Title: Spring 2022 Camp, Printed Circuit Board Design

Instructor: Jonathan Valvano valvano@mail.utexas.edu

Class: Self-paced and asynchronous

Office Hours: See EE319K Syllabus Page on Canvas for zoom link
Sat, M, Tu, W, Th, F 8pm zoom for demos and questions

Lectures: Prerecorded, posted on YouTube, and linked here

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 EE319K/EE319H grade.

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

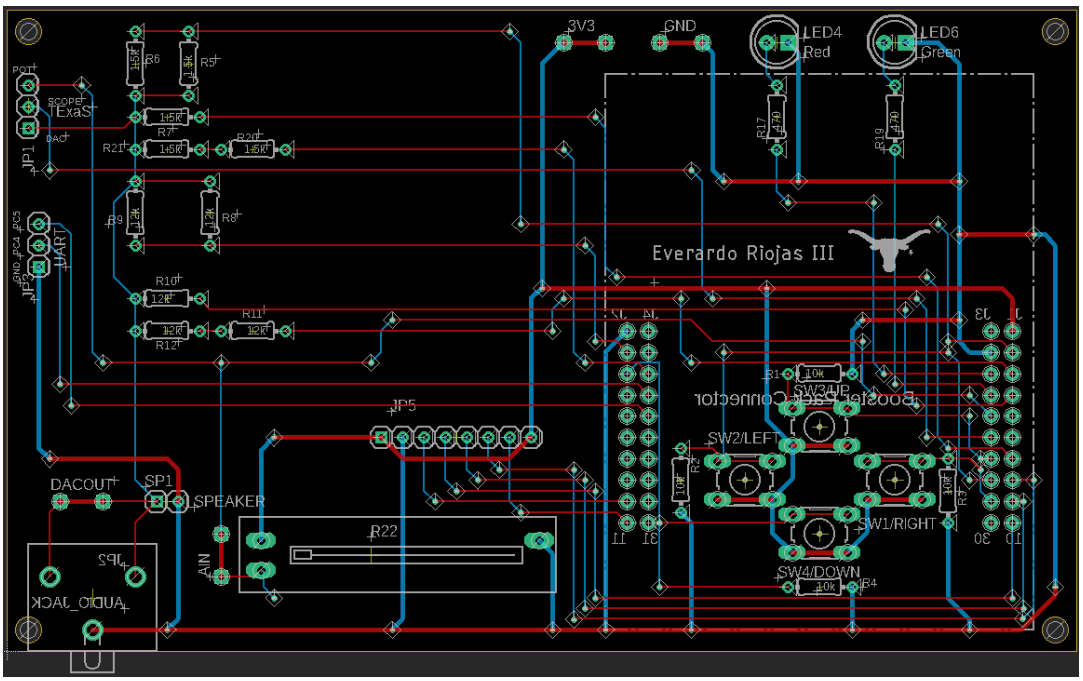


Figure 1. Eagle PCB showing a typical EE319K project (from Spring 2022).

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 EE319K/EE319H PCB that contains all the circuits required to complete Labs 6, 7, 8, and 10. The circuit design and software will be presented as part of regular EE319K/EE319H, 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 EE319K/EE319H lab kit. Students will learn to consider issues like drill holes, high current routing, standard values, tolerances, and testing. Students draw the electrical

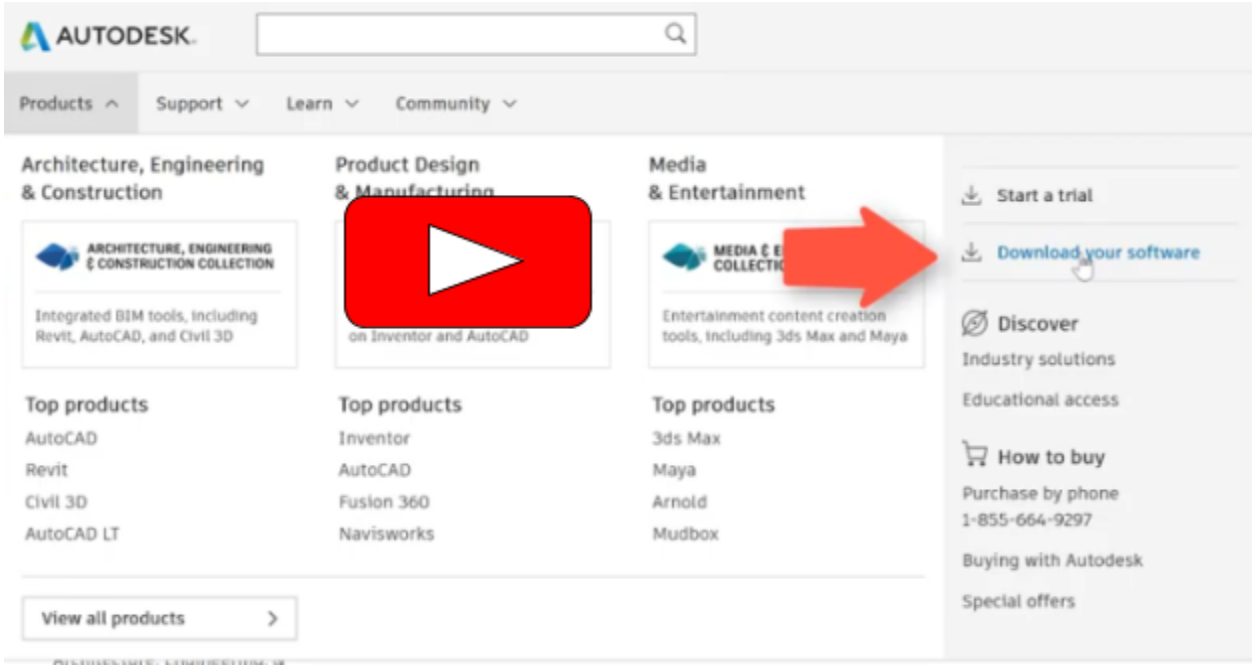
circuits in schematic form as the first assignment, performed individually. If you complete assignment 1 by Wednesday 8pm, teams of 2 or 3 will be formed and the team will create one PCB design. Teams who complete their PCB layout by **Friday 3/18 8pm** will have their board manufactured, making five copies. 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, 9, and 10. To use the PCB for EE319K/EE319H 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 Eagle by Autodesk.

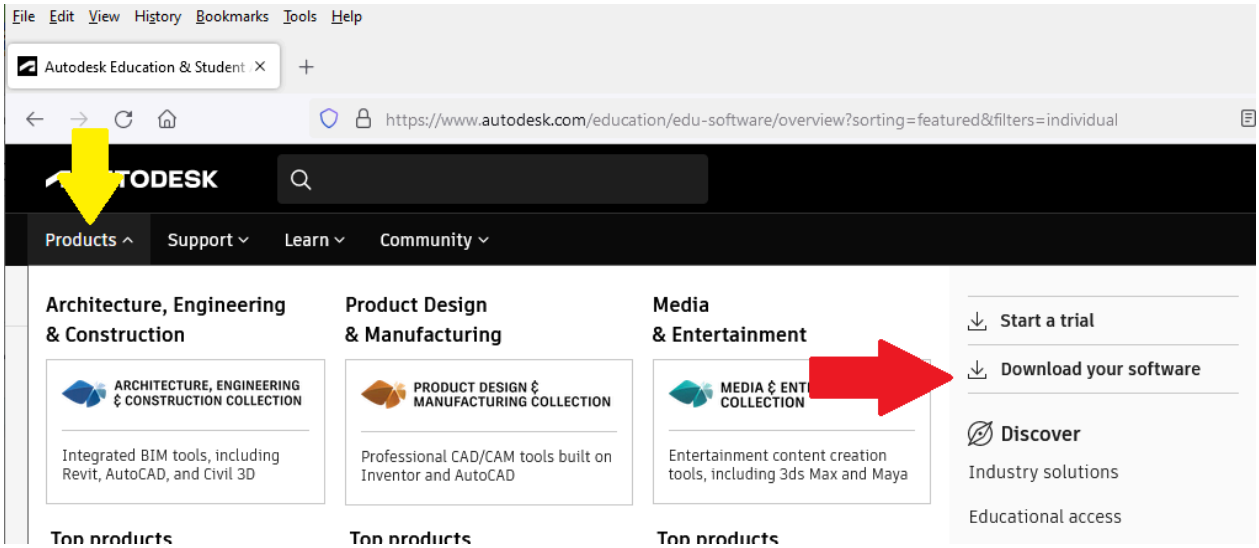
Pre-Assignment 1) Install Autodesk Eagle (prior to Monday 8pm):
Step 1) Create an student account on Autodesk, and register as a student:
https://www.autodesk.com/education/free-software/eagle

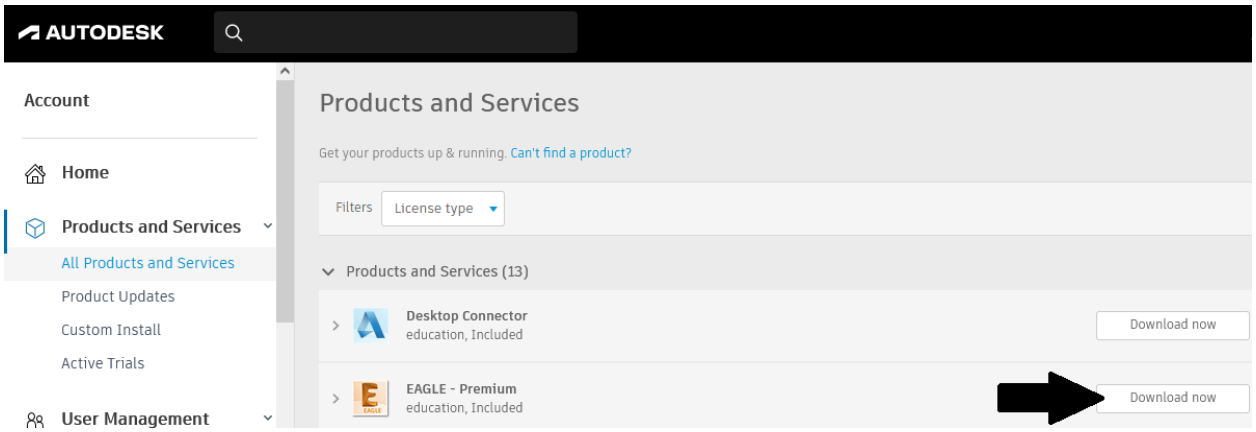
Step 2) Watch video 1



Video 1) Installing Eagle https://youtu.be/KWYJ02tlG4I

Step 3) Install latest version of educational version Eagle (do not spend any money)
Download and install **Autodesk_EAGLE_9.6.2_English_Win_64bit.exe**





Step 3 option B) Try these 3B) steps only if the above web page does not include downloading Eagle.

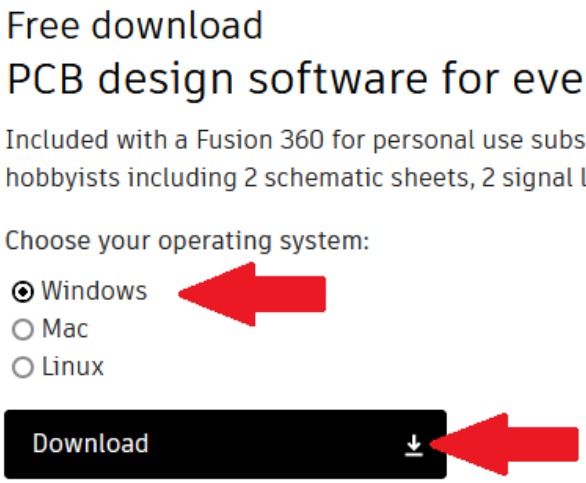
https://engineering.purdue.edu/ECN/Support/KB/Docs/AutodeskEagleStudentL

a) Double check your Autodesk account is a student account using a **utexas.edu** email

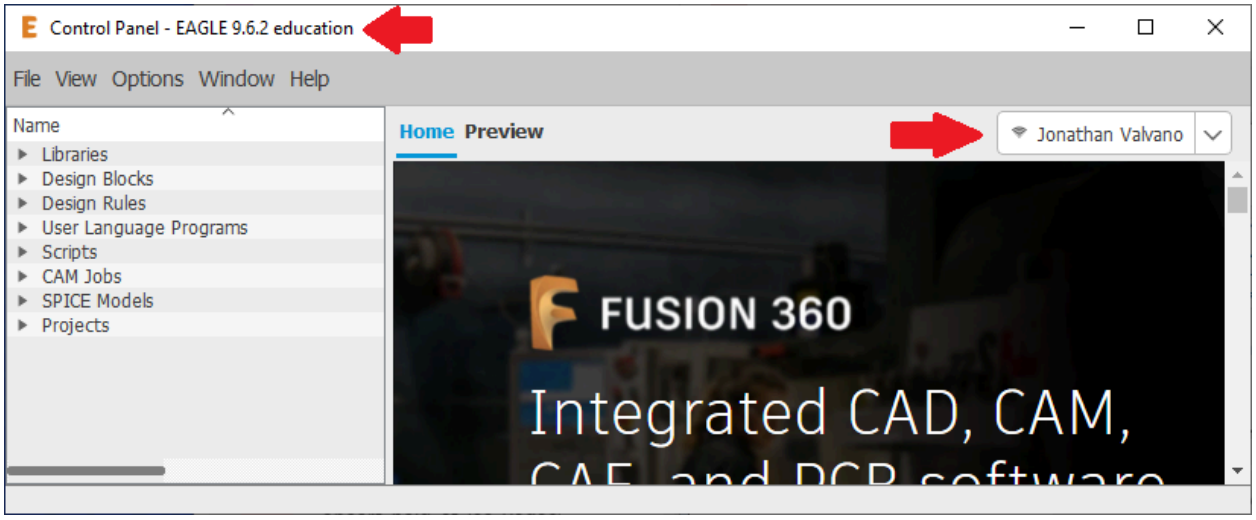
b) Log out of Autodesk

c) Go to https://www.autodesk.com/products/eagle/free-download

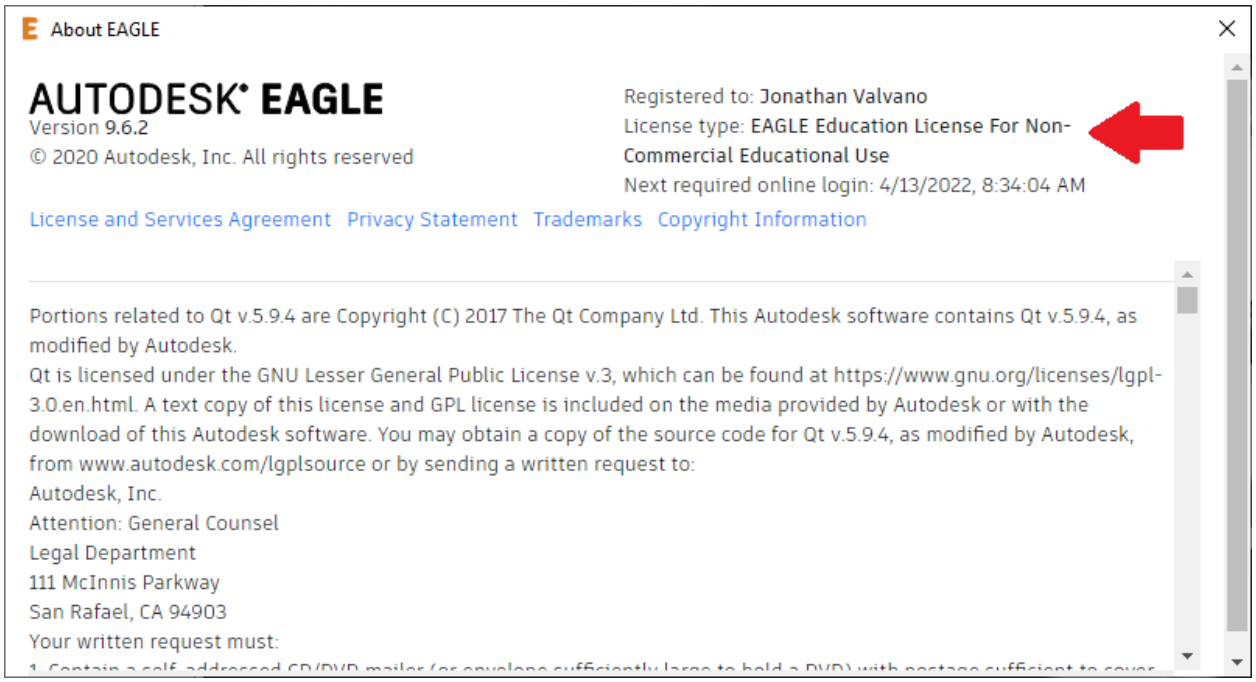
d) Click on Eagle Free Trial, download and install the free version



e) When you launch Eagle for the first time, it will ask you to log in. Log in with your two-factor and Eagle should switch to the educational version, showing **education** in the top bar and your name on the right top side.



f) Once you have download Eagle while logged in with your Autodesk account, it may take some time before you will have access to the full version of Eagle. Autodesk has suggested to us to give it up to two days. If you still do not have access to the full version of Eagle, email support.eagle@autodesk.com. The following image is the license information when Eagle is working as full educational version.



Pre-Assignment 2) Bookmark these links for your reference (prior to Monday 8pm):
There is a lot of useful information on the web for using Eagle

Eagle quits immediately on Launch
https://knowledge.autodesk.com/support/eagle/troubleshooting/caas/sfdcarticles/sfdcarticles/Eagle-crashes-seconds-after-launching-splash-screen.html

Tutorials:

https://www.shawnvictor.net/autodesk-eagle.html ***my favorite****
https://learn.sparkfun.com/tutorials/using-eagle-schematic
https://learn.sparkfun.com/tutorials/using-eagle-board-layout
http://www.instructables.com/id/Turn-your-EAGLE-schematic-into-a-PCB/

Pre-Assignment 3) Configure Eagle to work with this PCB camp (prior to Monday 8pm)
Link to my directories https://utexas.box.com/s/ru4zazogvpg95d08h20r7te8zkz5k4to
Step 1) Create a folder for this camp

Step 2) Create a subfolder called **Library** on your computer and copy this file into it
https://www.dropbox.com/s/0ejc0y3xmyl91u8/EE445L.lbr?dl=1

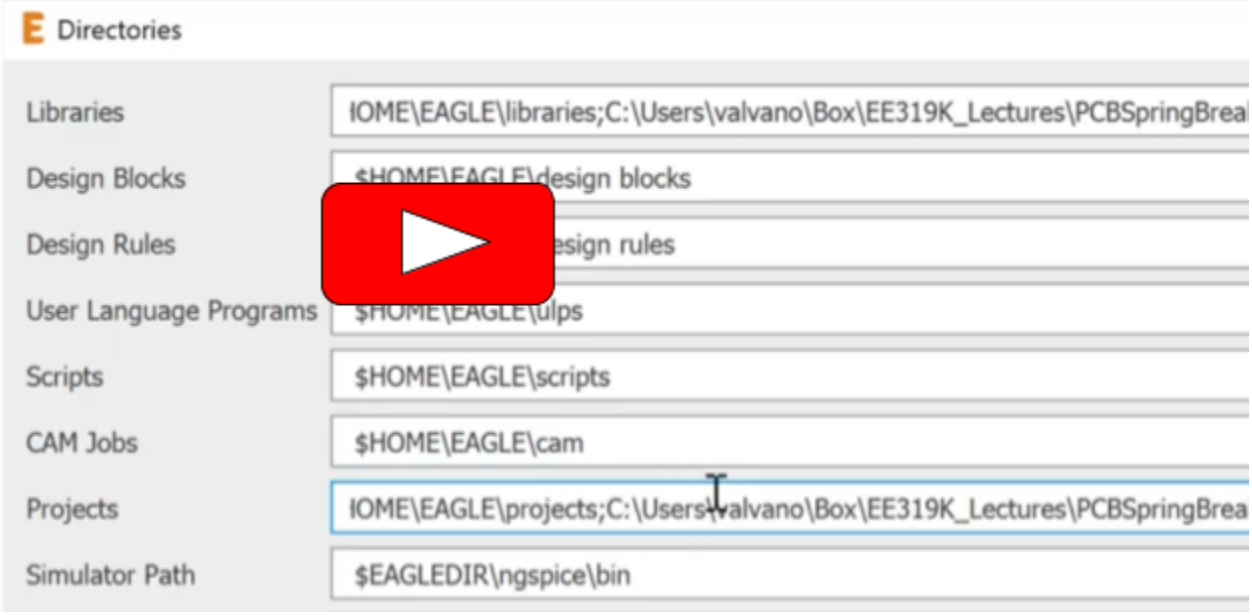
Step 3) Create a subfolder called **Design** on your computer and copy these two files into it
https://www.dropbox.com/s/xq232ss35m2szbw/EE319K_Baseline_Schematic.sch?dl=1
https://www.dropbox.com/s/h3x5kxdiq5l4u8h/EE319K_Baseline_Schematic.brd?dl=1

Rename these two files with your EID (extensions are different, name must match)

WARNING: You must have both the SCH and BRD files open whenever you edit either. Editing with one file open will result in SO MANY errors you will have to start over!

Step 4) Create a subfolder called **Datasheets** on your computer and copy these files into it
https://drive.google.com/drive/folders/1-5_a7SrrhwGThlpLn_bKOAbU1F1jL4xG?usp=sharing

Step 5) Watch Video 2 and link your library and design folders into your Eagle



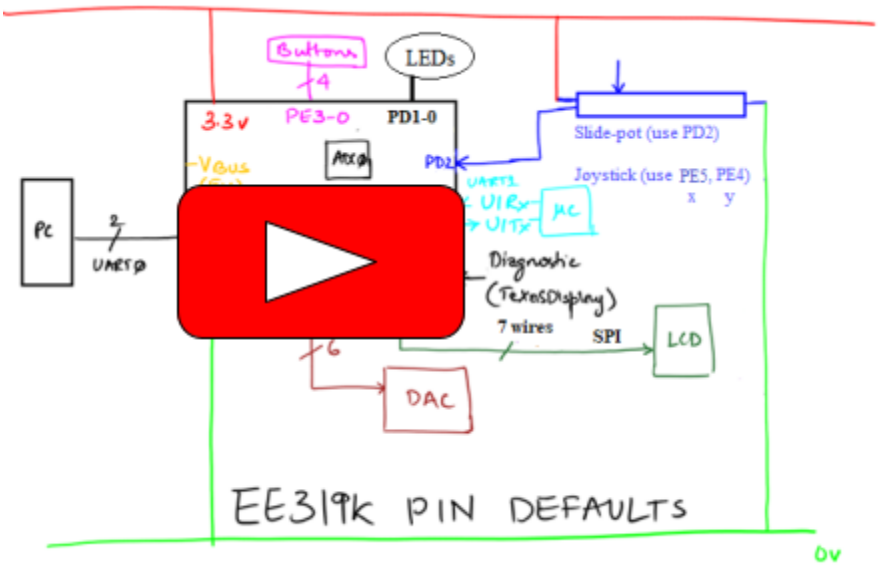
Video 2) Configuring Eagle for PCB Camp https://youtu.be/i4pFbAbk2hY

Optional additional Eagle Libraries
Sparkfun
https://learn.sparkfun.com/tutorials/how-to-install-and-setup-eagle/using-the-sparkfun-libraries
Element14 https://github.com/steakunderscore/eagle-cad-libraries
https://www.element14.com/community/community/cadsoft_eagle/eagle_cad_libraries

Pre-Assignment 4) Think about Labs 6, 7, 8, 9, and 10 (prior to Monday 8pm)
Think about all the hardware you will need for Labs 6-10. 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.

Warning: PB7 is connected to PD1 on the LaunchPad. Similarly PB6 is connected to PD0. You can remove R9 and R10 from your LaunchPads to separate these two connections, allowing for the independent use of PB7 PB6 PD1 and PD0.

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 EE319K or EE319H? (EE319H needs TSOP31438, IR LED and ULN2003B)
- do you want the headphone jack and speaker or just the speaker?
- do you want to have a joystick?



Video 3) Planning the design. https://youtu.be/EhE8tGz1FI4 (from Fall 2021).

Step 1) Expand your Lab 3 circuits to have four switches and two LEDs. You are welcome to add additional LEDs to any unused pins.

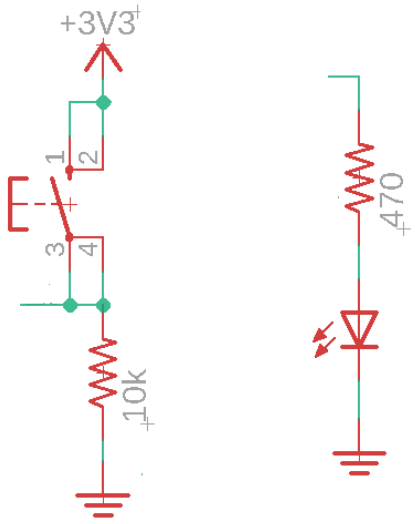


Figure 2. Eagle schematic drawing showing one switch, and one LED.

Step 2) Plan your 6-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.

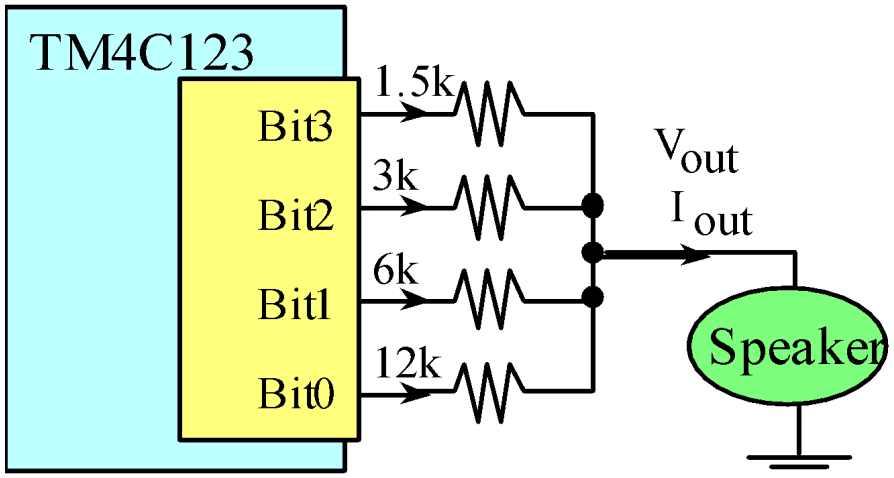


Figure 3. Four bit binary weighted DAC. You will make a 6 bit DAC, just like Lab 6.

Step 3) The Adafruit LCD to the microcontroller has been completed for you in the starter project. The **CARD_CS** signal is connected to PD7 to allow you to use the SD card feature on the ST7735 in future projects. Your software must set PD7 to output=3.3V, so the SDC is off.

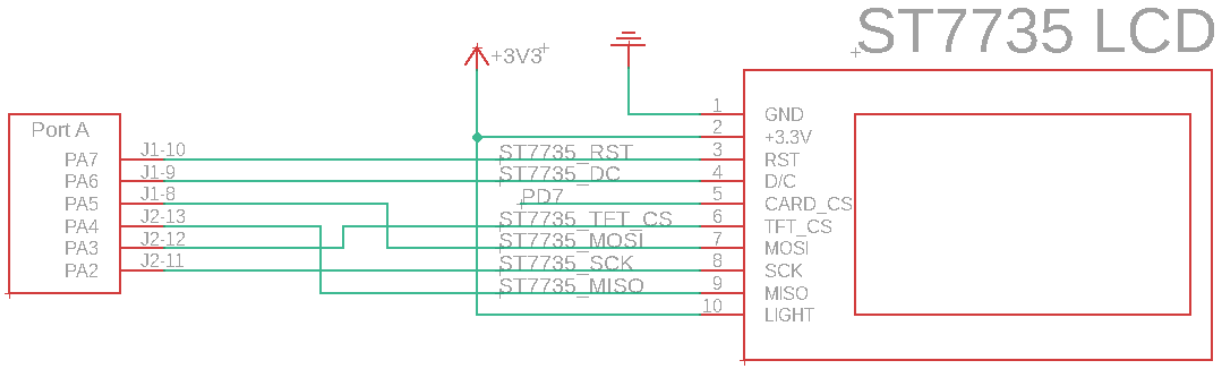


Figure 4. Interface connections for the ST7735R LCD.

Warning: spring 2022 starter files have mistakenly connected CARD_CS to PB0. Disconnect CARD_CS from PB0 and then connect CARD_CS to PD7. Be careful not to merge the PB0 and PD7 nets.

It is ok if your ST7735 is slightly different from the Adafruit version in Figure 4. For example, the HiLetgo 1.8" inch ST7735R SPI 128160 TFT LCD Display Module with PCB has 16 pins. In this case, we place a 1-by-16 0.1in-spacing pin header into the schematic and carefully wire the pins. For details on the connections, see the ST7735.c file in your Lab 7 starter project. The EE445L library part is PINHD-1X16. If you have a HiLetgo, copy the following text into the schematic to help you wire it up correctly.

```
// **********HiLetgo ST7735 TFT and SDC
// GND (pin 1)  connected to ground
// VCC (pin 2)  connected to +3.3 V
// NC  (pins 3,4,5) not connected
// RESET  (pin 6)  TFT, to PA7 (GPIO)
// A0  (pin 7)  TFT, Data/Command PA6 (GPIO), high for data, low for command
// SDA (pin 8)  TFT, MOSI PA5 (SSI0Tx)
```

```
// SCL (pin 9)  TFT, SCK  PA2 (SSI0Clk)
// CS  (pin 10) TFT, PA3 (SSI0Fss)
// SCK (pin 11) SDC, serial clock, also connect to PA2 ST7735_SCK
// MISO   (pin 12) SDC, MISO, connect to PA4 ST7735_MISO
// MOSI   (pin 13) SDC, MOSI, also connect to ST7735_MOSI PA5
// SD_CS  (pin 14) SDC, chip select, connect to PD7
// LED+   (pin 15) TFT, connected to +3.3 V
// LED-   (pin 16) TFT, connected to ground
```

To create a team of 2 or more, we could create a PCB that can support two LCDs. You will route both but solder only one. There is a column in the signup sheet to specify the LCD part.

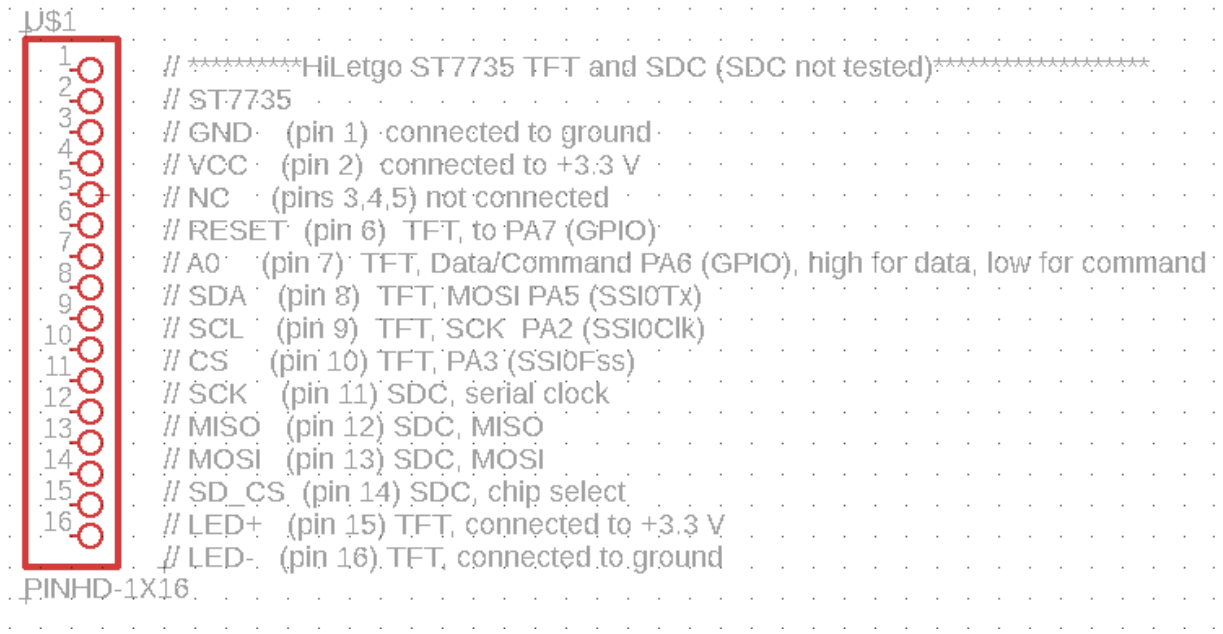


Figure 4b. Part for the HiLetgo ST7735R LCD.

Step 5) Connect the Slidepot to the microcontroller as described in Lab 8.

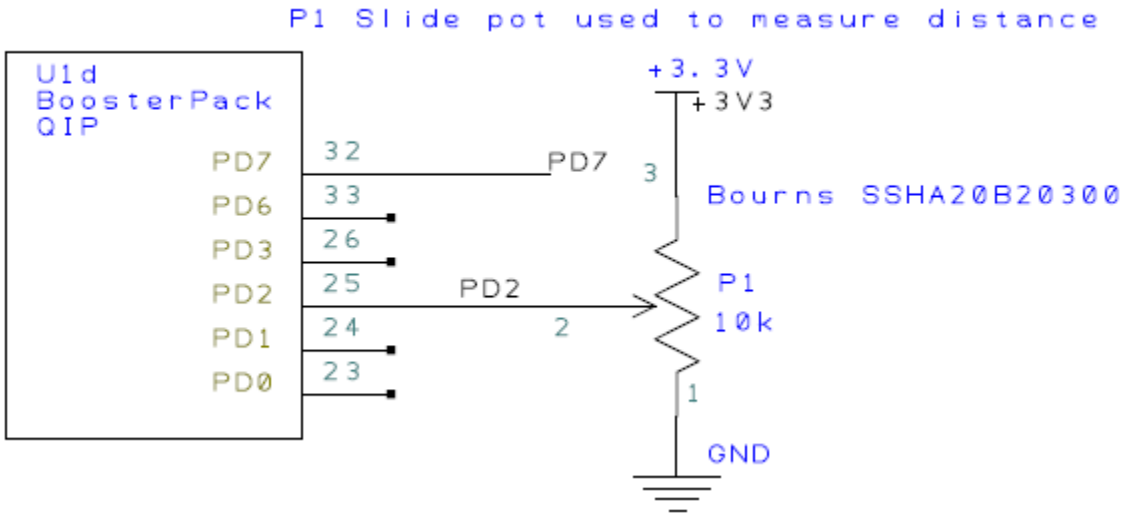


Figure 5. Possible circuit to interface the sensor.

Step 5) To use the TExaS oscilloscope, we connect an analog signal to PD3. A 3-pin jumper allows the operator to connect PD3 to either DACOUT or to PD2 (slide pot).

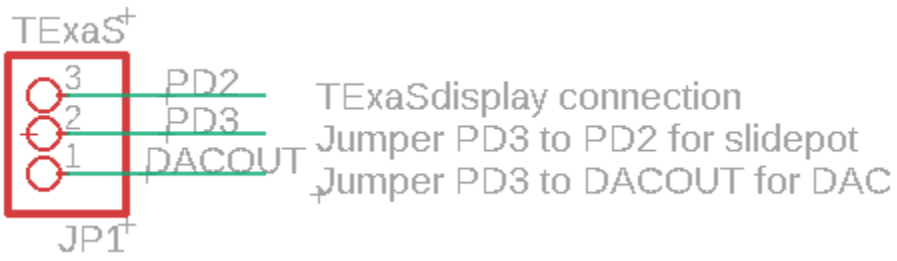


Figure 6. Placing the jumper between PD2 and PD3 connects the slide pot analog voltage to the TExaS scope. Placing the jumper between PD2 and DACOUT connects the DAC analog voltage to the TExaS scope. Removing the jumper altogether allows you to connect the PD3 pin to any analog input.

Step 6) Lab 9 uses PC4 and PC5 to implement serial communication with another system. To simplify Lab 9, we will connect PC4, PC5, GND to a 3-pin header. We can then place a shorting jump across PC4 to PC5 to complete Lab 9 (socially distancing). Or in Lab 10 you can use two boards with a 3-pin cable connecting PC4-PC5 PC5-PC4 and GND-GND.

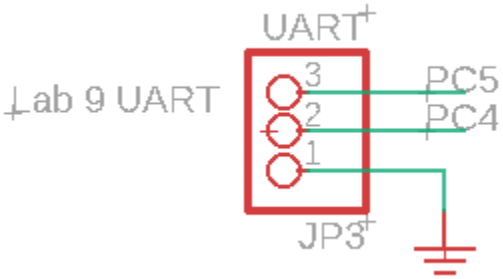


Figure 7. Placing the jumper between PC5 and PC4 allows you to test Lab 9 with one board.

Step 6b) EE319H Lab 9 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 PC5 output pin using a ULN2003B, a 220 ohm resistor and the +5V supply. I = (5-1.2-0.5)/220 = 15 mA.

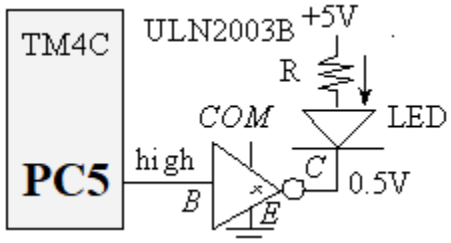
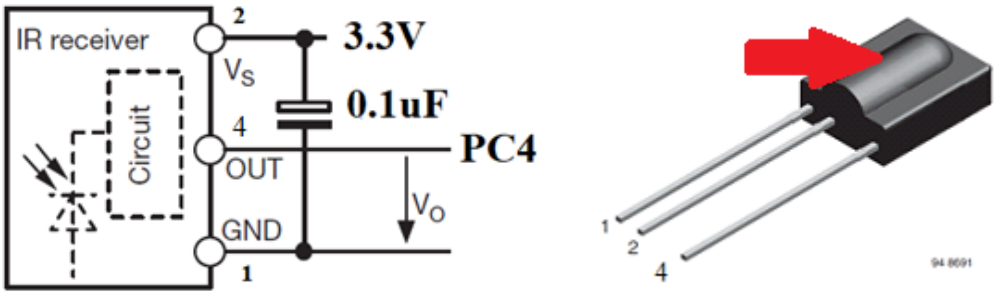


Figure 8. IR LED interface for EE319H.

For the TSOP31438 receiver simply use a **PINHD-1X4** part with 0.1in spacing, with 0.1in between pins 1 and 2, and 0.2in between pins 2 and 4. You will connect three of the four pins of

the header to the receiver, leaving pin 3 disconnected. The red arrow is the IR sensor. Position the TSOP31438 so the sensor points away from PCB.



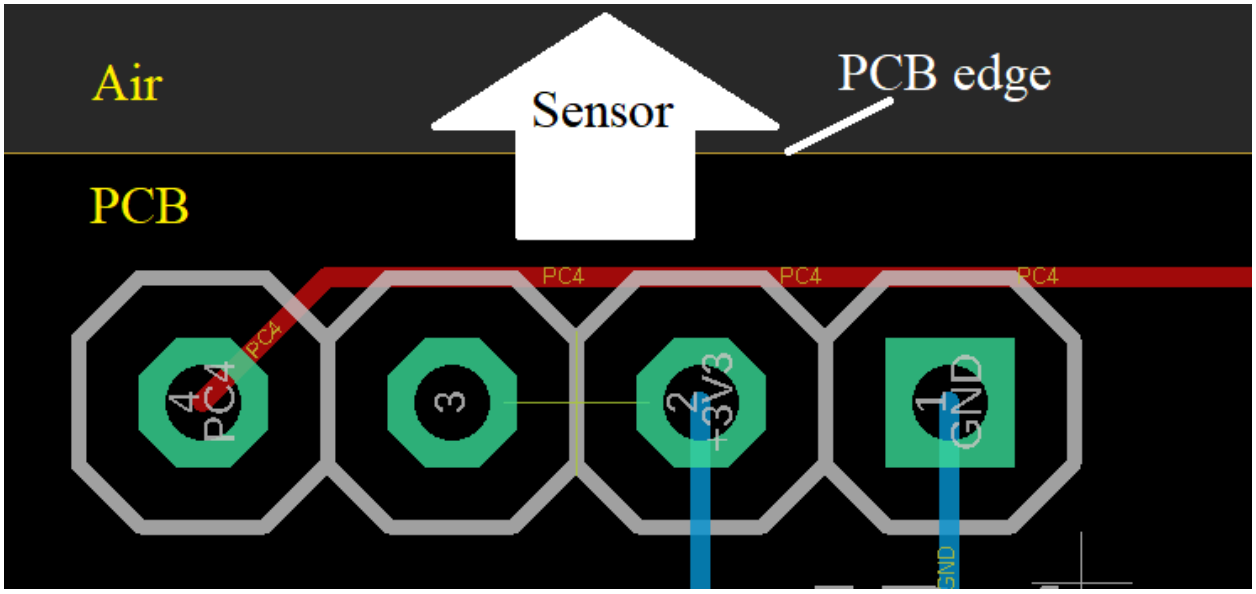
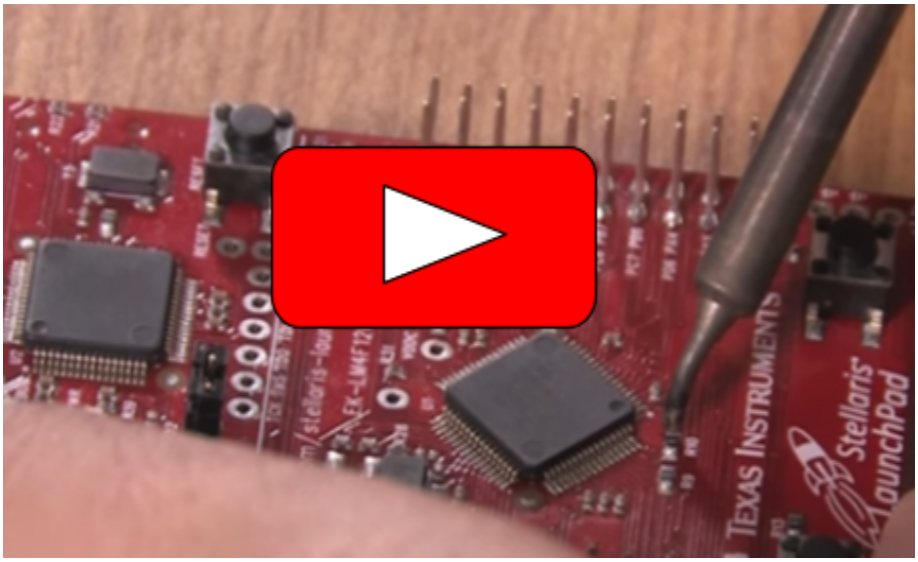


Figure 9. TSOP31438 IR sensor interface for EE319H. +Vs is 3.3V, C_1 is 0.1 uF, and OUT is connected to PC4 on the receiver. GND is pin 1, Vs is pin 2, OUT is pin 4.

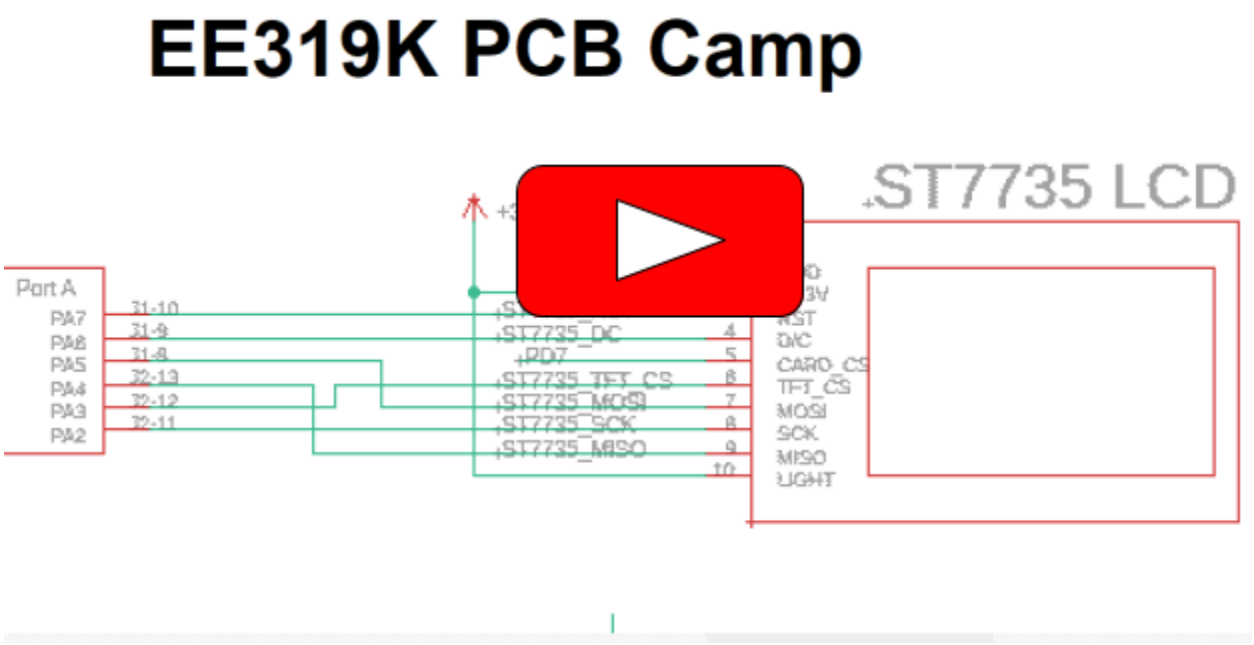
Optional: remove R9 and R10 to allow use of PB7 PB6 PD1 and PD0



Optional Video 4) Removing R9 R10 https://www.youtube.com/watch?v=MWIX7wgS9PM

Assignment 1) Draw the circuits for Labs 6,7,8,9,10 using Eagle (due Wednesday 8pm)
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. The goal for assignment 1 is for you to learn the following Eagle commands

- Grid and layers
- Parts have a red cross to select it
- Info of a part
  - name, value, position, rotation, footprint on PCB
- Some parts are not parts at all 3V3, Gnd
- Nets
- Move
- Rotate
- Group-Move
- Copy
- Delete

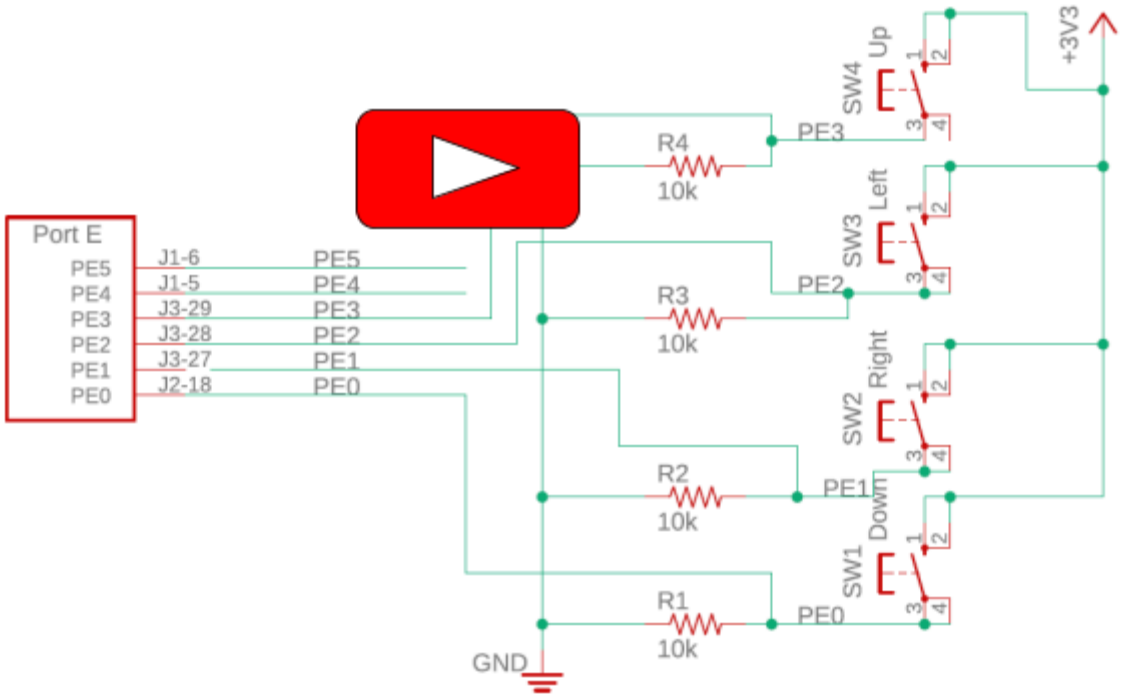


Video 5) Introduction to Eagle https://youtu.be/tal2p_uhG10

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

Move switches and 10k resistors together
Add net to complete circuit
Name net to connect to LaunchPad pins PE3-0
Info on switches to document functionality

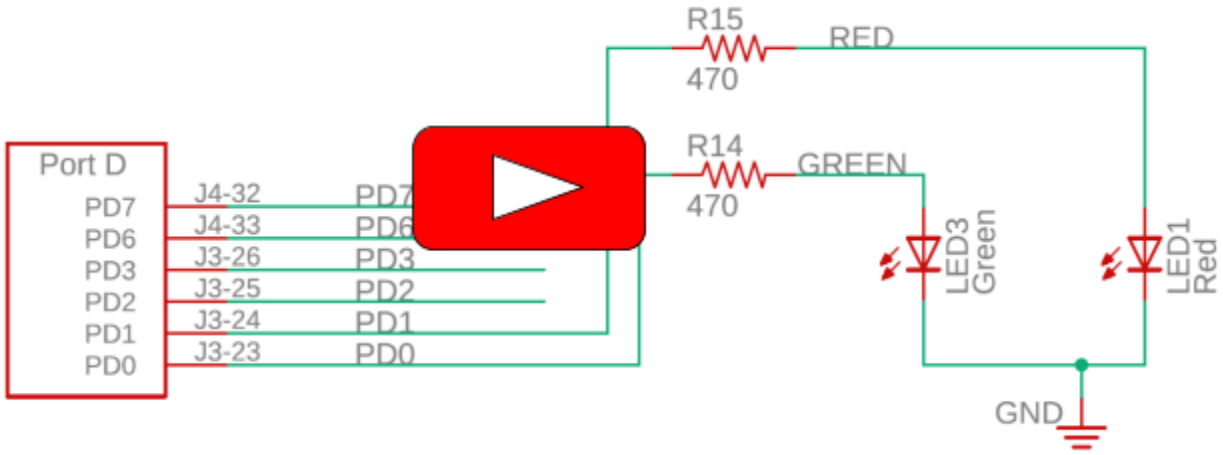
Hint: Place run an ERC check early and often to catch mistakes you might be making.



Video 6) Interface 4 switches https://youtu.be/gQFxn_cEEHI

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

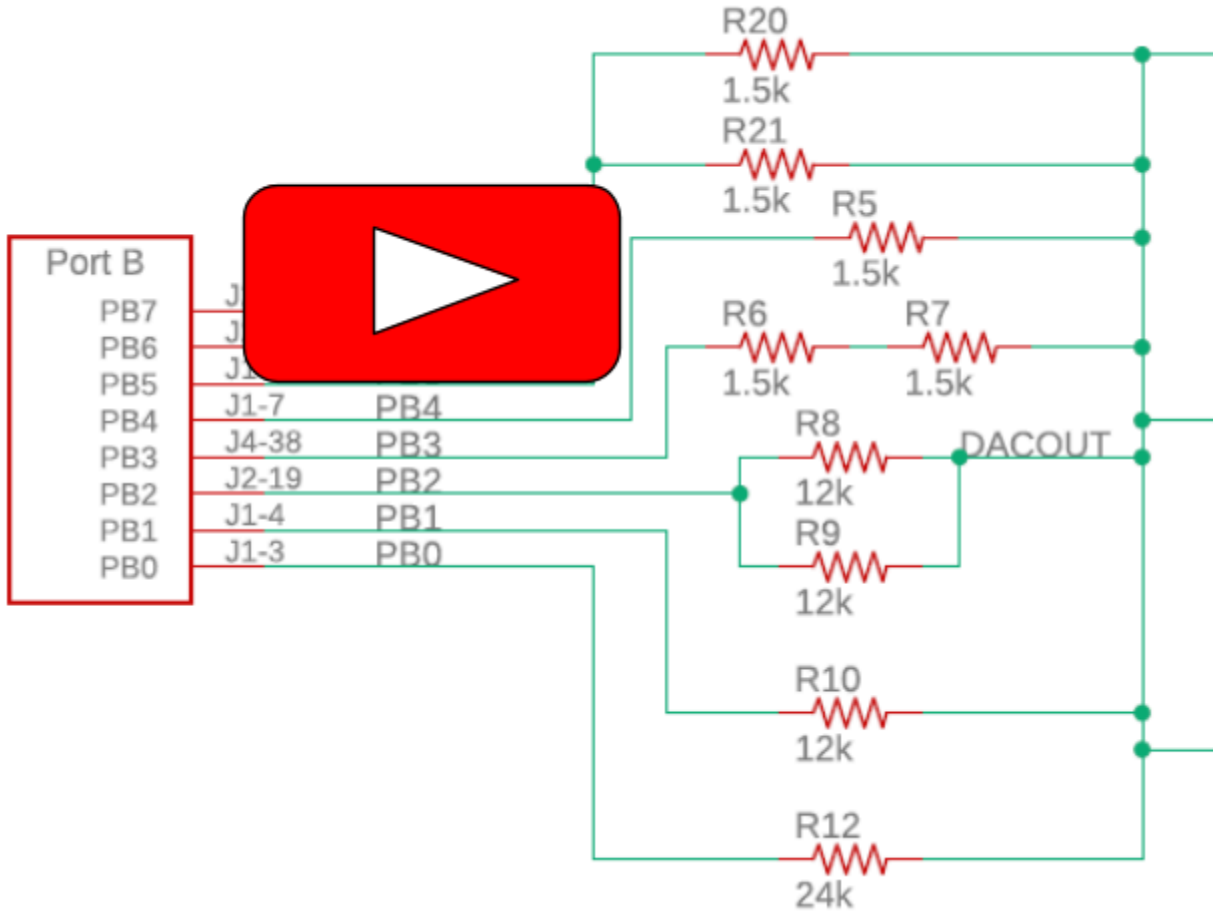
Move LEDs and 470 resistors together
Add net to complete circuit
Name net to connect to LaunchPad pins PD1 PD0
Info on LEDs to document functionality



Video 7) Interface two LEDs https://youtu.be/VJ6HfIG1cCo

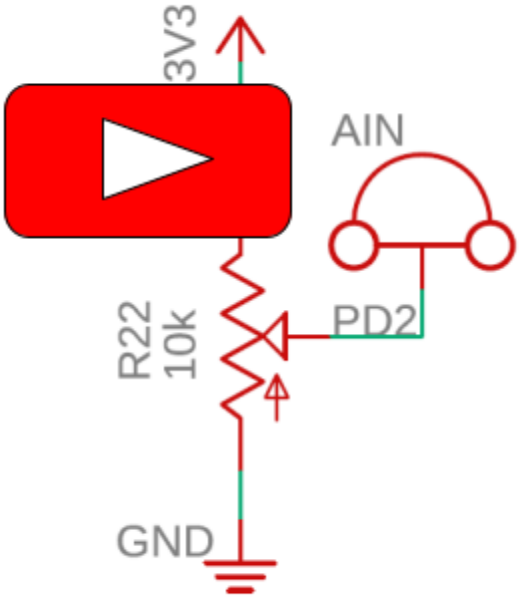
Step 3) Build DAC/speaker interfaces with Testpoint

Move Speaker**,** Jack, five 1.5k, and five 12k resistors together
Add net to complete circuit
Name net to connect to LaunchPad pins PB5-0



Video 8) Interface 6-bit DAC https://youtu.be/RYnsjcCwM50 . Spring 2022 has five 12k resistors and no 24k resistors.

Step 4) Build slide pot interface with Testpoint



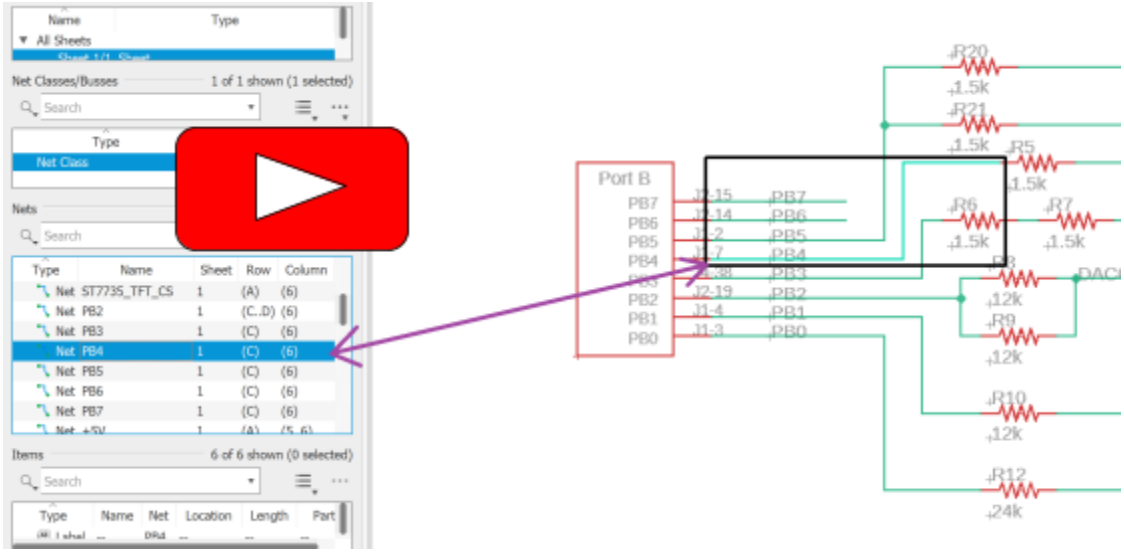
Video 9) Interface slidepot https://youtu.be/zEkeCP2Y8oI

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 it is correct

Observe the PCB to make sure all the parts are correct



Video 10) Review SCH, look at PCB https://youtu.be/dyoIrQPQxXQ

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

Add another library and connect joystick to PE4 and PE5

Step 6) Run an ERC check before submission

Assignment 2) Place the components on the BRD file (due Friday 8pm)
If you want to have Valvano purchase the PCB, you will perform assignment 2 in groups of two to five students. If you wish to purchase your own PCB (about $20), you can perform assignment 2 in groups less than 3. 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 I want Valvano to create the team. You can add one additional student. 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

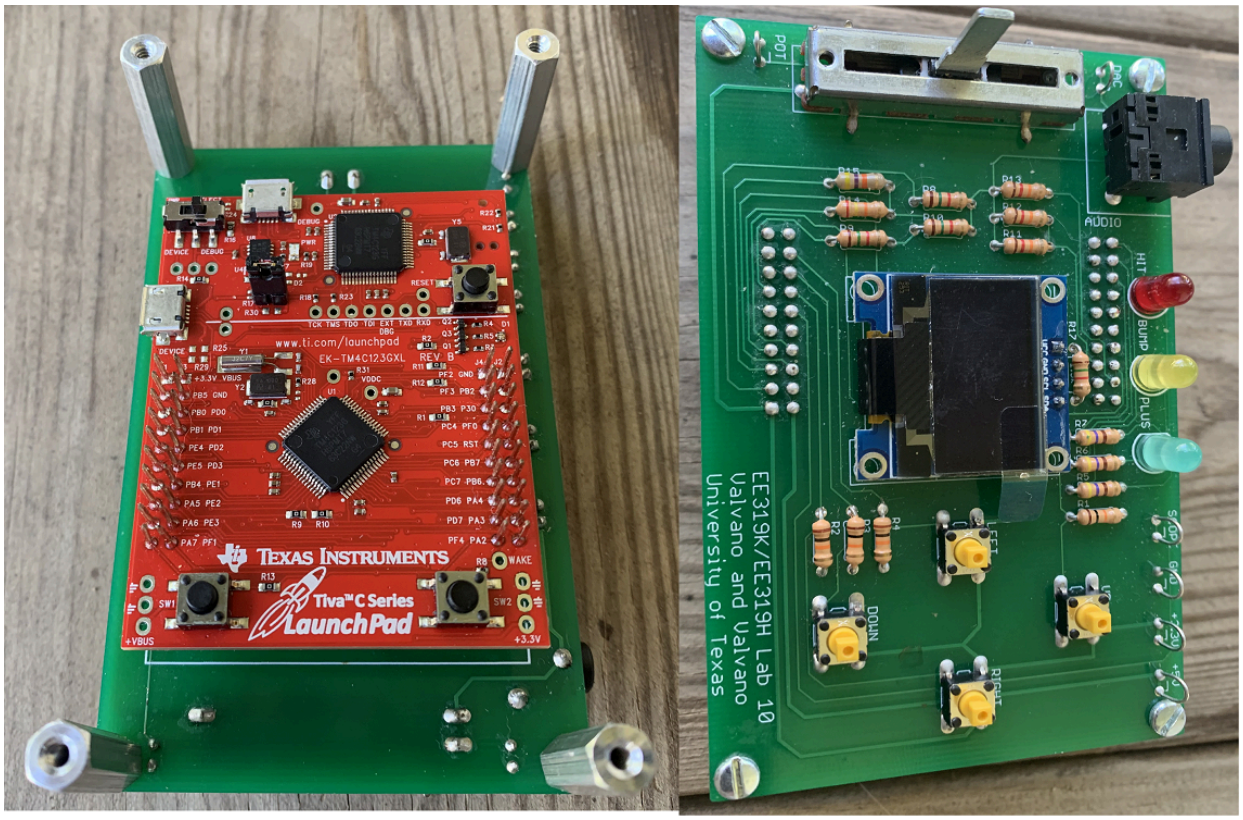
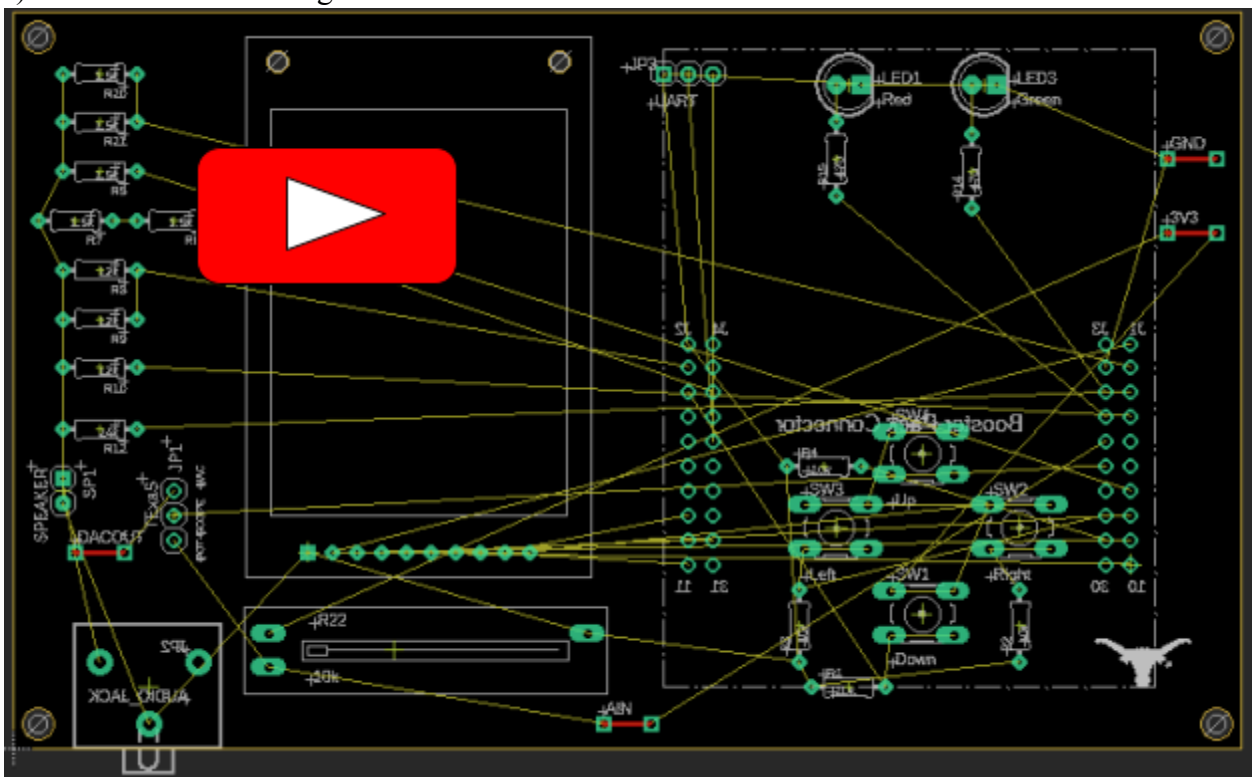


Figure 10. Example system from Spring 2021.

Step 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.



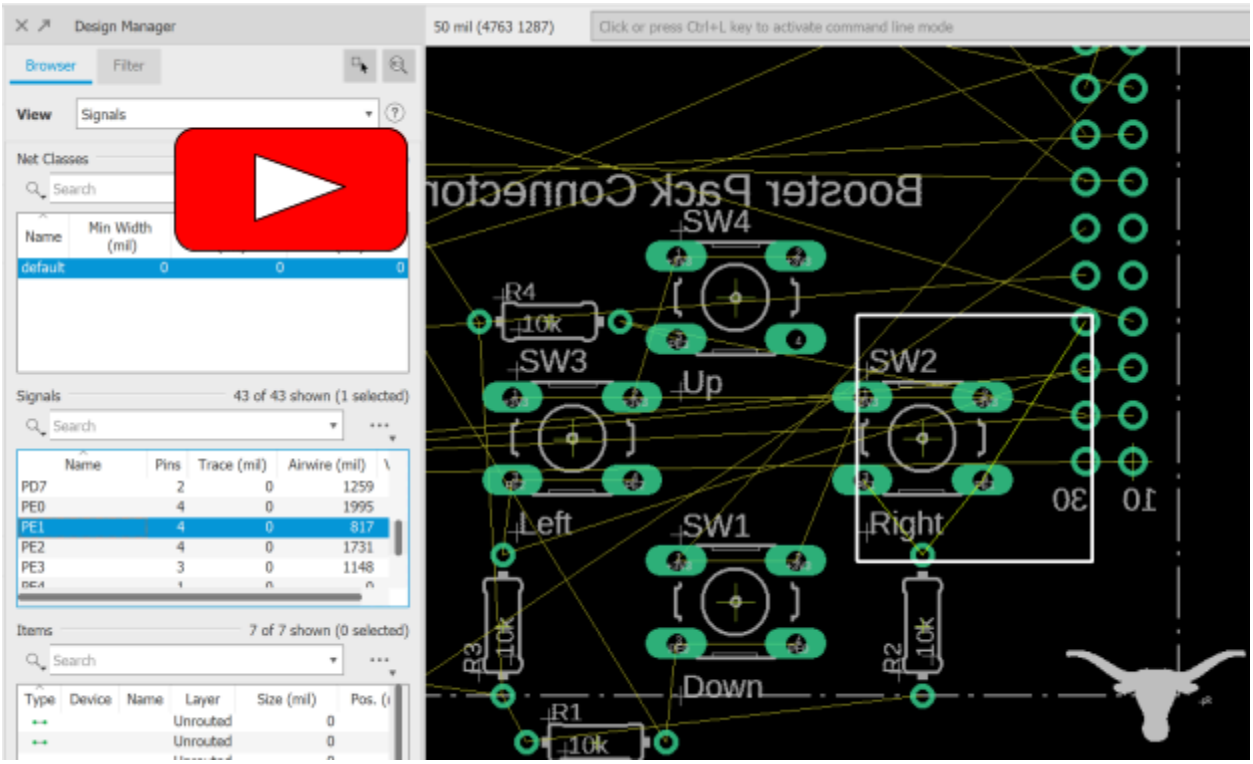
Video 11) Place parts https://youtu.be/84VTGInoiBY

Step 2) Do a sanity check

0) Check airwires, does it make sense?

1) Check nets, does it make sense?

2) Do a DRC (checking for overlaps)



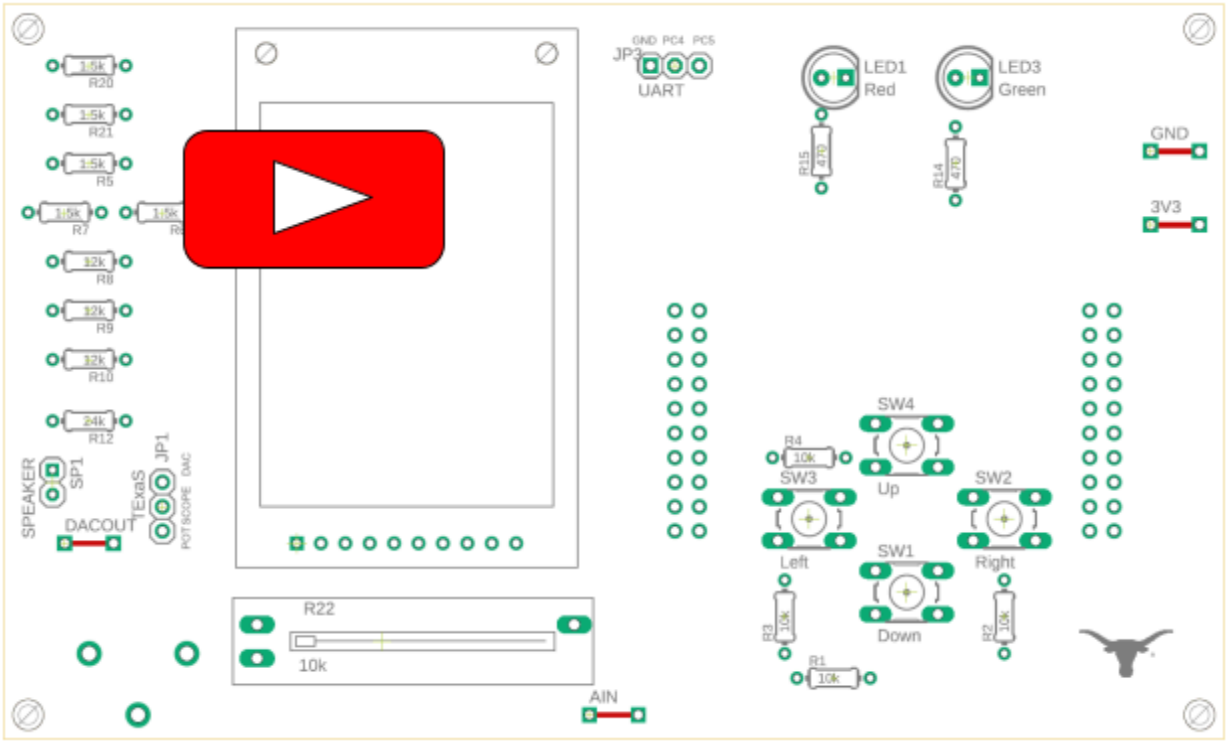
Video 12) Sanity Check https://youtu.be/FsllfVdFibc

Step 3) Make a mock up of the physical PCB

0) Print top

1) Print bottom, mirrored

2) glue to cardboard and arrange parts



Video 13) Mock up https://youtu.be/MgEVJINWSDM

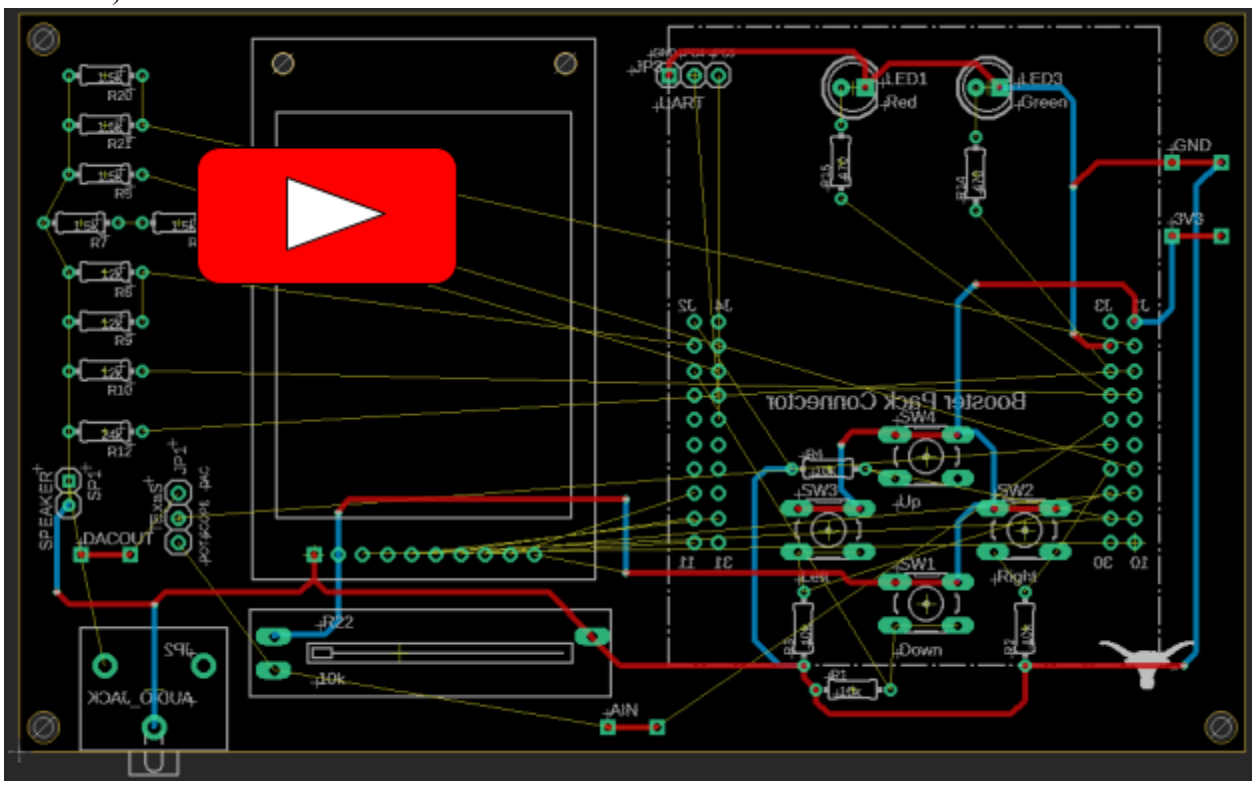
Step 4) Route power and ground

0) Route ground with 20mil trace, make a star pattern (no circles)

1) Execute DRC

2) Route +3.3V, make a star pattern (no circles)

3) Execute DRC

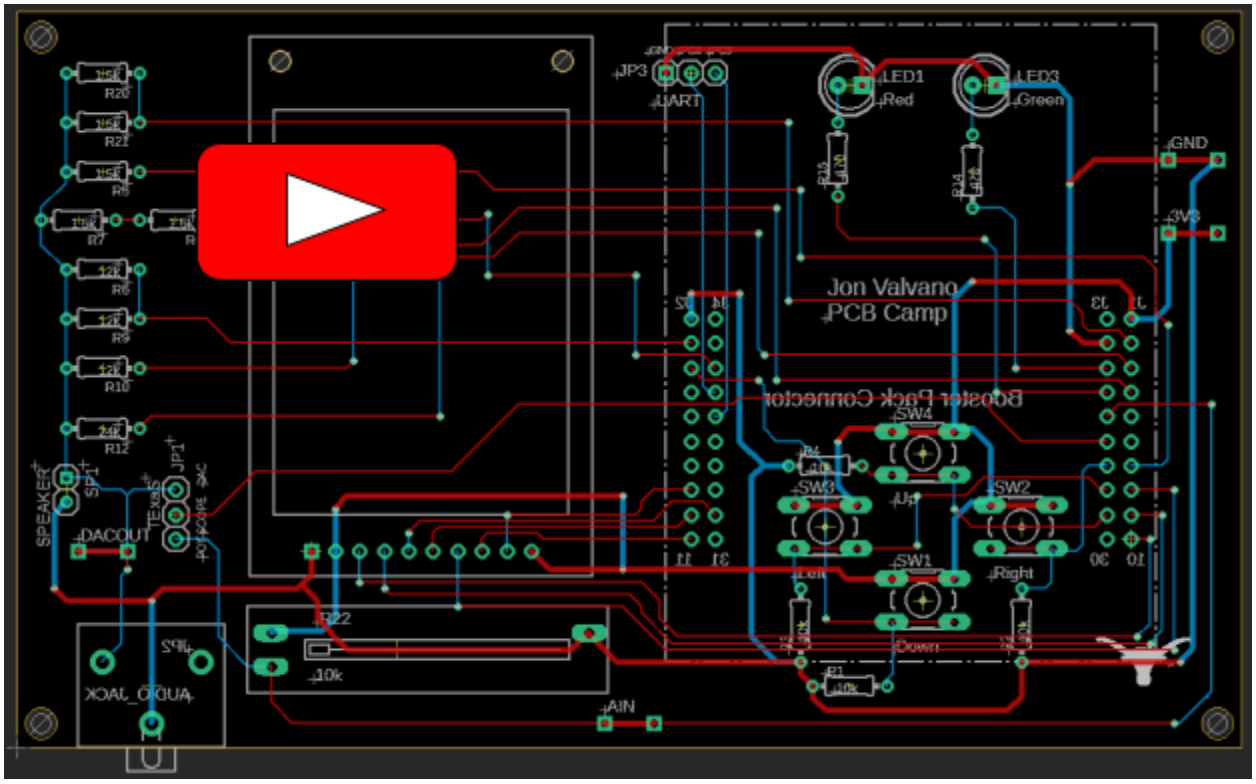


Video 14) Route power and ground [*https://youtu.be/L5Mzjvnqde4*](https://youtu.be/L5Mzjvnqde4)

Step 5) Route traces, add silk

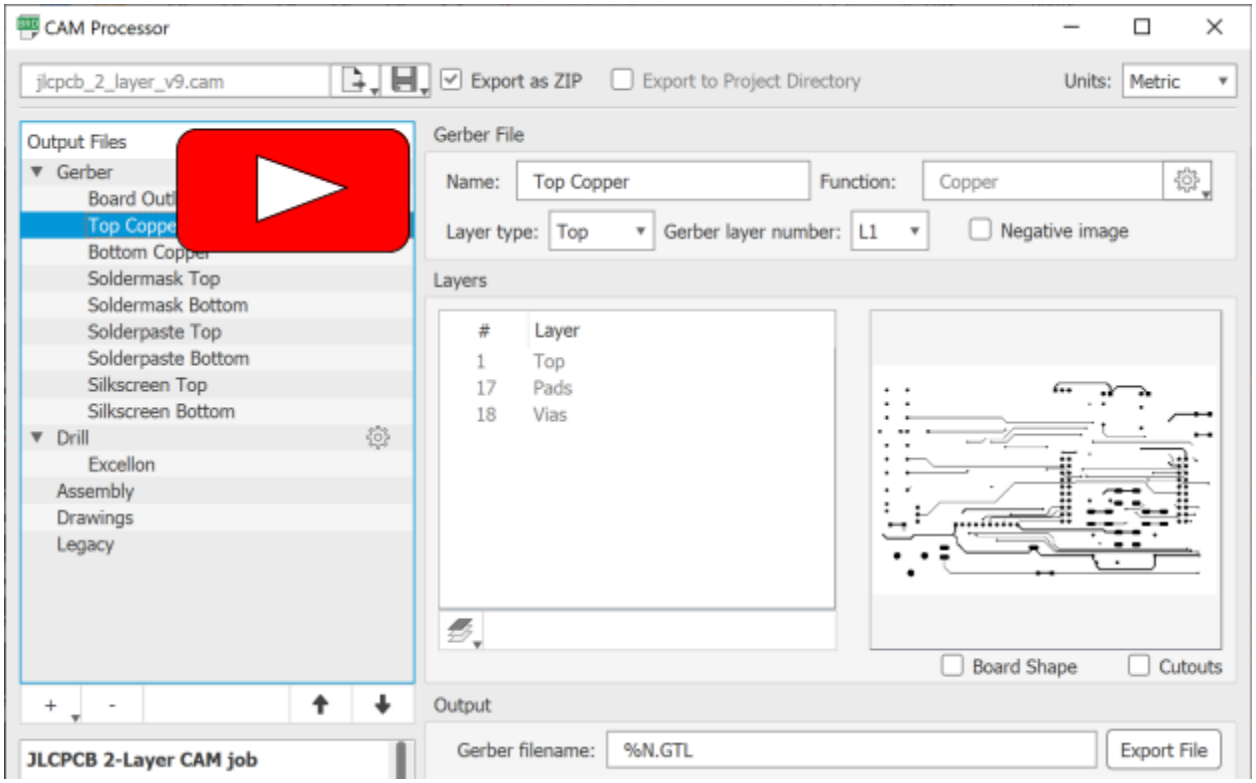
0) Route traces with 8mil traces,

1) Execute DRC



Video 15) Route traces https://youtu.be/ag79sGYwcyE

Step 6) Run the CAM processor
Get CAM file from manufacturer
https://support.jlcpcb.com/article/137-how-to-generate-gerber-and-drill-files-in-autodesk-eagle



Video 16) Run the CAM processor https://youtu.be/wVmbrwERY70

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
start layer is 21 (top silk)

Ordering PCBs (Valvano does this)
0) Upload PCB to https://oshpark.com/
https://docs.oshpark.com/services/two-layer/

1) Make gerbers and upload zip to JLCPCB: PCB Prototype & PCB Fabrication Manufacturer
Specifications for JLCPCB can be found at
https://jlcpcb.com/capabilities/Capabilities

2) make PCB width less than 100mm, upload PCB to https://dirtypcbs.com/store/pcbs

After camp is over
Valvano orders green PCBs from JLCPCB
Students get PCB from Valvano
Students get Parts

JV will provide two 2by20 headers, two 1by3 header, two 2-pin jumper, 4 standoff
Students use parts they already have (ask TA for second bag of parts)

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 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 two good pins to the PCB

PCB production when you want to make your own PCBs

Oshpark https://oshpark.com/
3 copies, purple, less than 2 weeks with regular shipping
1) Pass DRC on Eagle PCB
2) Upload Eagle BRD file to Oshpark

JLCPCB https://jlcpcb.com
5 copies, cool color options, less than 2 weeks with expensive shipping
1) Pass DRC on Eagle PCB
2) Put **jlcpcb_2_layer_v9.cam** file in
Users\your_user_name\Documents\EAGLE\cam folder
https://support.jlcpcb.com/article/22-how-to-generate-the-gerber-files
3) Create Gerber files,
4) Upload Gerber.zip file to JCLPCB

DirtyPCB https://dirtypcbs.com/store/pcbs
10 copies, cool color options, less than 100mm each side, lots of shipping options
1) Pass DRC on Eagle PCB
2) Upload Eagle BRD file to DirtyPCB