Title: Sp21 Spring Break Camp, Printed Circuit Board Design

Instructor: Jonathan Valvano valvano@mail.utexas.edu
Class: MTWTF, 10-11am, March 15-19
Office Hours MTWTF, 11-noon, March 15-19
Room: Zoom link on EE319K/EE319H Canvas zoom page
EE319K/EE319H MWF7-8pm office hours see EE319K Canvas syllabus page
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.

Overview: This hands-on course will walk through the process of PCB design, layout, construction, and test. Each team of two or three 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 10am, teams of 2 or 3 will be formed and the team will create one PCB design. Teams who complete their PCB layout by Friday 3/19 10am will have their board manufactured, making three 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.

Installations prior to class on Monday:
Option A) Register a student account: watch https://youtu.be/KWYJ02tlG4I
    https://www.autodesk.com/education/free-software/eagle
Install latest version of educational version Eagle (do not spend any money)
    Download and install **Autodesk_EAGLE_9.6.2_English_Win_64bit.exe**
    https://www.autodesk.com/products/eagle/
Option B) Install latest version of free/limited version Eagle (do not spend any money)

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

Bookmark these two tutorials prior to class on Monday:
        https://learn.sparkfun.com/tutorials/using-eagle-schematic
        https://learn.sparkfun.com/tutorials/using-eagle-board-layout

Computer work prior to class Monday: https://youtu.be/i4pFbAbk2hY
Create a folder called **Library** on your computer and copy this file into it
        https://www.dropbox.com/s/admu7w03xiaplje/EE445L.lbr?dl=1

Create a folder called **Design** on your computer and copy these two files into it
https://drive.google.com/drive/folders/1kzL1rcTRE0Ty1l9V_MoXqrGKaG6G--_i?usp=sharing
        EE319KLabKit.sch
        EE319KLabKit.brd
        Rename these two files with your EID (extensions are different, name must match)

Create a folder called **Datasheets** on your computer and copy these 8 files into it
https://drive.google.com/drive/folders/1-5_a7SrrhwGThlpLn_bKOAbU1F1jL4xG?usp=sharing

More Eagle instructions
http://www.instructables.com/id/Turn-your-EAGLE-schematic-into-a-PCB/

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

Monday Goals:
1) Overall plan for the week
        Monday-Tuesday schematic
        Wednesday-Thursday PCB
        Friday PCB design reviews
2) Installing Eagle (come to office hours 11am or 7pm if stuck)
3) Linking EE445L library
4) Overview of EE319K/EE319H Labs 6,7,8,9,10
5) 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
        Move
        Rotate
        Group-Move
        Copy

Delete
Add part: 3-pin header
6) 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
7) 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 PA2-5
Info on LEDs to document functionality
8) Build DAC/speaker interfaces
Move Jack, three 1.5k, three 12k, 24k, and 47k resistors together
Add net to complete circuit
Name net to connect to LaunchPad pins PB5-0

Tuesday Goals:
9) Name date on schematic
10) **Copy** testpoint, place near DAC/JACK, Info it to document functionality
11) Build OLED interface with 1.5k resistor
12) Build slide pot interface with TP
13) Build Lab 9 serial communication interface (optional)
14) Scope connections 3-pin header connecting PD3 to PD2 or PD3 to DACout
15) Optional stuff: Sparkfun https://www.sparkfun.com/products/9032
Add another library and connect joystick
16) Look at PCB view and develop a strategy for game layout

Wednesday Goals
17) Form teams for PCB layout; put your names here in this tab
https://docs.google.com/spreadsheets/d/1JlcZKAabRoVn5QLbOWWUosIQ9vHJUVuZPtwNsVPdiMY/edit#gid=1957258355
18) Upload SCH (finished) and BRD (not started) into this folder; name your files with your EID
You should have gotten an email invite to a box folder
19) Place parts to (show layers and grid again)
0) avoid overlaps;
1) make fun to play;
2) minimize airwire length;
3) reduce airwire crossings.
20) Check airwires, does it make sense?
21) Check nets, does it make sense?
22) Execute DRC (checking for overlaps)
23) Route ground with 20mil trace, make a star pattern (no circles)
Making via with middle button
Changing width
24) Execute DRC

25) Route +3.3V and +5V, make a star pattern (no circles)
26) Execute DRC

Thursdays Goals
17 revisit) Finalize teams for PCB layout; put your names here in this tab
https://docs.google.com/spreadsheets/d/1JlcZKAabRoVn5QLbOWWUosIQ9vHJUVuZPtwNsVPdiMY/edit#gid=1957258355
27) Print top
28) Print bottom, mirrored
29) glue to cardboard and arrange parts
30) Route signals with 8mil traces, make sure it makes sense
31) Execute DRC
32) Clean up silk, make sure everything as label
33) Put names on top and bottom
34) Execute DRC

Optional Image on Silk
        Paint BMP 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

Friday Goals
35) Upload your SCH/BRD files to shared box folder with EID_EID names
36) Review student PCB designs
37) Upload PCB to https://oshpark.com/
38) Make gerbers and upload zip to JLCPCB: PCB Prototype & PCB Fabrication Manufacturer
39) make PCB width less than 100mm, upload PCB to https://dirtypcbs.com/store/pcbs
41) Advanced Circuits www.4pcb.com
40) Due Monday 9am
Get PCB
Get Parts
        JV will mail two 2by20 headers, two 1by3 header, two 2-pin jumper, 4 standoff
   Students use parts they already have
Soldering
        Collect solder, soldering iron, cutter, solder paste, patience
        Collect all parts: use an ohmmeter to check resistor values
        https://www.instructables.com/Simple-PCB-soldering/
        https://www.youtube.com/watch?v=AqvHogekDI4
        We will set up soldering station in EER EE319K/EE319H/EE302 Lab EER 1.826
        One at a time in the Remove
        Solder the parts in reverse size order (resistors first and OLED last)

Assignment 1: Draw EE319K/EE319H circuits in Eagle for labs 6,7,8,9,10 in one SCH file

EE319k PIN DEFAULTS

Buttons
LEDs
3.3v
PE3-0   PA4-2
ADC0
-VBUS (5v)
PD2
Slide-pot (use PD2)
Joystick (use PE5, PE4)   x   y
MC
PC4 ← UIRx   UART1
PC5 → UITx   μc
PC   PA0   PA1   UART0
ADC1 - PD3
Diagnostic (TexasDisplay)
GND   PB6-0   PD1-0
I2C   LCD
DAC
0v

Expand your Lab 3 circuits to have four switches and three LEDs

+3.3V
+3V3

U1e
BoosterPack
QIP

PE5	6	
PE4	5	
PE3	29	
PE2	28	
PE1	27	
PE0	18	

3   4
SW3
NO switch
1   2
PE2
PE1

R19
470

R1
10k

D1
LEDT1.75
Red

GND

GND

Figure 3.8. PCB Artist drawing showing Port E, 3.3V power, a 10kΩ resistor, a switch,, an LED, a 470Ω resistor, and ground.

Draw a 6-bit DAC using three 1.5k, three 12k, one 24k and and one 47k resistor.

TM4C123

Bit3 1.5k

Bit2 3k

Bit1 6k

Bit0 12k

V_{out}

I_{out}

Speaker
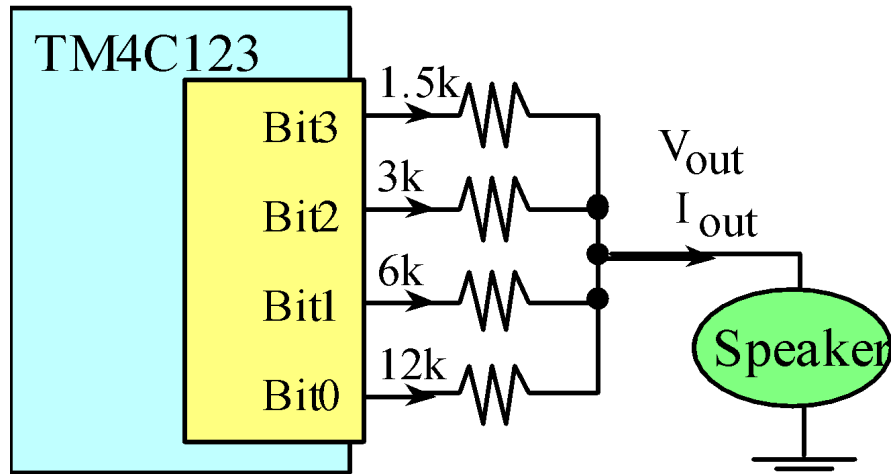
Connect the OLED to the microcontroller as described in Lab 7.

TM4C123

3.3V

1.5k

SCL PD0

SDA PD1

GND

VCC

SSD1306
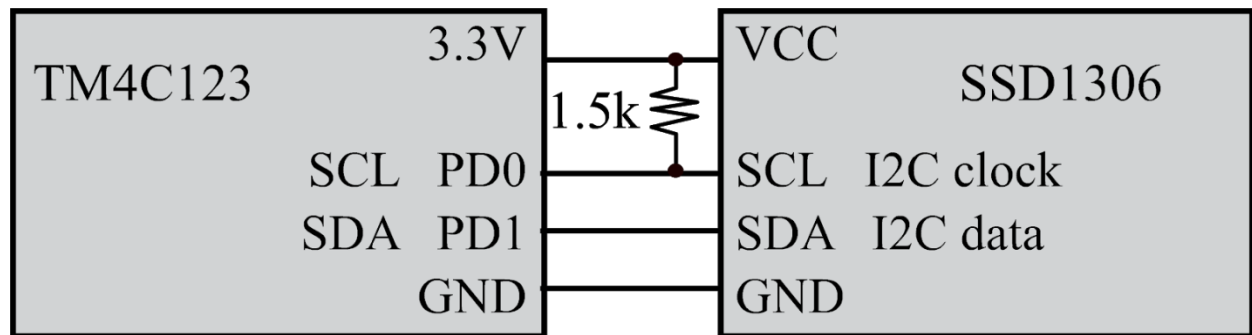
SCL I2C clock

SDA I2C data

GND

Figure 7.2. Interface connections for the SSD1306.

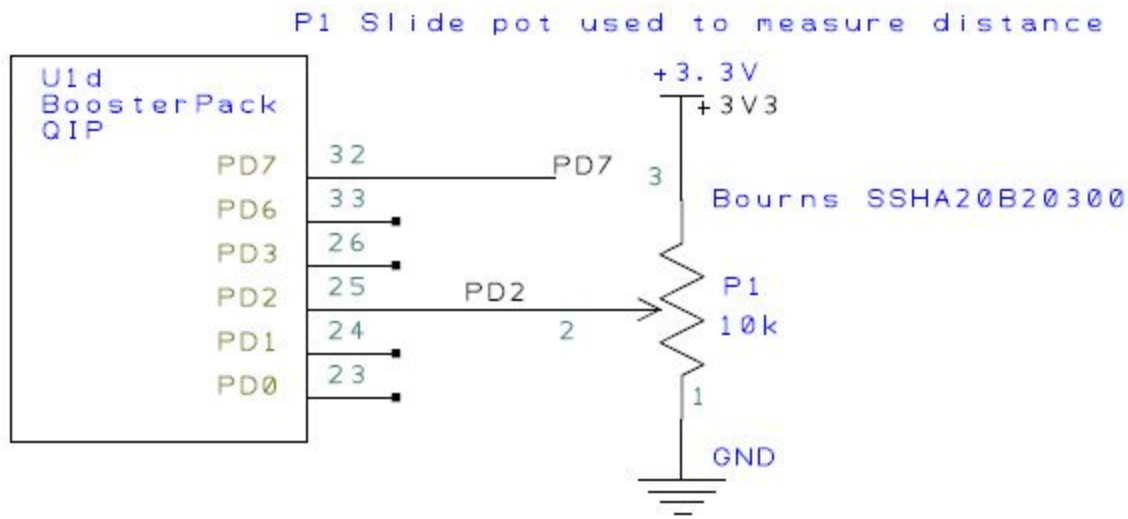Connect the Slidepoit to the microcontroller as described in Lab 8.

Figure 8.1. Possible circuit to interface the sensor (use your ohmmeter on the sensor to find pin numbers 1,2,3).

To use the TExaS oscilloscope, also connect a testpoint to PD3, allowing you to connect PD3 to either DACOUT or to PD2 (slide pot in)

Lab 9 does not need any circuits. Please avoid using PC4 and PC5. However as an option, you could 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.

Lab 10 uses Labs 3,6,7,8,9 into one project

Avoid PB7 because it is connected to PD1. Avoid PB6 because it is connected to PD0.

Assignment 2: Layout EE319K/EE319H circuits in Eagle for labs 6,7,8,9,10 in one PCB file less than 10 square inches. Specifications for OSHpark can be found at
https://docs.oshpark.com/services/two-layer/

Specifications for JLCPCB can be found at
https://jlcpcb.com/capabilities/Capabilities

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.

PCB production
Oshpark https://oshpark.com/
1) Pass DRC on Eagle PCB
2) Upload eagle PCB file to Oshpark

JLCPCB https://jlcpcb.com

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