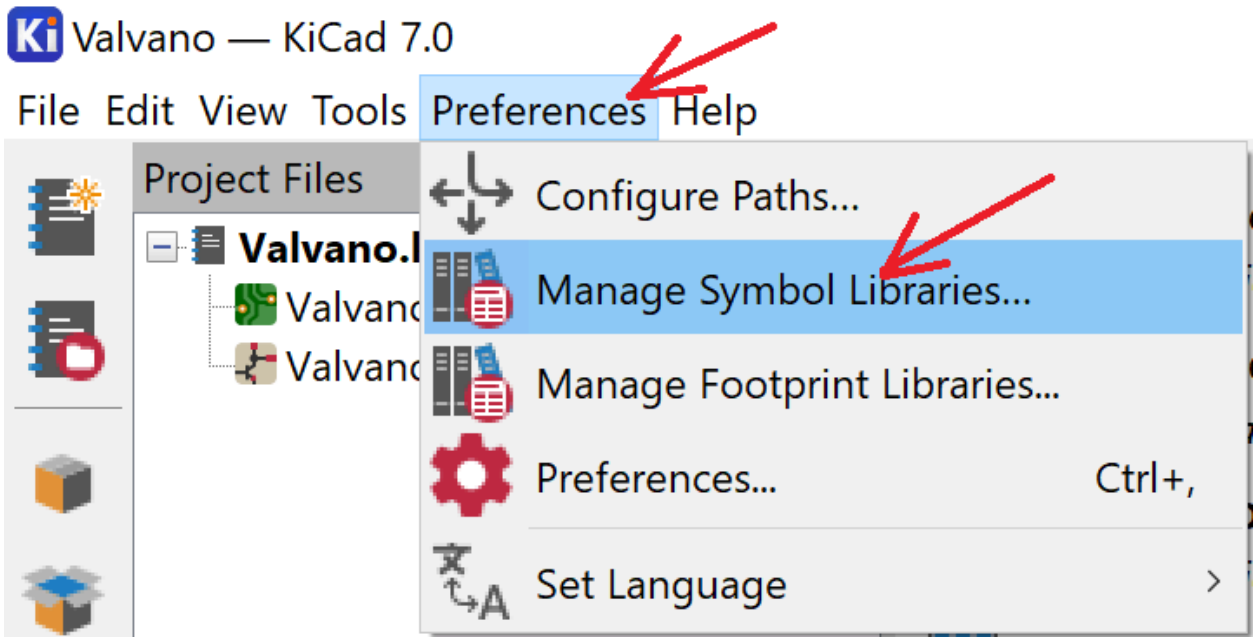



3) Copy the **ECE319K.kicad_sym** file onto your computer (these are symbols).
https://drive.google.com/file/d/1PTcwJrMAq-hL1xInbpiC1G8xil8dOrC7/view?usp=sharing
4) Rename "Starter" to "YourEID" in all four file names

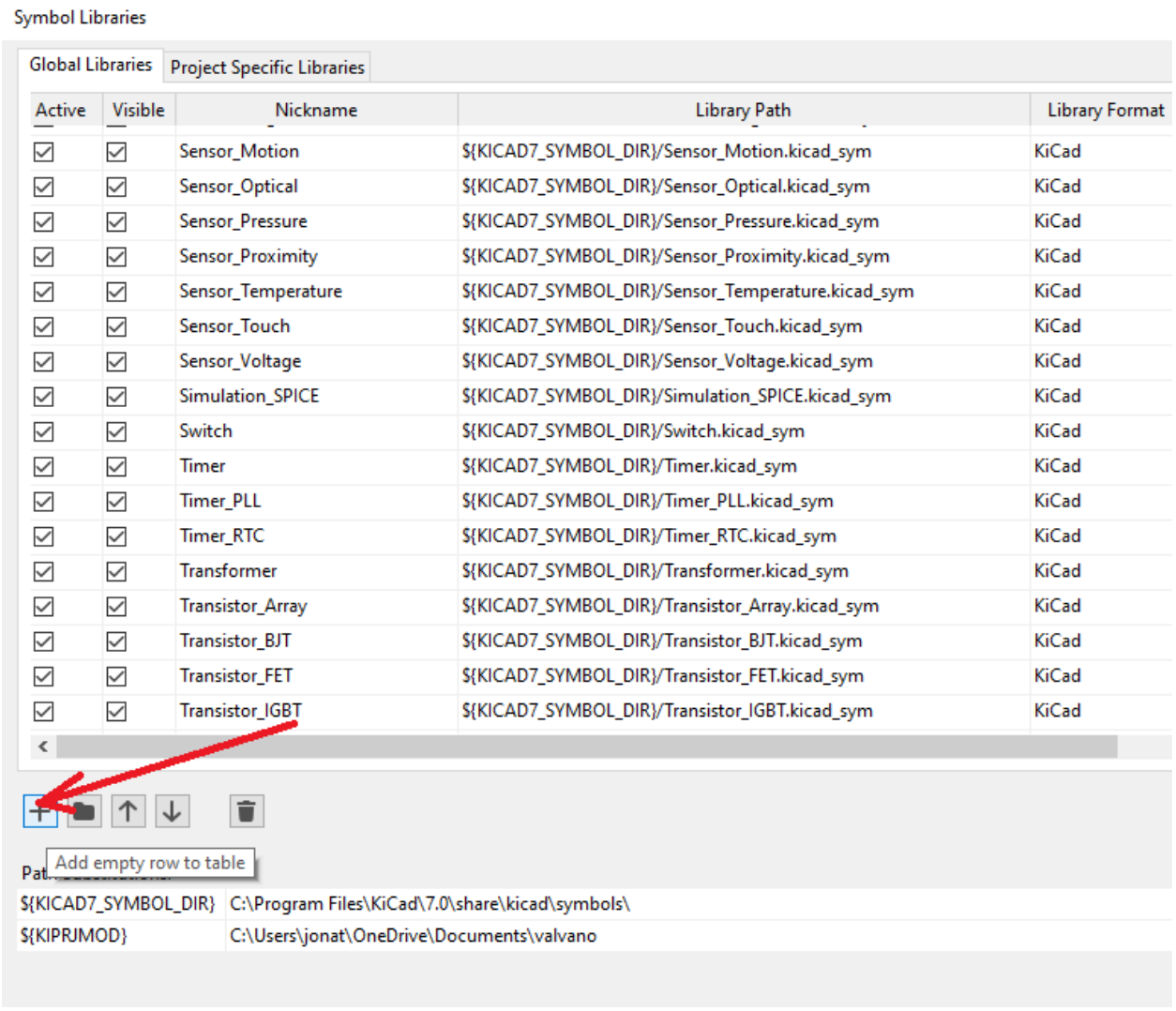



**Video** Preassignment 2) Downloading and arranging starter files

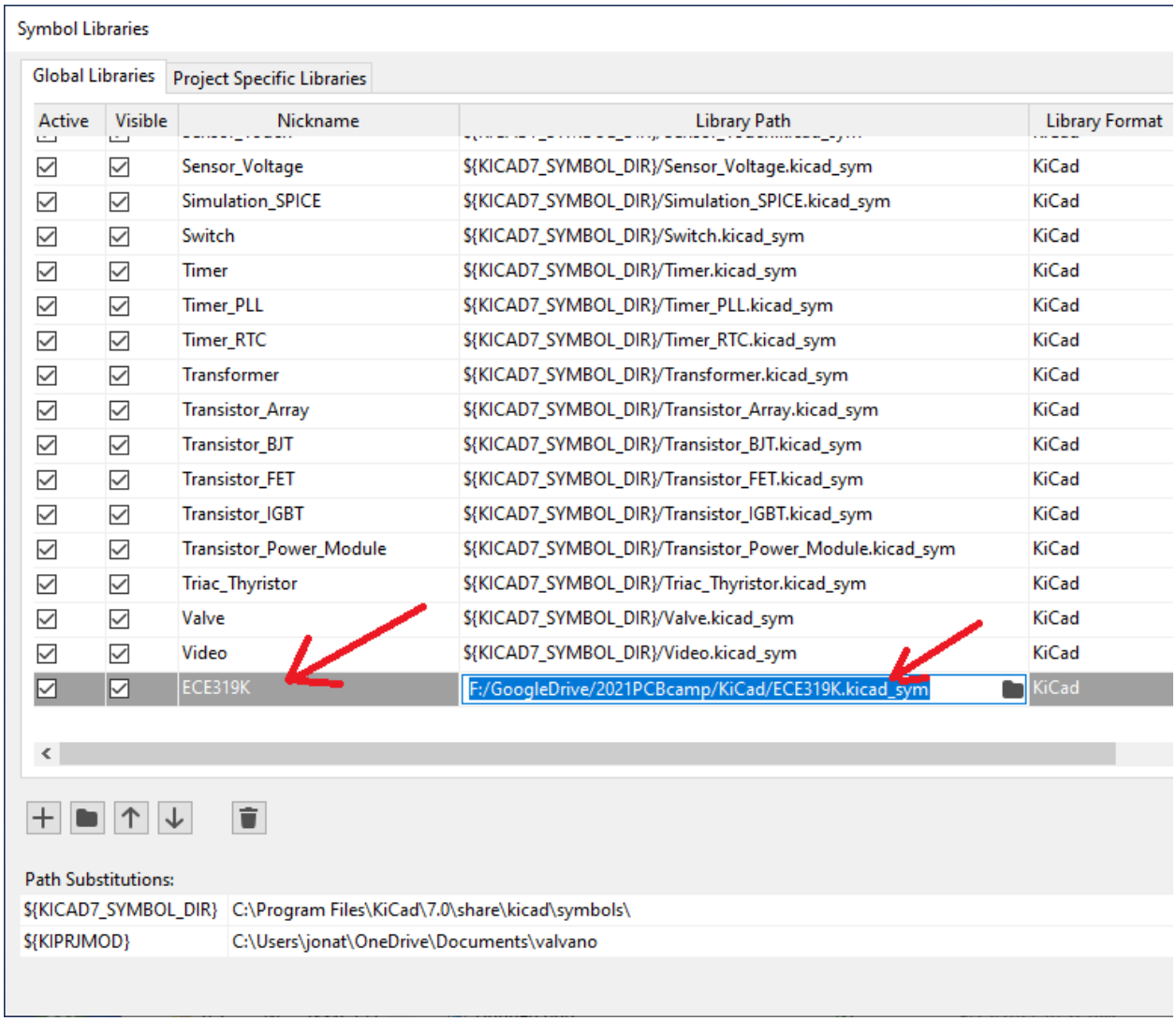
Link the **ECE319K.kicad_sym** symbol library into KiCad version 9.0.5
**Video** Attaching libraries to KiCad



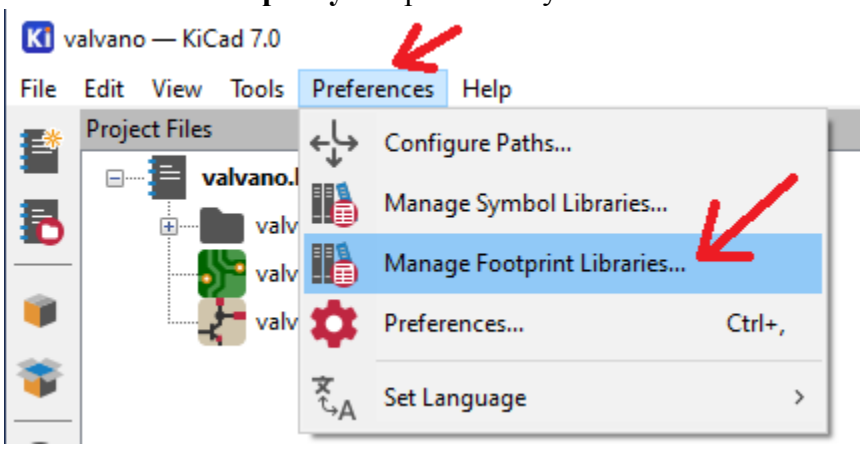

Click **Add empty row to table**



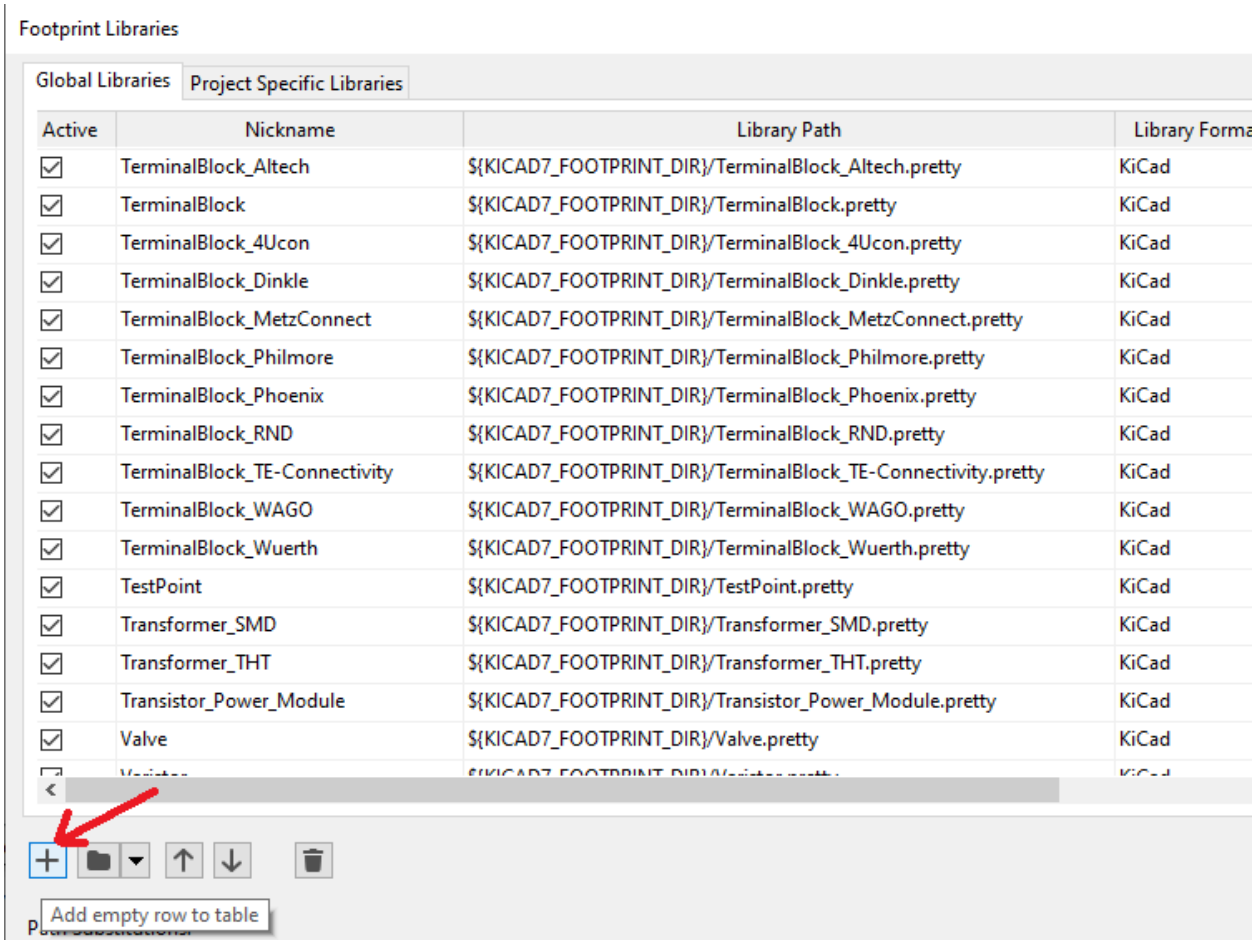

Specify the nickname to **ECE319K** and the path to the **ECE319K.kicad_sym** file



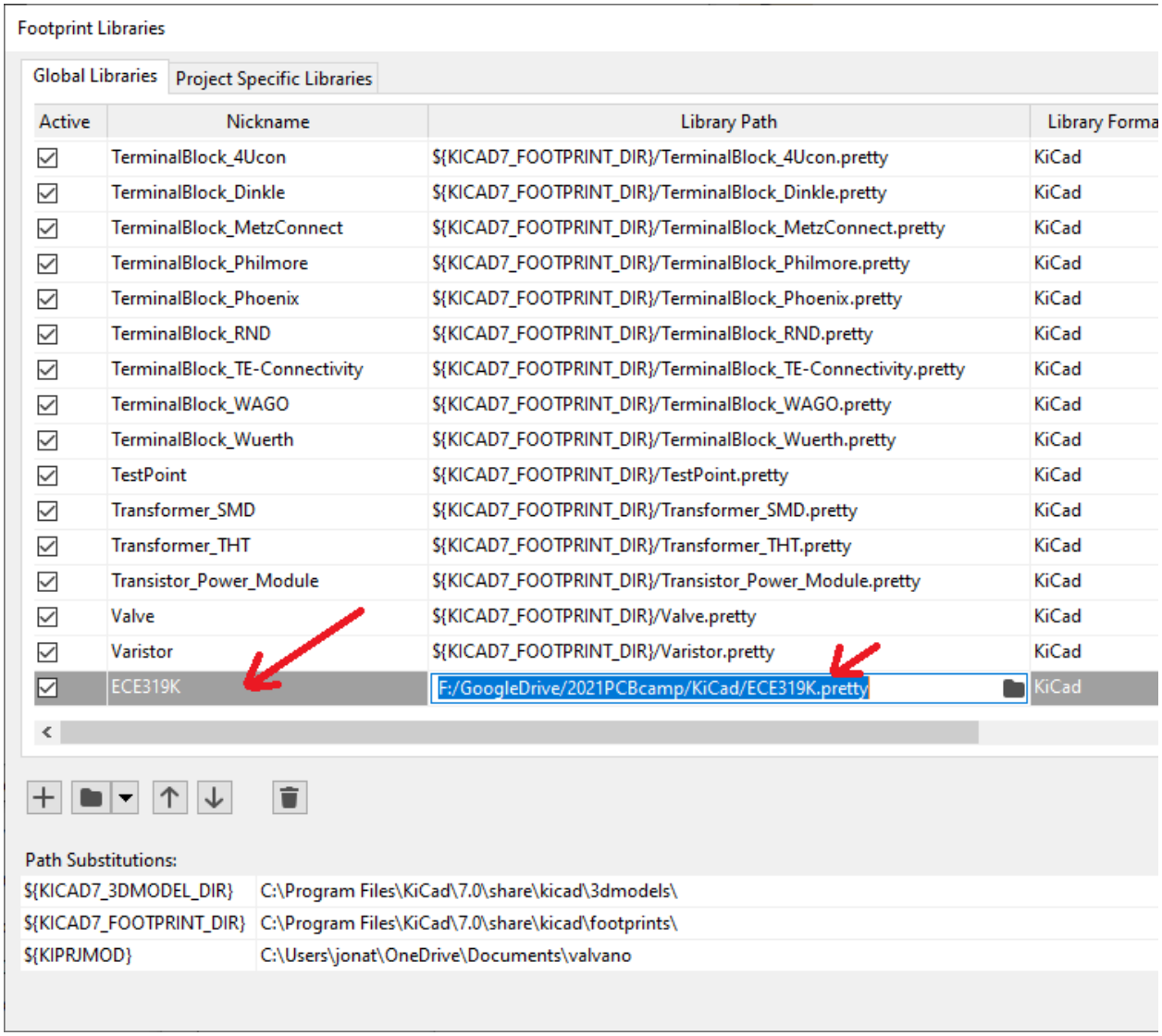

Link the **ECE319K.pretty** footprint library into KiCad 9.0.5




Click **Add empty row to table**




Specify the nickname to ECE319K and the path to the **ECE319K.pretty** folder




**Pre-assignment 3) Open your project YourEID.kicad_pro in KiCad**
Make sure you can see both the schematic (sch) and PCB (pcb) views. You should be able to switch back and forth without quitting and restarting KiCad.

**Pre-assignment 4) Think about Labs 6, 7, 8, and 9**
Think about all the hardware you will need for Labs 6-9. Most students use the same interface circuits, so it will be easy to merge 2-5 students into one PCB design later during assignment 2.

You do not turn in Pre-assignment 4. Simply decide:
- how many switches do you want?
- how many LEDs do you want?
- which LCD you are doing?
- are you in ECE319K or ECE319H? (ECE319H needs TSOP31438, IR LED and ULN2003B)
- do you want the headphone jack and speaker or just the speaker?
- do you want the audio amplifier (MC34119)

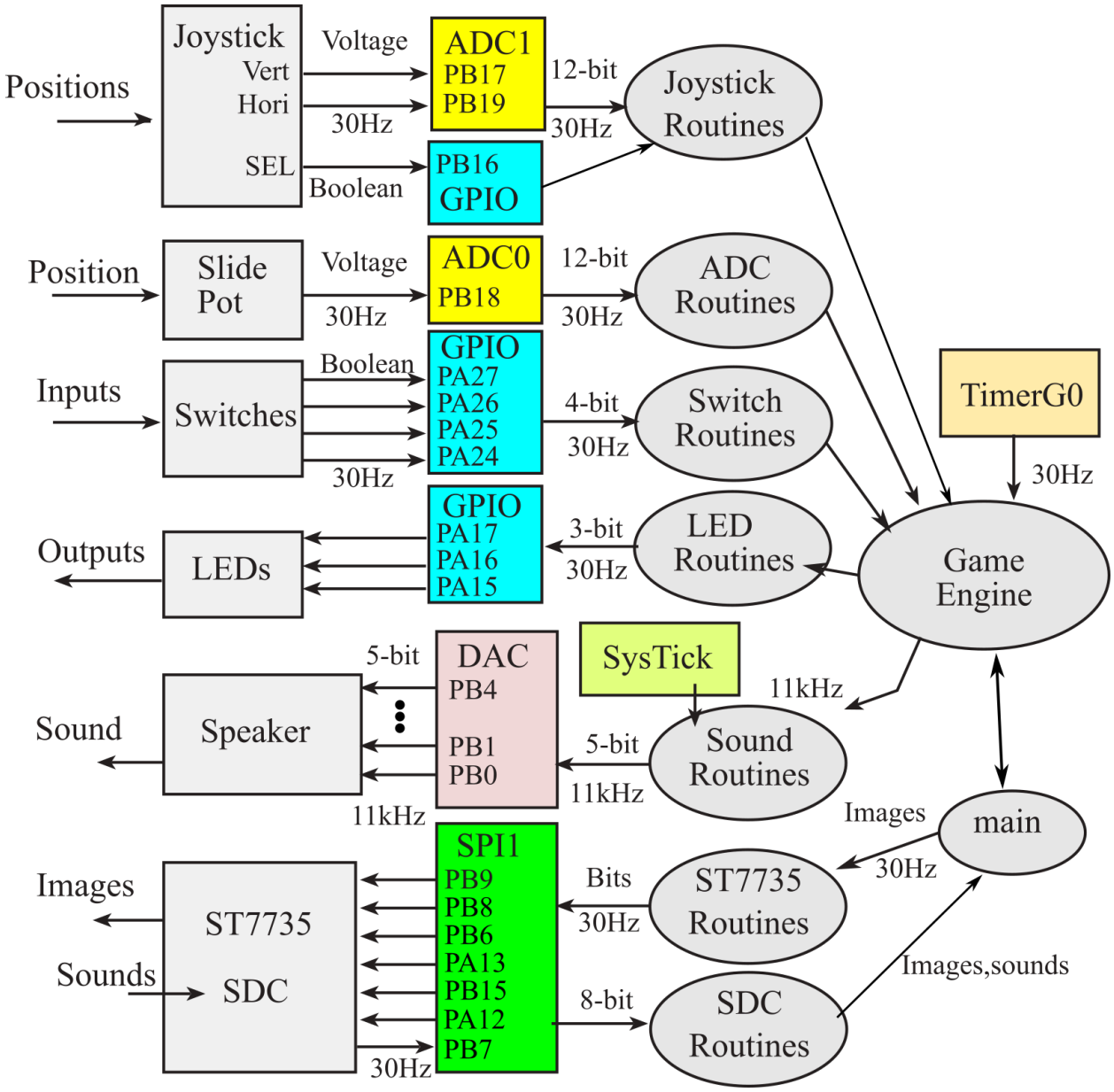
- do you want to have a joystick (which you have to buy)?
- you are allowed to use the 12-bit DAC, just also include the 5-bit DAC (but don't connect DACs together

Decide which pins to use. These pins are already used
- PA10, PA11 UART to PC (runs Terminal window)
- PA22 UART2, PA8 UART1 to other microcontroller Lab 8
- PB6, PB7, PB8, PB9, PB15, PA13 SPI/GPIO to LCD
- PB18 ADC to slidepot
- PB0, PB1, PB2, PB3, PB4 to 5-bit DAC
- PA15 12-bit DAC output
- PA18, PB21 to LaunchPad switches
- PA0, PB22, PB26, PB27 to LaunchPad LEDs
-

**Video** Thinking about Labs 6,7,8,9

**Notice, the most common mistake students make is to connect the same microcontroller pin to two different and unrelatied I/O devices*** In other words, use each pin for at most one function.

The following data flow graph shows one possible pin assignment for Lab 9




Step 1) Expand your Lab 2 circuits to have four switches and three LEDs. You are welcome to add additional LEDs to any unused pins.

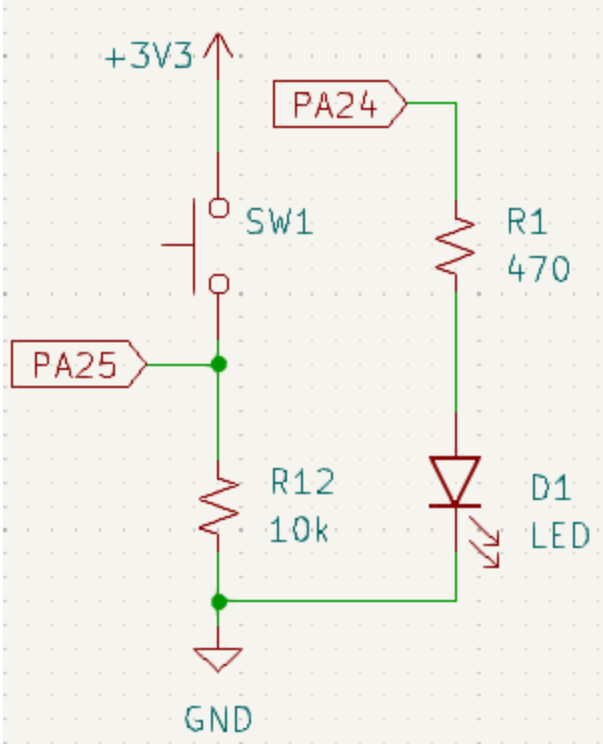



*Figure 2. KiCad schematic drawing showing one switch, and one LED.*

Step 2) Plan your 5-bit DAC using five 1.5k and five 12k resistors. The default connection is the 150-ohm speaker, but you could put both the speaker and headphone jack connections.

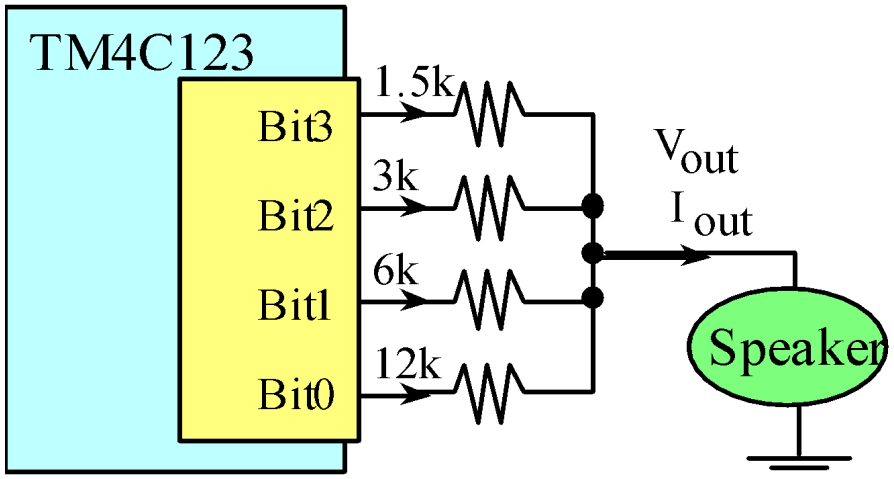



*Figure 3. Four bit binary weighted DAC. You will make a 5 bit DAC, just like Lab 5.*

Step 3) The starter project has both the Adafruit and HiLetGo LCDs interfaced to the microcontroller. The **CARD_CS** signal could be connected to a GPIO to allow you to use the SD card feature on the ST7735 in future projects. If you connect CARD_CS, your software must set it to output=3.3V, so the SDC is off. Even though this is the third semester using KiCad, please double check the LCD interface making sure it matches the connections from Lab 6.

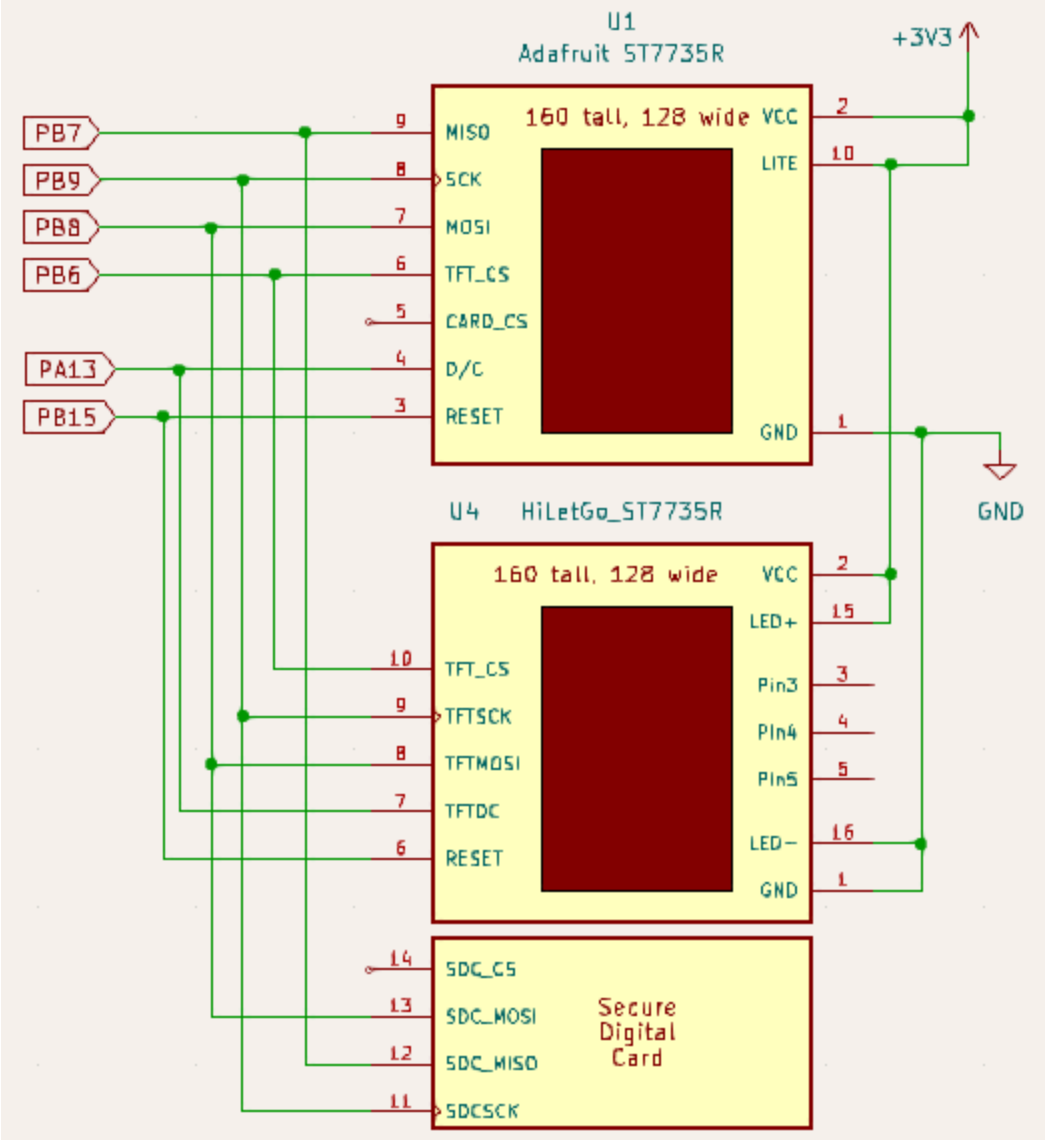



*Figure 4. Interface connections for the ST7735R LCD.*

For now leave both LCDs in the schematic. If everyone in your PCB group has the same type, you can remove the other one to simplify routing during Assignment 2.

Step 5) Connect the Slidepot to the microcontroller as described in Lab 7. The 2-pin testpoint allows you to solder a U-shaped solid wire (clipped off wire from a resistor), to which you can connect a voltmeter or scope during testing.

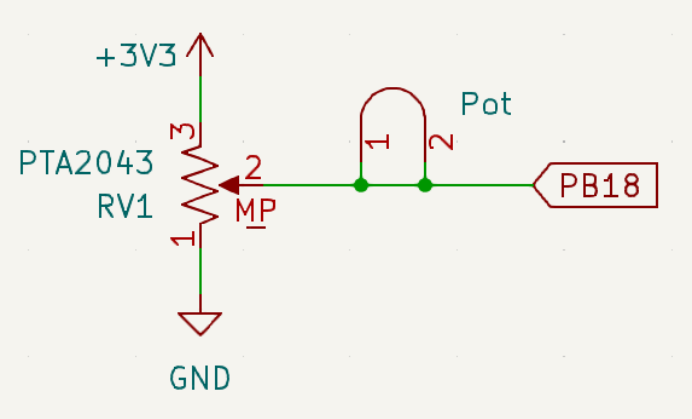



*Figure 5. Possible circuit to interface the sensor.*

Step 5) To use the TExaS oscilloscope, we connect an analog signal to PB20. A 3-pin jumper allows the operator to connect PB20 to either DACOUT or to PB18 (slide pot).

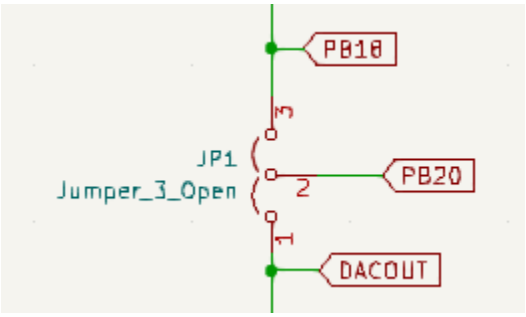



*Figure 6. Placing the jumper between PB20 and PB18 connects the slide pot analog voltage to the TExaS scope. Placing the jumper between PB20 and DACOUT connects the DAC analog voltage to the TExaS scope. Removing the jumper altogether allows you to connect the PD3 pin to any analog input.*

Step 6) ECE319K Lab 8 uses PA8 (U1Tx) and PA22 (U2Rx) to implement serial communication with another system. Add a 3-pin shorting jumper, like J3 in Figure 8. To simplify Lab 8, we will connect PA8, PA22, GND to this 3-pin header. We can then place a shorting jumper across PA8 to PA22 to debug Lab 8 (loop back mode). The normal solution to Lab 8 uses two boards with a 3-pin cable connecting PA8-PA22, PA22-PA8, and GND-GND.

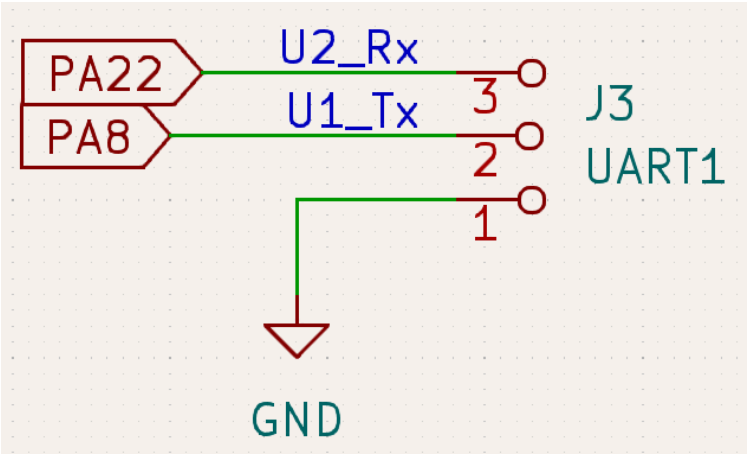



*Figure 7. Placing the jumper between PA8 (U1Tx) and PA22 (U2Rx) allows you to test Lab 8 with one board.*

**Step 6b)** ECE319H Lab 8 uses an IR LED output and IR receiver input. For the IR LED output, you need an LED interface that can handle 20mA current. The IR LED (OED-EL-1L2) requires about 10 to 20 mA, 1.2V to activate. Therefore, you will interface the IR LED to the PA8 output pin using a ULN2003B, a 220 ohm resistor and the +5V supply. I = (5-1.2-0.5)/220 = 15 mA.

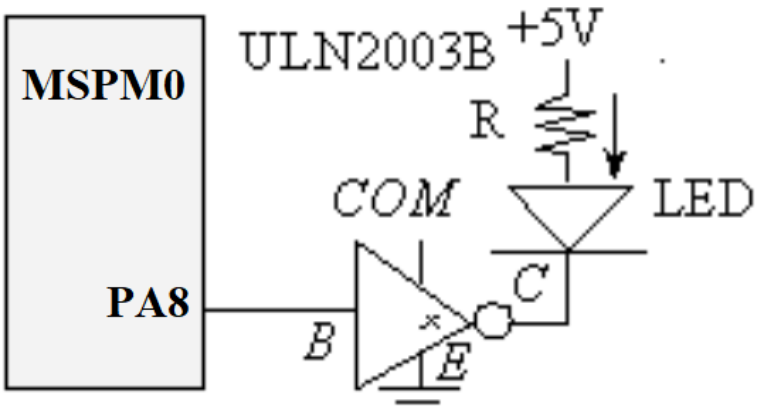



*Figure 8. IR LED interface for ECE319H. COM is floating*

For the TSOP31438 receiver has three pins with 0.1in spacing, with 0.1in between pins 1 and 2, and 0.2in between pins 2 and 3. The red arrow is the IR sensor. Position the TSOP31438 so the sensor points away from PCB. IR light received from spot shown with red arrow.

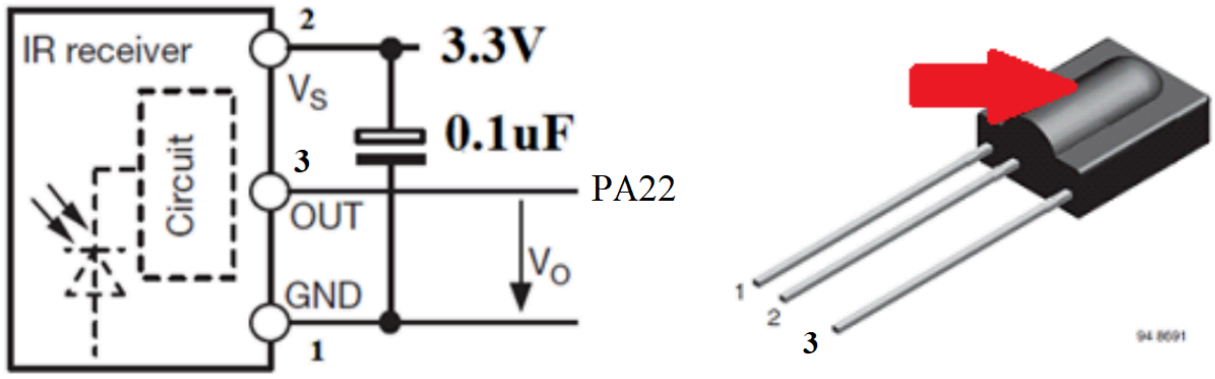



*Figure 9. TSOP31438 IR sensor interface for ECE319H. +Vs is 3.3V, $C_1$ is 0.1 uF, and OUT is connected to PA22 on the receiver. GND is pin 1, Vs is pin 2, OUT is pin 3.*

**Assignment 1) Draw the circuits for Labs 6,7,8,9 using KiCad**
Please complete assignment 1 before moving on to assignment 2). There is a google drive directory into which you will upload your files for review or submission. Please do not edit/change files from other students. You will find the link to this directory in the announcement on Canvas.
**Video** Assignment 1

**Valvano file review:**
Place three Assignment1 files (**.kicad_pro**, **.kicad_pcb** and **.kicad_sch**) in a separate folder, with the folder name matching your EID, in this googledoc
https://drive.google.com/drive/folders/1v8xCaj9u7tnt4OzHcpSUaDIHQcR3Y7S_

and Valvano will review, look for a docx file with my comments. I will not edit your SCH files (you have the master). When review is complete, I will make a comment in the associated docx file. You may look at designs from other students, but please do not edit/delete files that are not yours. After review, Valvano will move entire folder to
https://drive.google.com/drive/folders/1l6FRspJJTwW1pT89TTHGMSgrnHIXXxLz

Step 1) Build switch interfaces: 2 is minimum, 4 is awesome

Step 2) Build LED interfaces: choose how many LEDs you want

Step 3) Build DAC/speaker interfaces with Testpoint

Step 4) Build slide pot interface with Testpoint, one way to create testpoints is to add a two-pin jumper, connect both pins to the signal. On the PCB we will solder a 'U-shaped' solid wire (clipped off resistor wire) into both holes to create a place to which we can clip the scope or voltmeter.

Step 5) Look at sch and pcb to make sure the SCH is perfect. At this step you will

**Check** each net to make sure it is correct

**Check** all parts to make sure they are correct

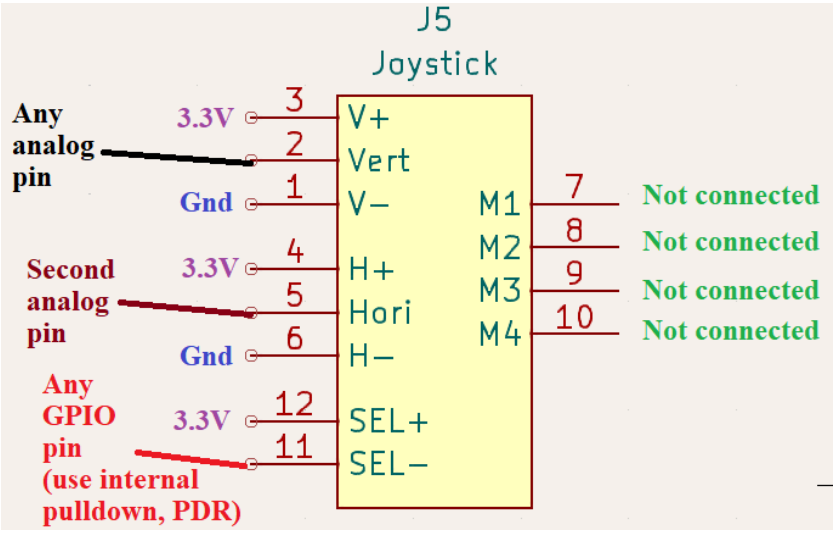
**Observe** the PCB to make sure all the parts are correct

*Optional stuff*: Sparkfun https://www.sparkfun.com/products/9032

Buy a joystick and connect it to two analog inputs; connect its switch to a GPIO pin

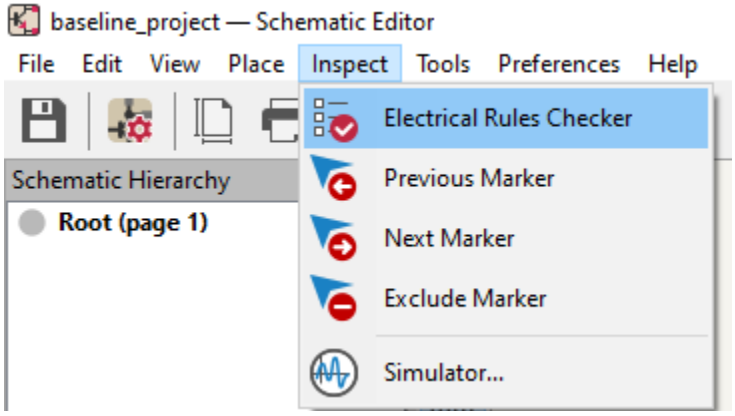
The Sparkfun #9032 Joystick part is in the ECE319K library

Wire up the two potentiometers like two copies of Lab 7. SEL+ and SEL- are a simple switch. Connect **Vert** and **Hori** to two analog inputs on the same ADC (i.e., both on ADC0 or both on ADC1)

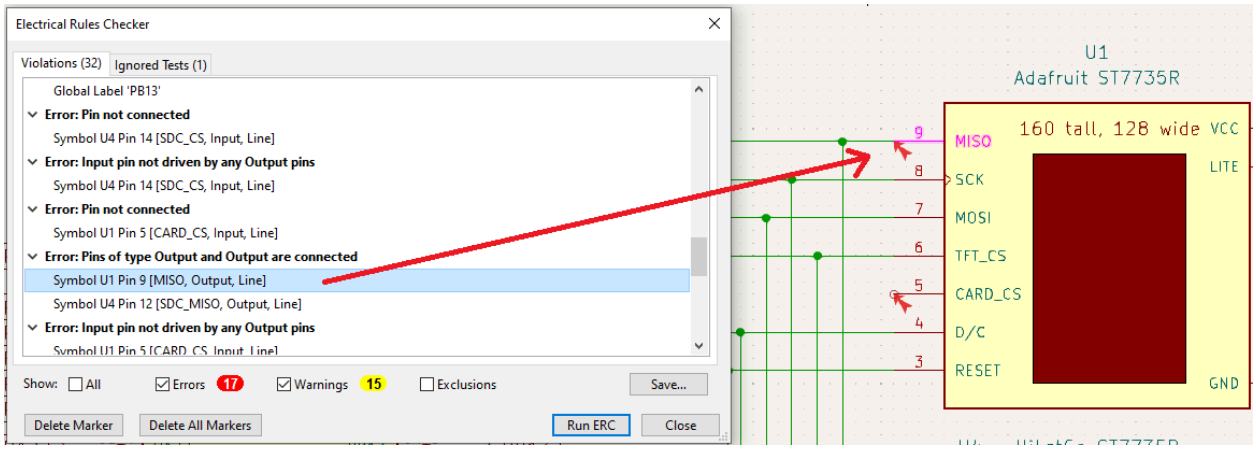



Look at the **ADC_InitDual** function in **ADC.c** project already on your computer to see how to sample two analog channels

Step 6) Run an ERC check before submission




Click on the errors one by one to see if it is a real error or something to be ignored




In the above figure, this  was flagged as an error because two outputs are connected. This one can be ignored because we will not solder both Adafruit and HiLetgo modules onto the same PCB. You can ignore

- Two MISO outputs connected together (like above Figure)
- errors for unused unconnected pins
- RESET pin is not driven
- Two GND pins are connected on the LaunchPad
- 3.3V power from the LaunchPad is connected to other 3.3V power
- The H1 H2 H3 H4 pins are not connected (these are mounting holes for 4-40 screws)

**Assignment 2) Place the components on the PCB file**

If you want to have Valvano purchase the PCB, you will perform assignment 2 in groups of two to five students. Please do not start assignment 2 until you have formed your group of 2-5.  If you wish to purchase your own PCB (about $27), you can perform assignment 2 in groups of any size. Enter your choice on this googleDoc

Option 1) I wish to form a group of 2 to 5 students and these are the names of the other students. Valvano will purchase the PCB.

Option 2) I wish to form a group of 2 to 5 students and invite others to join the team.. Valvano will purchase the PCB.

Option 3) I wish to form a group on my own. My group will purchase our own PCB.

https://docs.google.com/spreadsheets/d/1JlcZKAabRoVn5QLbOWWUosIQ9vHJUVuZPtwNsVPdiMY/edit#gid=1957258355

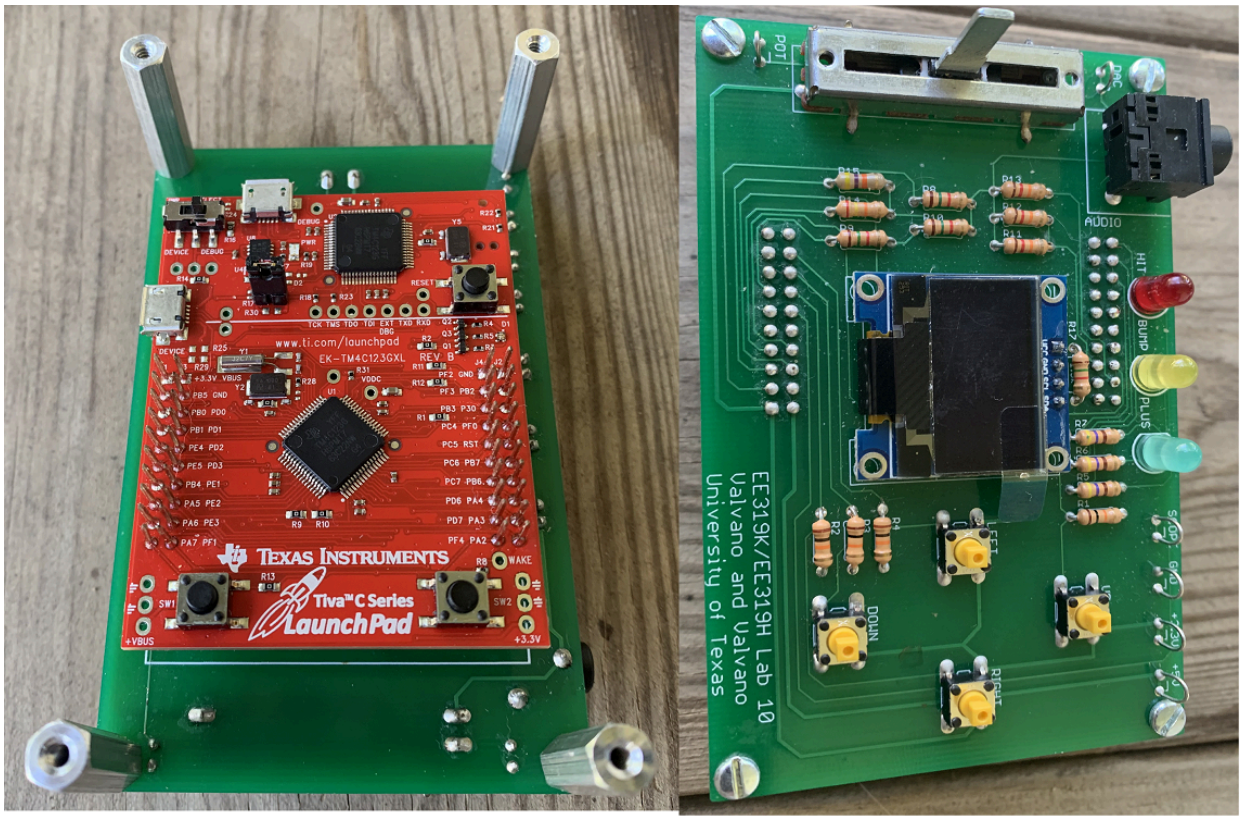



*Figure 10. Example system from Spring 2021.*

Step 0) Translate the schematic to the PCB
[Video Translating to the PCB]

PCB has colors, representing different aspects of the PCB. Figure 11, shows some of the colors we need to understand. **Copper** layers will conduct electricity. The PCB has copper on the pads and on the traces. **Silk** is the white objects we see on the PCB, used to simplify soldering and system operation. The **mask** layer over copper means it will have a tin layer, allowing us to solder to the PCB. You will notice most copper traces do not have a mask, so we cannot solder to traces. However, pads will have mask, so we can solder to the pads. The edge cut determines the size of the PCB. For PCB camp, I request you do not increase the size of the PCB, because the cost of the PCB is determined by the size. Similarly, PCB camp will be limited to two layers. The **courtyard** layers define the physical size of the components, and we should not overlap courtyards. The **fab** layers are comments to assist us when laying out the board. The fab layers will not be visible on the PCB.
[Video Understanding PCB colors]

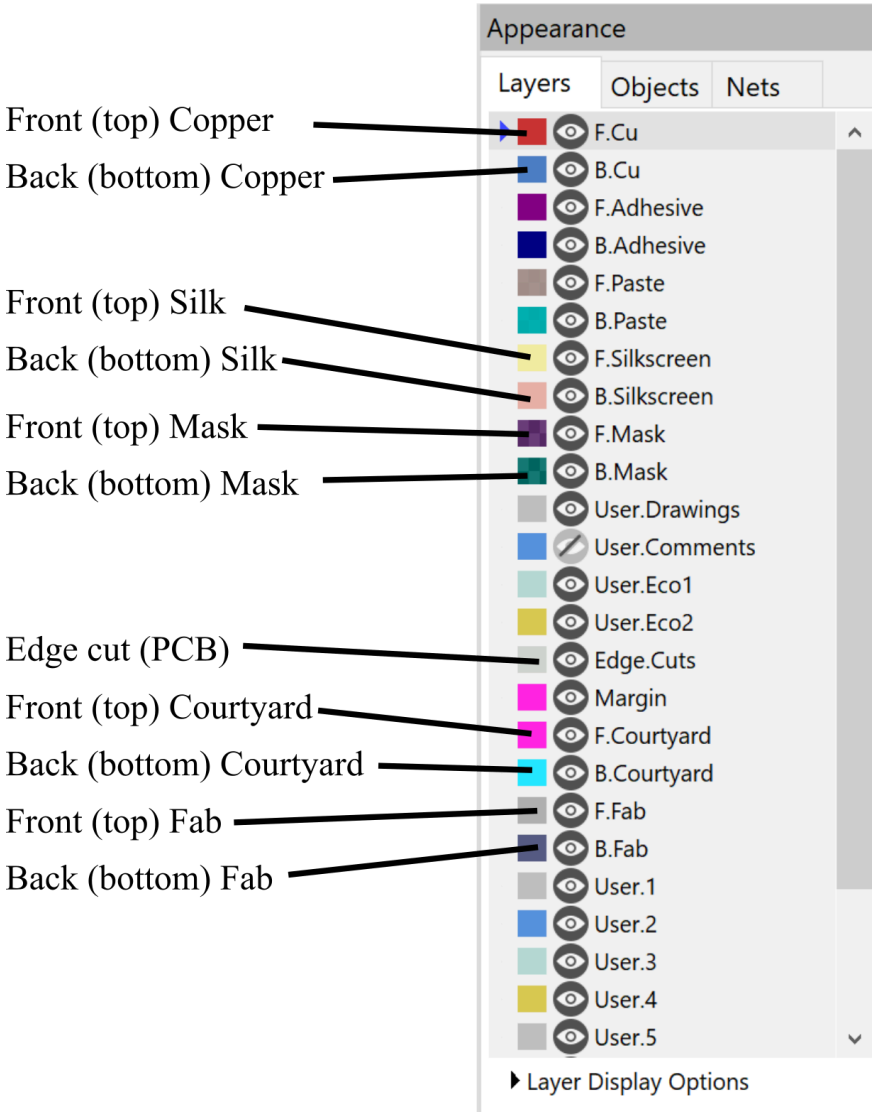


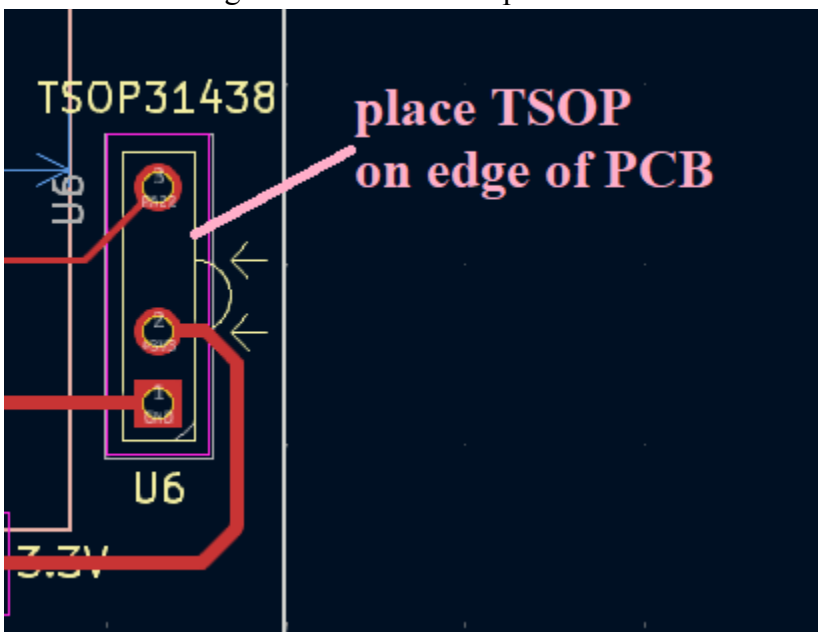

*Figure 11. PCB colors*

Step 1) Place parts to (show layers and grid again)
During layout you want to consider how the system will be used to create a hand-held game, placing display, slidepot, and buttons in positions to make game fun to play.

0) look at layers
1) avoid overlaps, think about 3-D nature of physical components;
2) decide which side of the PCB will we put the components
3) make fun to play;
4) minimize airwire length by grouping and rotating;
5) reduce airwire crossings.

Video Placement of Parts

Place TSOP at edge of PCB. Arrows represent the received IR light.




Step 2) Do a sanity check
0) Check airwires, do they all make sense?
1) Check nets, do they all make sense?
2) Do a DRC (checking for overlaps)

Step 3) Route power and ground

0) Route ground with traces with 0.1969in (0.500 mm ~20mil) traces, make a star pattern (no circles)
1) Execute DRC
2) Route +3.3V with traces with 0.1969in (0.500 mm ~20mil) traces, make a star pattern (no circles)
3) Execute DRC

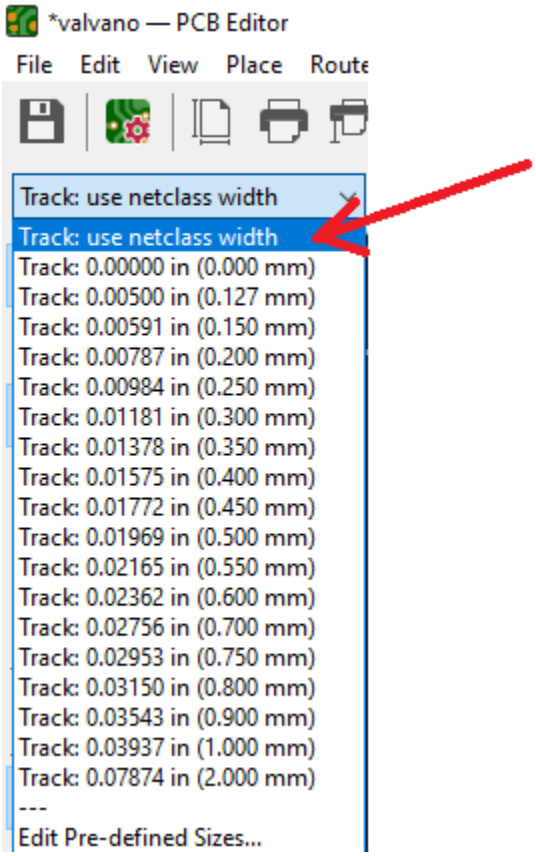
Video Route Power and Ground

Step 4) Route traces

0) Route traces with 0.0787in (0.200 mm ~8mil) traces,

1) Execute DRC

[Video Route Signal Traces]



Step 5) Arrange silkscreen as needed
    Label all testpoints, switches, LEDs, connectors, and jumpers
    Put your team members on PCB
    Execute DRC
Optional Image on Silk
    Paint BMP save as monochrome bitmap
    run import-bmp
    select black for image (which ever you wish to see)
    scale from 0.1 to 1 with trial and error
    place it top silk

Video Adding Silkscreen to the PCB

Step 6) Make a mock up of the physical PCB
    0) Print top
    1) Print bottom, mirrored
    2) glue to cardboard and arrange parts

Place three Assignment2 files (**.kicad_pro**, **.kicad_pcb** and **.kicad_sch**) in a separate folder, with the folder name matching one of your EIDs, in this googledoc
https://drive.google.com/drive/folders/15nr45plC7ZSaE-sAzlAO6NLI1AO18F88

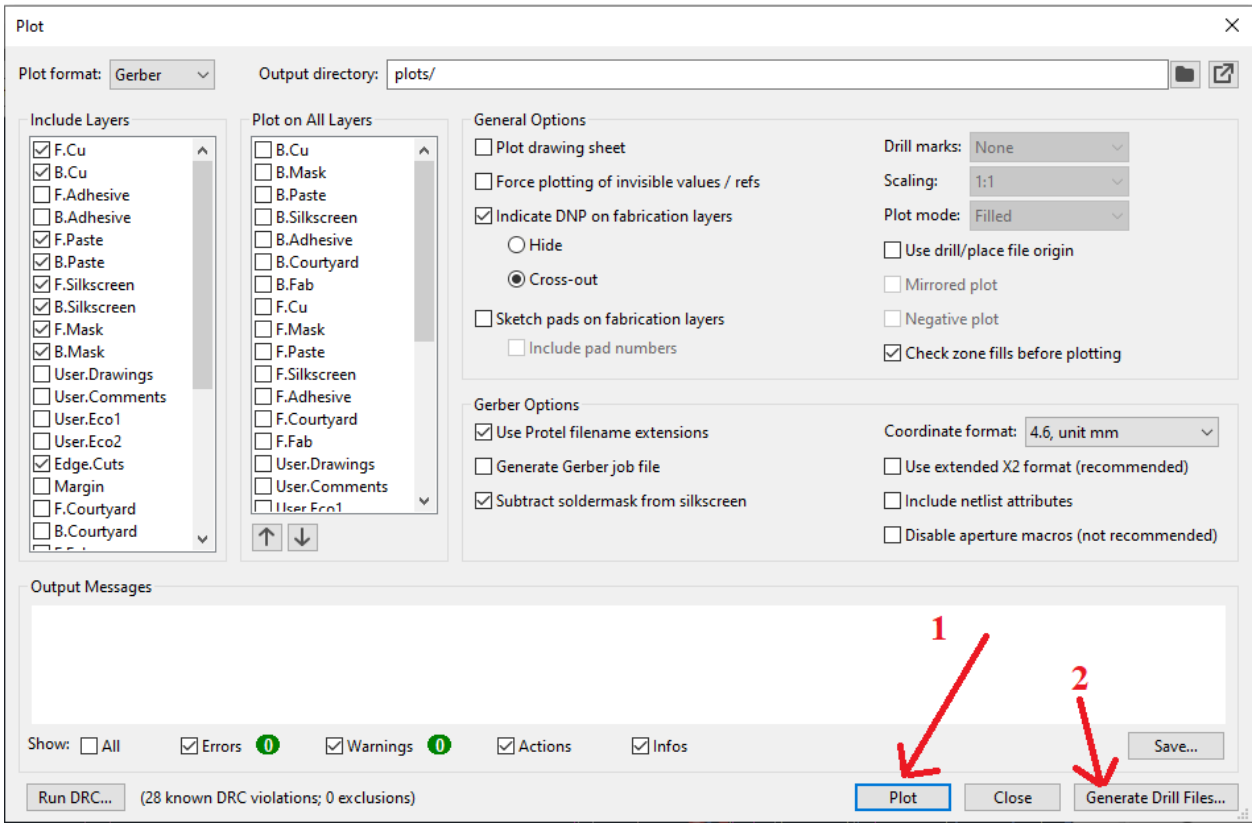
and Valvano will review, look for a docx file with my comments. I will not edit your SCH/PCB files (you have the master). When review is complete, I will make a comment in the associated docx file. You may look at designs from other students, but please do not edit/delete files that are not yours. After review but not completely done, Valvano will move entire folder to
https://drive.google.com/drive/folders/152-47VKYSQPW_pt_igfjpkJXVeswatQ3

When your review satisfies my standards, Valvano will move entire folder to
https://drive.google.com/drive/folders/1xr5QaeHwUrPVE1b5f8Bfi7bwEETKVqkx
and you are done (then I buy it if finished by deadline and group is 2-5 students).

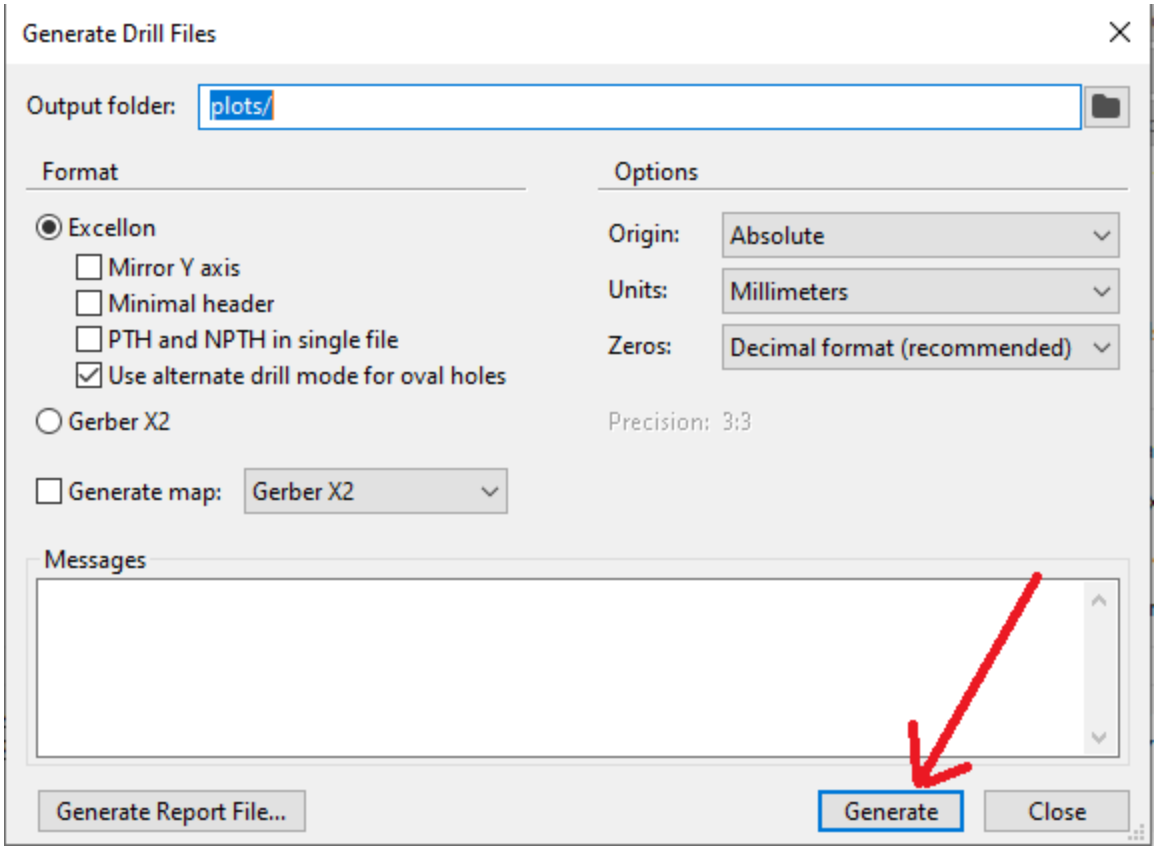
**Ordering PCBs (Valvano does this)**
There are many possible places to order boards. I suggest either of these:
https://oshpark.com/ is located in the US
https://jlcpcb.com/ is located in China

Video PCB Production
1) For oshpark, see https://docs.oshpark.com/services/two-layer/
For jlcpcb see
https://jlcpcb.com/help/article/362-how-to-generate-gerber-and-drill-files-in-kicad-7

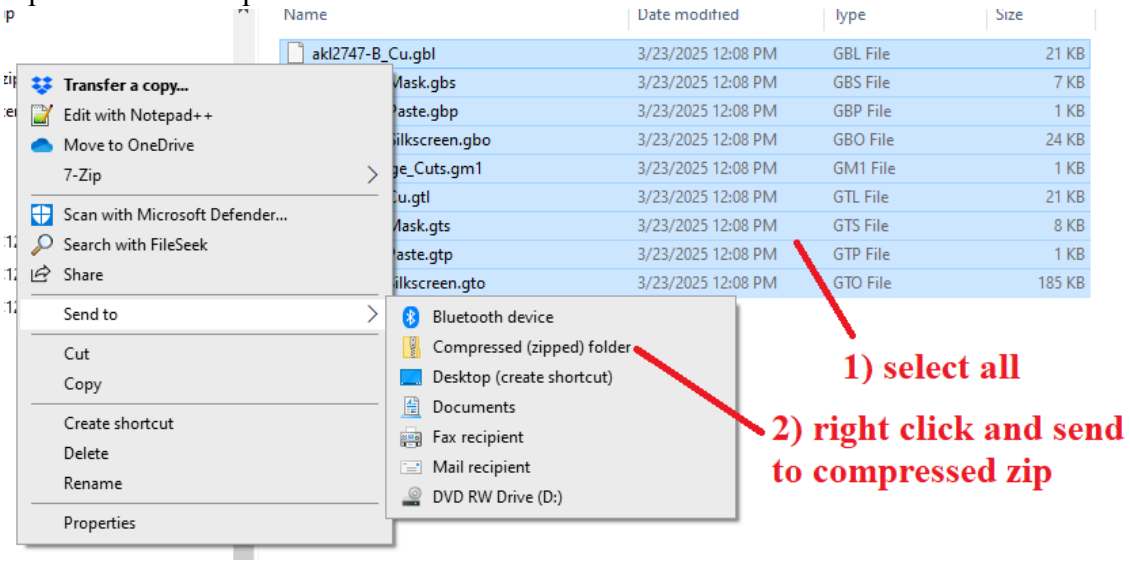
**Plot** makes Gerbers,




**Generate Drill Files** creates the holes




Zip the files in the plots folder




2) upload zip to oshpark or jlcpcb
Specifications for JLCPCB can be found at
https://jlcpcb.com/capabilities/Capabilities

For oshpark, see https://docs.oshpark.com/services/two-layer/

**After camp is over**
**Valvano orders green PCBs from JLCPCB**
**Students get PCB from Valvano or TA**

**PCB production when you want to make your own PCBs**
JLCPCB https://jlcpcb.com
5 copies, cool color options, less than 2 weeks with expensive shipping
1) Pass DRC
2) Create Gerber files,
https://jlcpcb.com/help/article/362-how-to-generate-gerber-and-drill-files-in-kicad-7
3) Upload Gerber.zip file to JCLPCB

**When PCBs arrive**
1) Fall 2025, Valvano puts PCBs in cabinet and TA gives out in lab. They are in a box in the cabinet in the 319K/319H/302 lab labeled PCB Camp. Ask any TA to give you the PCB.

If your team communicates effectively with each other, then one student can take all five and let every know how to get the board from you.

If your team has not communicated well, please just take one PCB

All the boards look the same, be careful to select the correct one.
Feel free to use extra boards for when you mess up soldering, or give away to friends
2) Make a list of parts you need. Most PCBs need

4 one-inch spacesA
4 4-40 screws
2 2by10 male-male headers for LaunchPad
2 3-pin headers
2 jumpers

If you are in 319H your PCB might need

1 ULN2003B
1 IR LED
1 IR sensor

If you added the amplifier, you will need

1 MC34119P

Resistors, capacitors can be gotten from EER checkout.
You will buy Joysticks, LaunchPads and LCDs on your own
ECE319K parts can be obtained from ECE 319K TA
3) Take the list to EER checkout and ECE319K TA. Please take one set of parts per student; don't try and get parts for all students. Get parts just for you.

**Students Solder**

Collect solder, soldering iron, cutter, solder paste, patience
Check all parts before soldering: the LCD, jack and switches may

Collect all parts: use an ohmmeter to check resistor values
https://www.instructables.com/Simple-PCB-soldering/
https://www.youtube.com/watch?v=AqvHogekDI4
Solder the parts in reverse size order (resistors first and OLED last)

If the jack is wrong, plug your headphones into the jack, and determine which two pins have one of the audio

- Audio channels (resistances between pins will be R, R, 2R). R is typically 32 ohms. pick one set of two pins that is R, cut off the other pin and solder the good pins to the PCB