ELECTRICAL ENGINEERING DEPARTMENT

EE143   California Polytechnic State University   Lab #4

PCB Design of Continuity-Tester

VIDEOS:

Experiment #5 Playlist at
https://www.youtube.com/playlist?list=PLkooZoxYRwMjdfxTdE9X5H-TBKhJwLsY4

PRELAB:

Watch Eagle tutorial videos on the experiment #5 playlist and "Goggle" for other Eagle videos as needed. Become familiar with the tools and functions of Eagle as you will be asked to design a PCB during lab.

NOTE: I STRONGLY RECOMMEND NOT TO INCLUDE A GROUND PLANE OR COPPER POUR, AS JEFFERY BLUM SUGGESTS, SINCE FOR THE CONYINUITY-TESTER IT IS NOT NECESSARY.

PURPOSE:

- To learn how to use PCB software to create a schematic and design a PCB.

This experiment relates to the following **course learning objectives** of the course:

1. Ability to simulate and design a PCB.
2. Ability to relate practical laboratory results with lecture theory.

LAB EQUIPMENT:

None

STUDENT PROVIDED EQUIPMENT:

Highlighter (manually check that layout traces correspond to schematic connections)

EXPERIMENTAL SECTION:

1) Continuity-Tester PCB Design

BACKGROUND:

PCB Design (before reading this background section watch Jeffrey Blum's Eagle tutorials, link on 1st pg.)

PCB design is accomplished using CAD (Computer Aided Design) software such as Eagle.
There are many different vendors of PCB design software, but Eagle is one of the most popular with hobbyist and professionals alike. Regardless of the vendor software used, there are three basic steps to designing a manufactured PCB; i.e., physical board.

The first step is schematic entry. In this lab, you will enter your DAC design that you simulated into Eagle following the instructions in Jeffrey Blum's first video. After your DAC schematic has been entered, with the simple click of a button, the CAD software will translate your schematic connections along with component "footprints" (component physical dimensions) into nets (lines that electrically connect the components) and geometric shapes based on the components footprints.

Now the real fun begins, as you are ready to design your PCB by placing the components with-in an enclosed area that represents the size of your PCB. For your DAC design you are limited to a 1" x 1" square board as to limit the cost of manufacture. Keep in mind that since you will be using all surface-mount components, you can place components on both sides of the board and they can be directly opposite each other since surface-mount components do not have leads that penetrate the board.

After you have placed components on your board, the next step is to "route" the board. Routing is turning the nets which are "logical" connections based on your schematic into actual physical connections (copper traces on the manufactured board). As shown in Jeffrey Blum's second video, Eagle has an auto-route feature that will route the board for you. You can also manually route the board. Typically, basic designs such as your DAC can be auto-routed successfully. More complex designs usually require at least some manual routing as the software is not perfect. I recommend to auto-route your DAC design and then modify manually as desired.

After routing, whether it was done by auto-route or manually or a combination of both, DRC (Design Rules Check) must be performed. DRC will check that your design is ready for manufacture by checking whether your trace widths, trace separations / clearance and drill hole diameters pass the tolerances that were inputted. For your design, use 10 mils for trace width, 10 mils for trace separation / clearance and 20 mils for drill hole diameter. Note: 1 mil equals 1 thousandth of an inch; 0.001 inch.

In addition, and this is **very important**, before proceeding to the last step, uploading .BRD file to Oshpark.com, check to make sure your PCB routes agree with your schematic connections. This is a final check to make sure the copper traces of your manufactured board will connect the components as shown on the schematic. If you find a discrepancy, edit your PCB design, and redo DRC.

A final comment before moving onto the last step, getting your board manufactured, Jeffrey Blum shows how to create a copper pour in his second video. Copper pours are necessary for PCB designs when there are components that dissipate a lot of heat or as a ground plane for RF (radio frequency) applications.. The copper pour acts as a heat sink. A copper pore is not necessary for your DAC and I would recommend not having one. As a first design I believe it would be best to keep it as simple as possible.

PROCEDURE:

NOTE: Each student designs their own PCB and has their own board manufactured.

a) Open a new schematic in Eagle and name it "Continuity_Tester_LastName_FirstInitial". Create your Continuity-Tester schematic in Eagle using the following components:

NOTE: In order for access to libraries connect to internet.

i. Through-hole resistors from the SparkFun Electronics library, Resistor Axial-0.3.

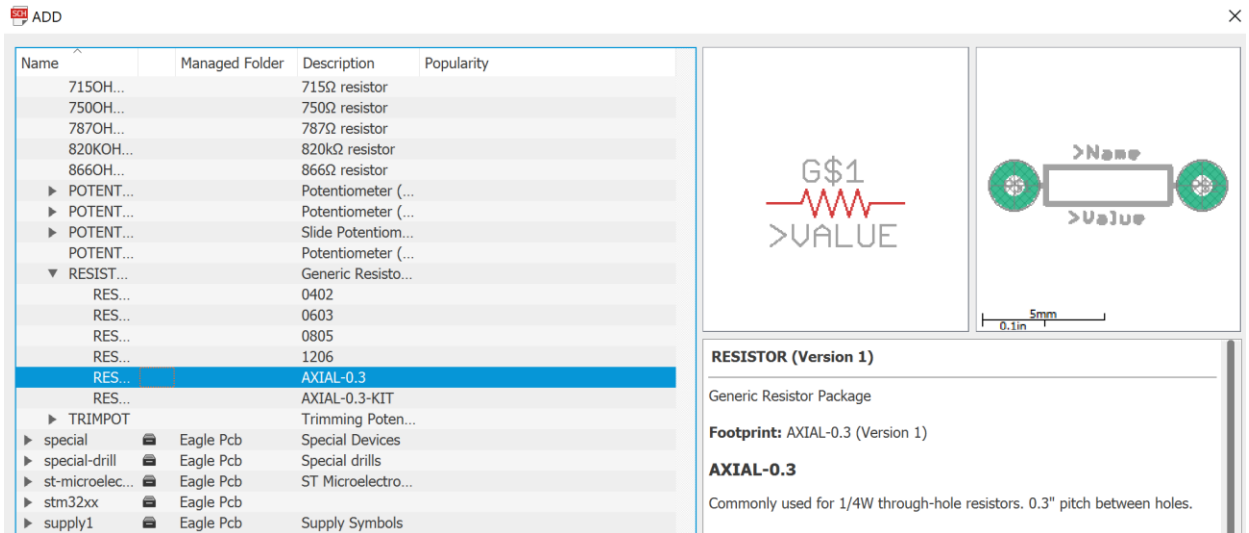Below shows the library and then the components within library.

Name		Managed Library	Description
SparkFun E...		SparkFun Electr...	SparkFun Electr...
SparkFun-R...		SparkFun Electr...	SparkFun Resis...
special		Eagle Pcb	Special Devices

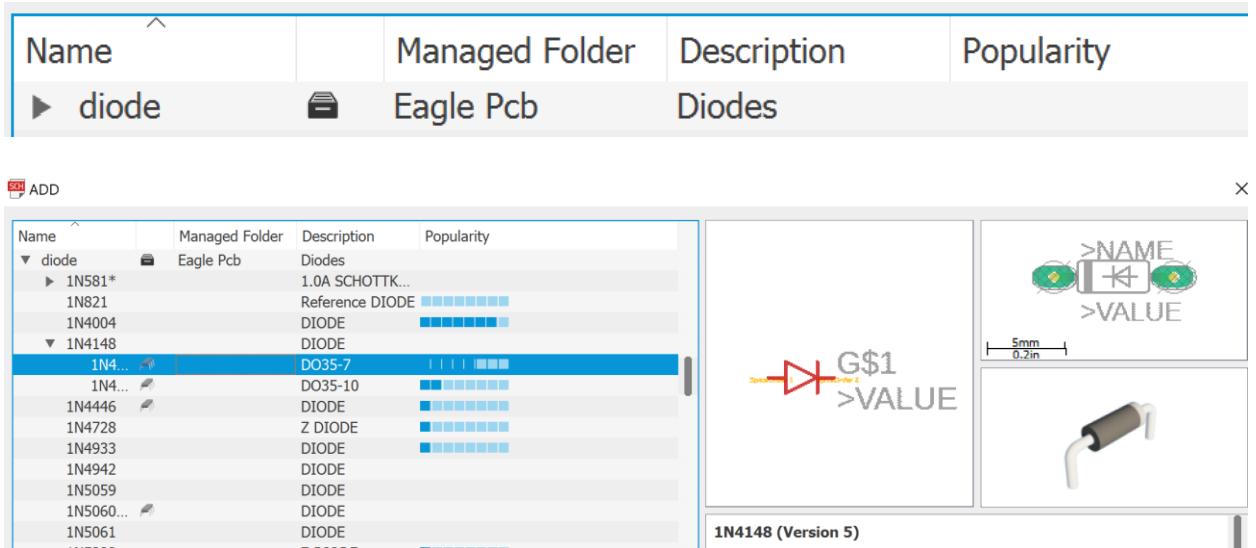ii.   LM311 comparator op-amp (through-hole) DIL08 which can be found in the Linear library.
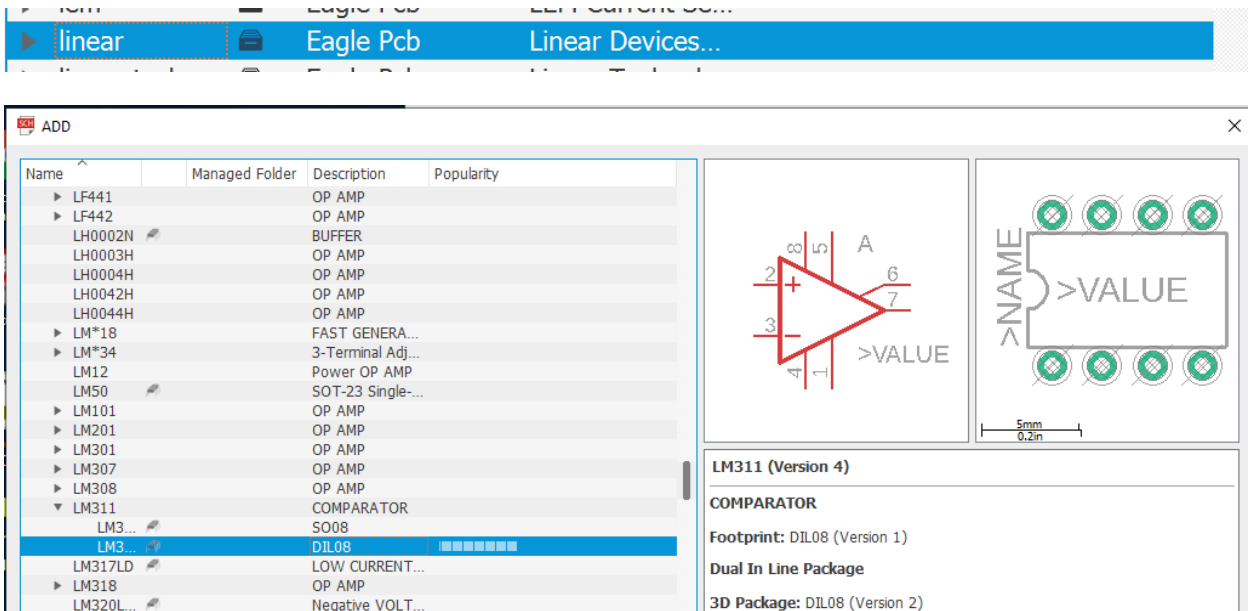
iii.   Two connection headers (two pins each) from the SparkFun connectors library.

Note: May have to add library with Library Manager.
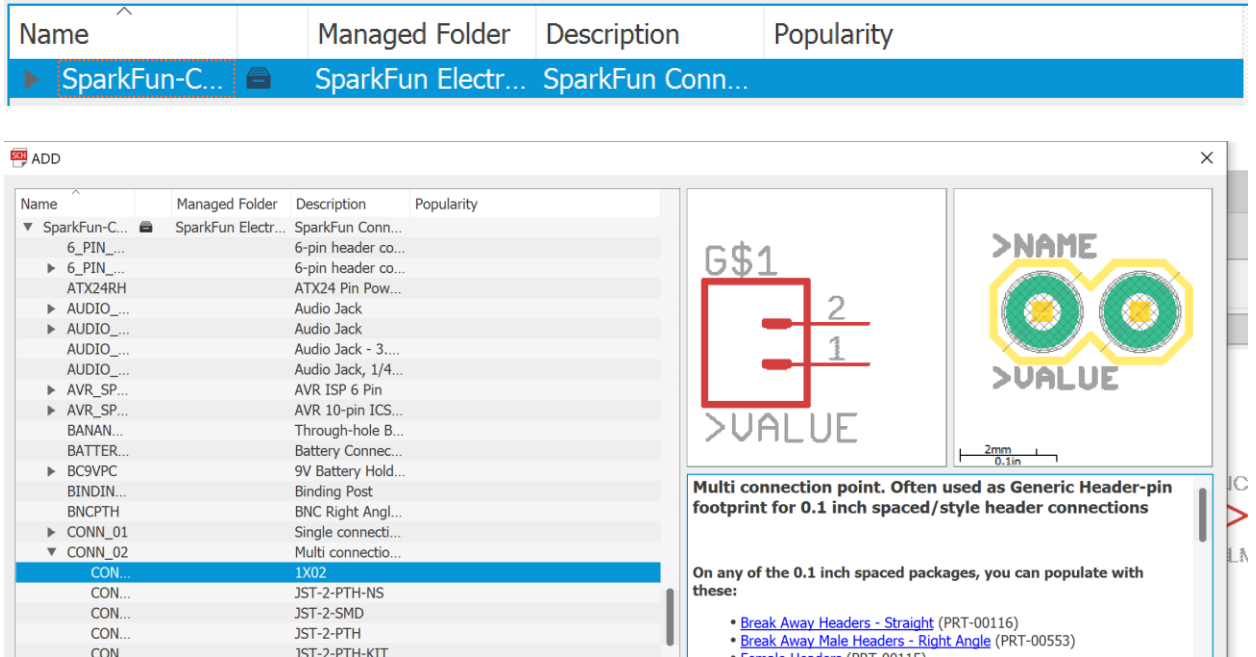
- Use headers to attach to nodes X and Y, 5V, GND. Use one two pin header for X and Y; use other two pin header for 5V and GND.

iv.   1N4148 diode from the Eagle PCB library.

v.   LED 5mm from the Eagle PCB library.

vi.   Vcc and GND (ground) from the Eagle PCB Supply Symbols library.

b) From the schematic, generate your PCB board layout.

i.   Make sure the actual size of the board is **1"x1"** (you'll have to spend more than $5 if it is larger) and move all your components to fit in this space. **NOTE: To check board dimensions, place cursor on the top right corner of board. Board length and width are displayed at the top left.**

ii. Use the same dimensions as Jeremy Blum used for the design rule check: Width, 10 mil, Drill 20 mil, Clearance 10 mil.

iii. **Auto-route** your design, minimize the number of vias and make sure design passes design rule check.

iii. **Print out a hardcopy of your board layout and check each trace against schematic to make sure all connections on board agree with schematic nodes. To do this, highlight the trace in the schematic and then highlight the same trace on the layout, until all traces have been verified.**

iv. Label all connector pins (GND, 5V, X and Y) so you know what pin is supposed to connect to what. You can put these text labels on the tPlace or bPlace layers so they'll be printed in the silk screen.

v.   Label the values of your resistors and pin 1 of the op-amp on the tPlace or bPlace layers.

vi. Put your name somewhere on the board on the tPlace or bPlace layers.

vii. Do <u>not</u> put text (labels, values, name, etc.) too close to PCB edges. If text is close to an edge, it may be cut-off during silk screening.

viii. Save your .BRD file. It <u>must</u> have the name Continuity_Tester_LastName_FirstInitial.

ix. Upload your .BRD file to OshPark (www.oshpark.com), verify that all of the board layers look correct (i.e. have all the labels showing, and all the pads in the right place for the top and bottom layers, etc.) and then order your board. If something looks wrong, you have to fix your design and re-upload it. When you order your board, you can choose free shipping if you have 2 weeks

or more before the lab 8 meeting. If there less than 2 weeks, you'll have to pay "speedy manufacturing" and expedited shipping to get the boards in time.

DELIVERABLES:

1) **Provide a copy of the OshPark email order confirmation to the instructor.**