

ECE480 Design Team 3

Application Note

Ballistic Chronometer Semiconductor application and PCB board design

Xuming Zhang

March 26th 2014

Executive Summary:

This document will be providing an instruction of LED and Laser combination usage as a speed sensor and prototype of wiring. Based on the sponsor's (Michigan State university ECE shop) requirement, our design must be compatible with the commercial ballistic chronometer. The design must also be accurate with low cost and low power consumption. Also, it will briefly introduce PCB design by using Eagle software.

Introduction and background:

The main function of ballistic chronometer is detecting projectile speed. In order to obtain the precise speed, the ballistic chronometer will involve humidity, altitude of environment facts to optimize the velocity equation instead of using the basic equation ($\text{velocity} = \text{distance} / \text{time}$). There are several common applications with different methods to measure speed such as: radar measurement system, ultra sonic measurement system, laser sensor system. In our project design, we are using laser measurement system concept, then develop a new sensor system which is using LED as receiver and Laser diode as emitter.



Figure 1: 5mW 650nm Red Line Laser Diode



Figure 2: 5mm diameter Red LED

Sensor system:

The LEDs are aligned in series, Line Laser Diode is mounted above 12 inches of the LED line. After Line Laser Diode is supplied by power, the line laser diode will emit laser light to hit on the LEDs. LEDs will produce voltage as reserve bias diode. When the projectile passes the system, it will block the laser light then cause one or two LEDs voltage drop. Since the circuit of LEDs is connecting MSP430 Launch Pad, the microcontroller will keep monitoring the voltage of circuit.

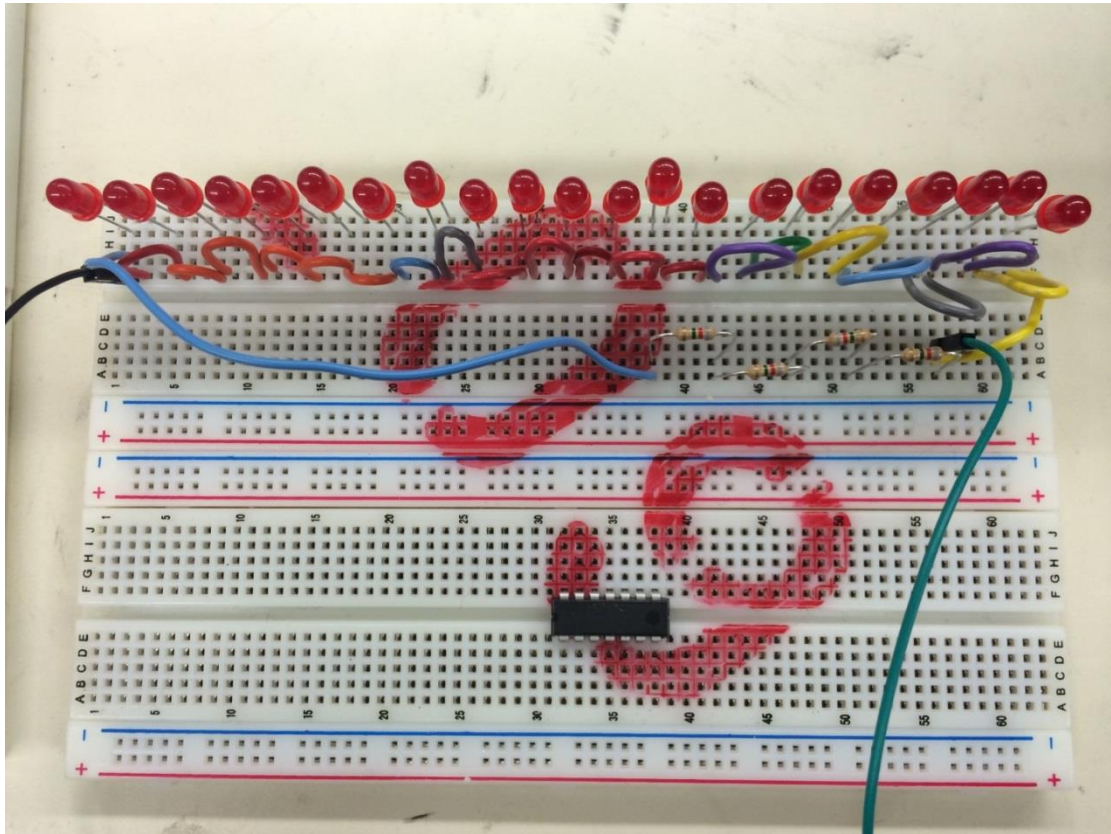


Figure 3: LEDs circuit prototype

Concept of LEDs:

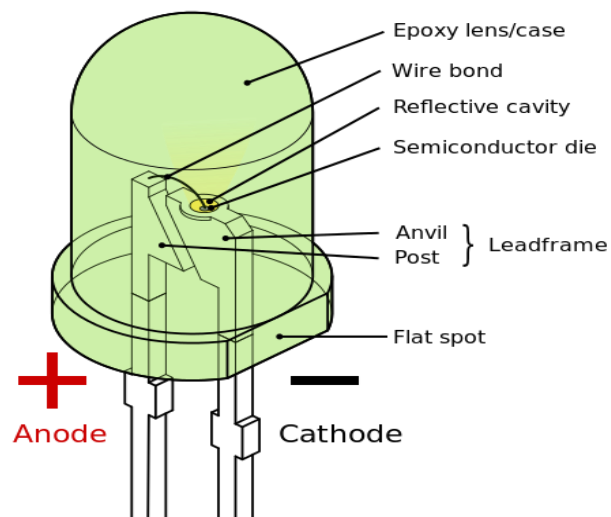


Figure 4: LED structure

Our purpose of using LEDs is to let semiconductor dead part to work as photodetector.

The characteristics of semiconductor will be demonstrated by the following figures:

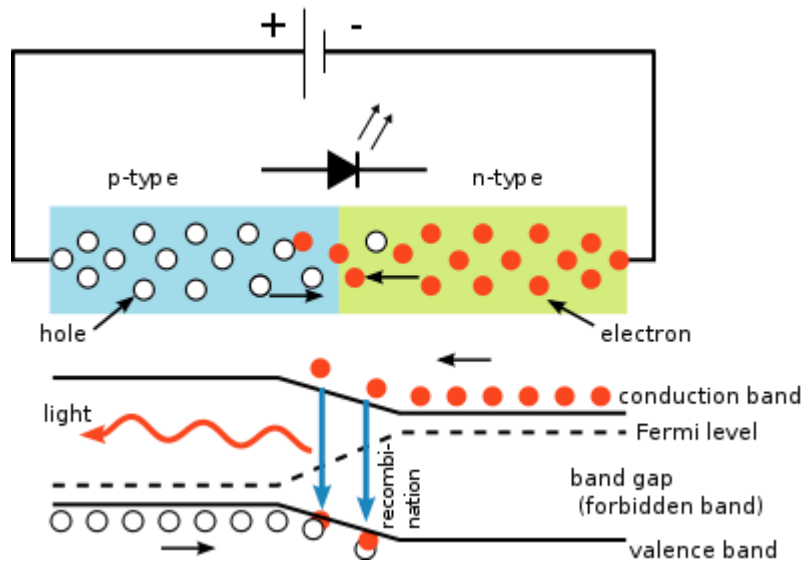


Figure5: basic concept of semiconductor

According to the figure, when power supply connects to the LED, it will release energy because the electron will fill the hole in P-type, where energy is released (emitted light). However, we are applying another characteristic of semiconductor.

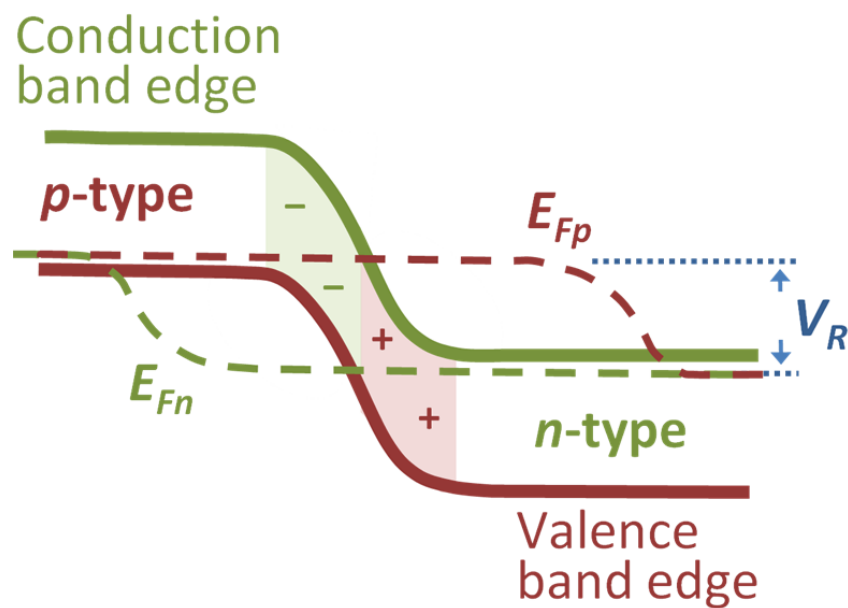


Figure 6: Conduction band demonstration

When the light from laser diode is hitting on the semiconductor, the electrons will move to the n-type then voltage will be produced.

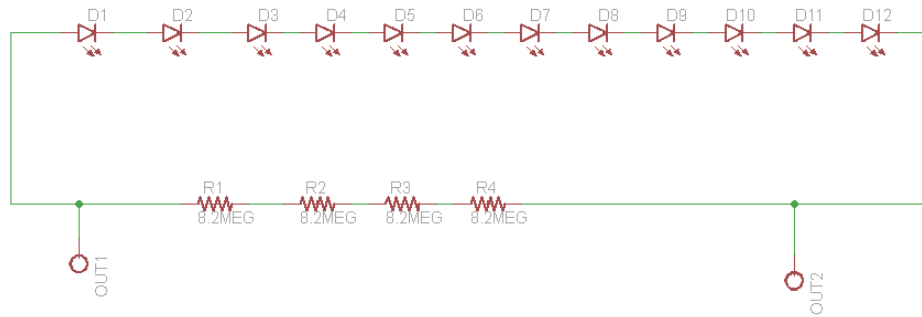


Figure 7: prototype of circuit design

Since the LEDs work as reverse bias diode, the total voltage is close to 75 millivolts. Our design added four 8.2MegaOhms resistors in parallel to drop the resistance in the circuit. Lower the resistance will make the voltage be more sensitive to be detected by microcontroller.

Printed circuit board design:

In order to install the circuit into the frame, our team decided to use PCB(printed circuit board) to replace the bread board in the prototype circuit designed. The Advantages of PCBs are: fixes the location of electronic parts, minimizes the miswiring and short-circuited wiring, and reduces the size of circuit board. In our project, the LEDs must be lined up straightly, PCB will keep the components stable. We are using single side layer PCB which costs lower price and reduces design error occurrence. Michigan State University provides student standard edition CadSoft Eagle PCB design software.

Design procedure:

Based on the circuit that our team built on breadboard, designing procedure will be done from schematic level design to layout design, and then extract the design file as an extended 274 Gerber format. As figure 7 shows, the prototype of circuit board is formed by four 8.2megaohms resistors, ten LEDs and two solder pad for output.

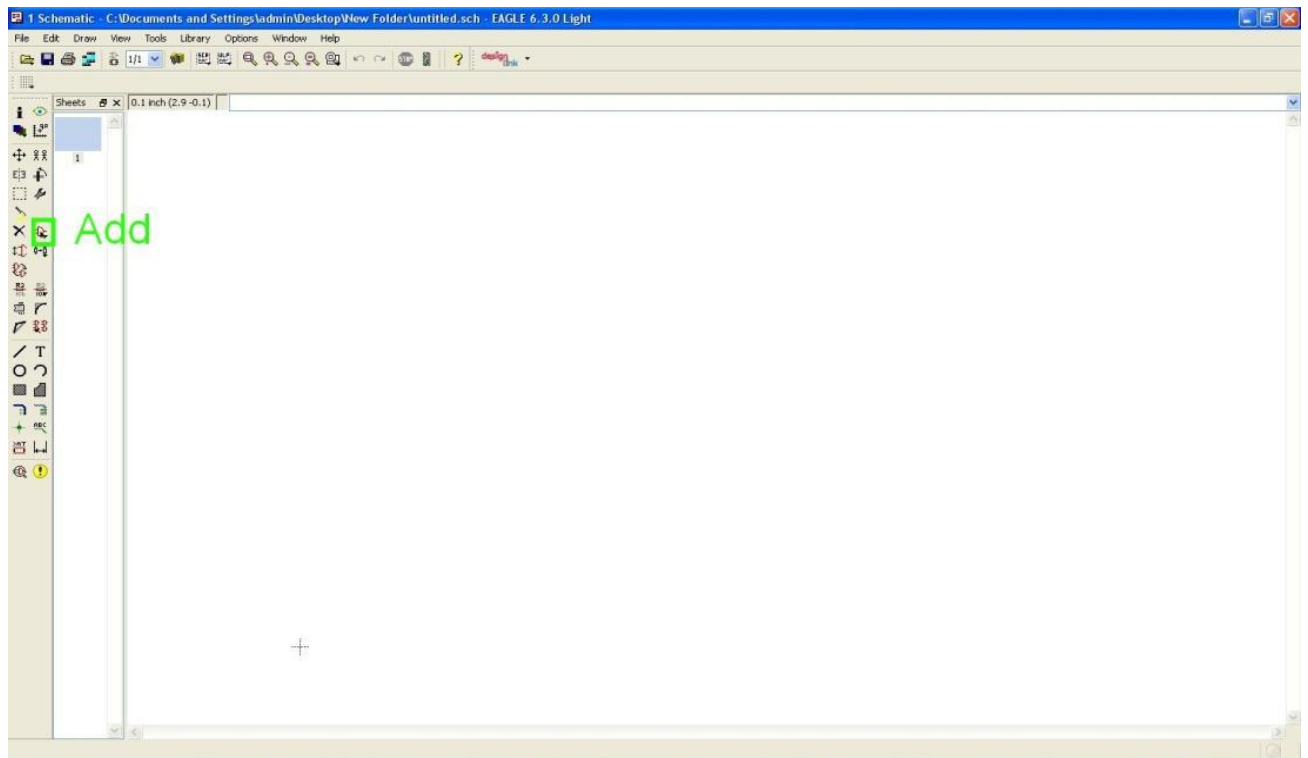


Figure 8: guide of starting schematic design

1. Run Eagle program. The Control Panel will be displayed. On the Control Panel, select File>New>Schematic. That will open a new schematic window. On the left of the schematic window, there is an add tool
2. This Add tool will add elements such as resistor, capacitor and ICs, all of which are very important in designing the PCB.
3. Select “wire” on the left of tool bar and wire components

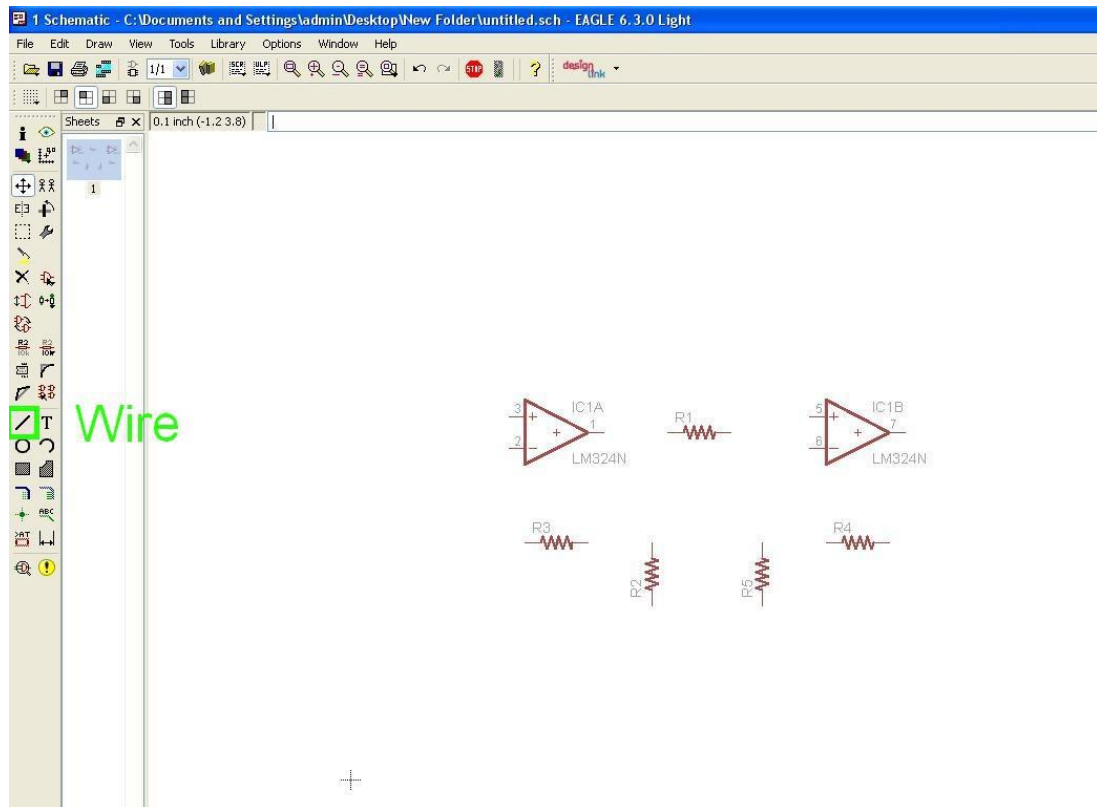


Figure9: guide of schematic design (wire selection)

4. Click “Board” icon on tool bar>place components>click”net” on the tool bar> connect components with net wire> save to a created folder with “.brd” format.

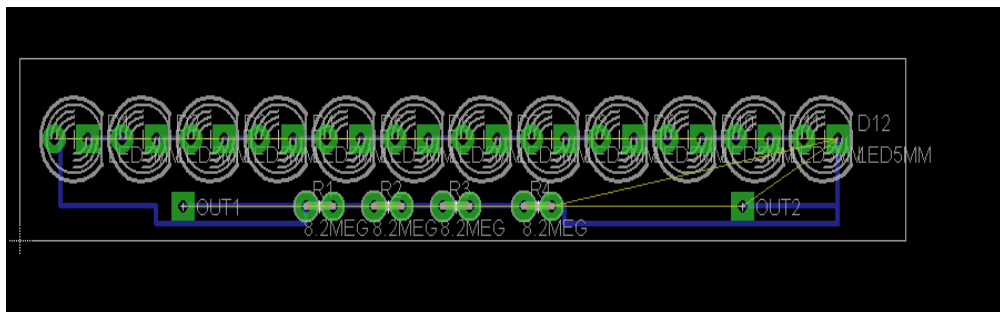


Figure 10: PCB layout design sample

5. Download “Eagle CAM Job File” > Unzip the file into the directory "Eagle/cam".
6. From the Eagle board editor (NOT the schematic editor), do "CAM Processor, File, Open, Job, and select "gerb274xECEShop.cam".

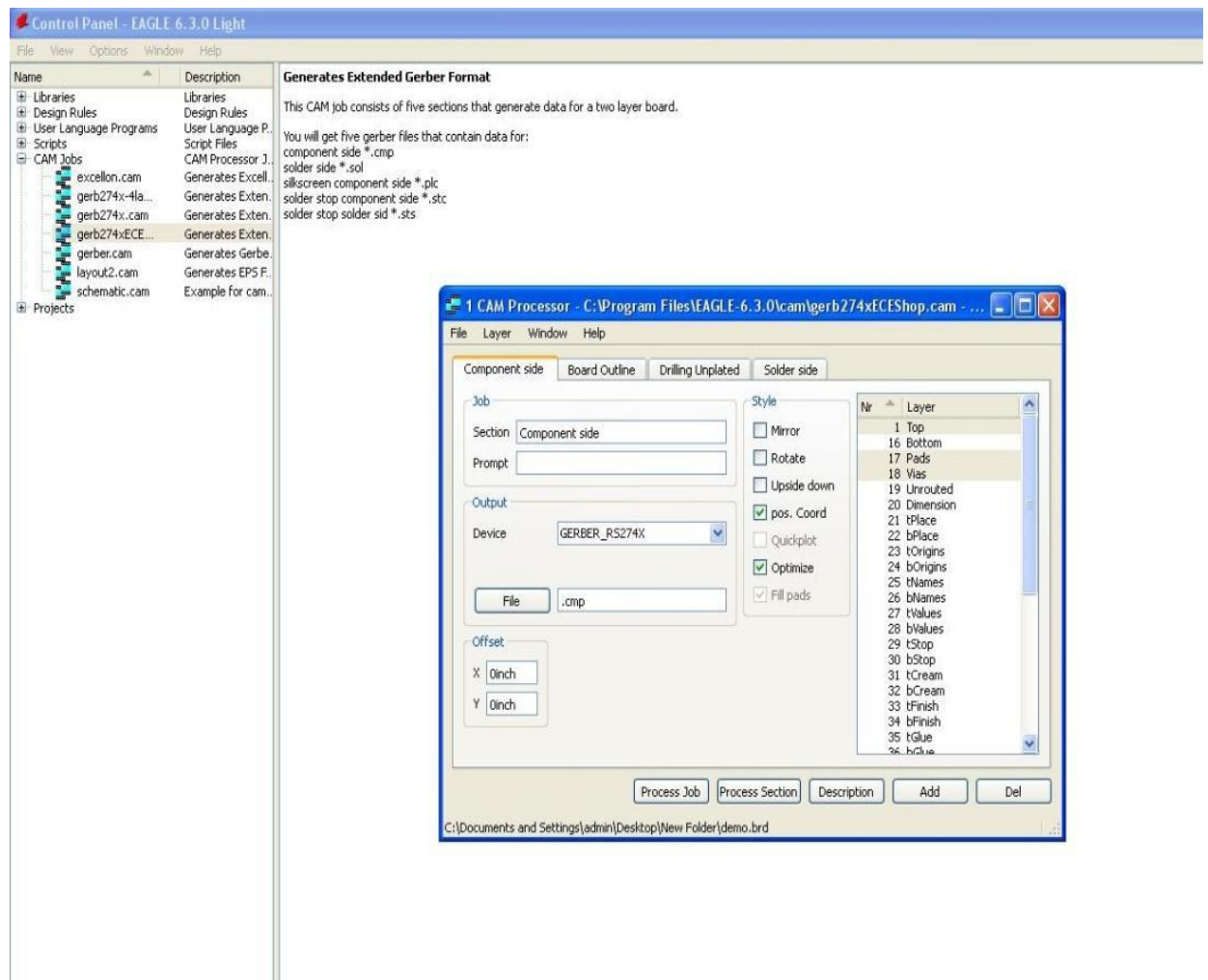


Figure 11: Eagle process job demonstration

7. After click “process job”, Eagle will generate five different extension files as .bol .cmp .gbr .gpi .sol
8. Send these files to eceshop@egr.msu.edu to fabricated

Discussion:

In PCB design, users have to be aware of design rule. “Check DRC” button (on the left tool bar) It will help users to detect errors in the design. It is very difficult to modify the circuit once the PCB is fabricated. Also, users have to calculate trace

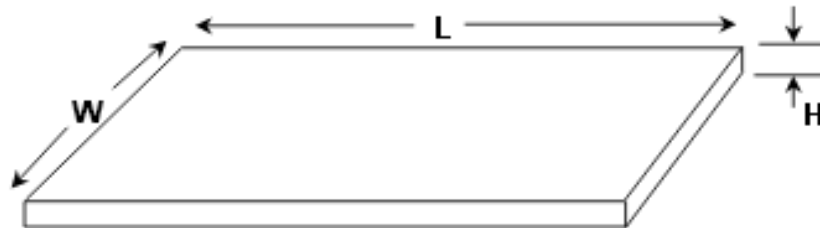
capacitance or inductance before fabrication.



L, R in mm

$$\text{WIRE INDUCTANCE} = 0.0002L \left[\ln \left(\frac{2L}{R} \right) - 0.75 \right] \mu\text{H}$$

EXAMPLE: 1cm of 0.5mm o.d. wire has an inductance of 7.26nH
(2R = 0.5mm, L = 1cm)



$$\text{STRIP INDUCTANCE} = 0.0002L \left[\ln \left(\frac{2L}{W+H} \right) + 0.2235 \left(\frac{W+H}{L} \right) + 0.5 \right] \mu\text{H}$$

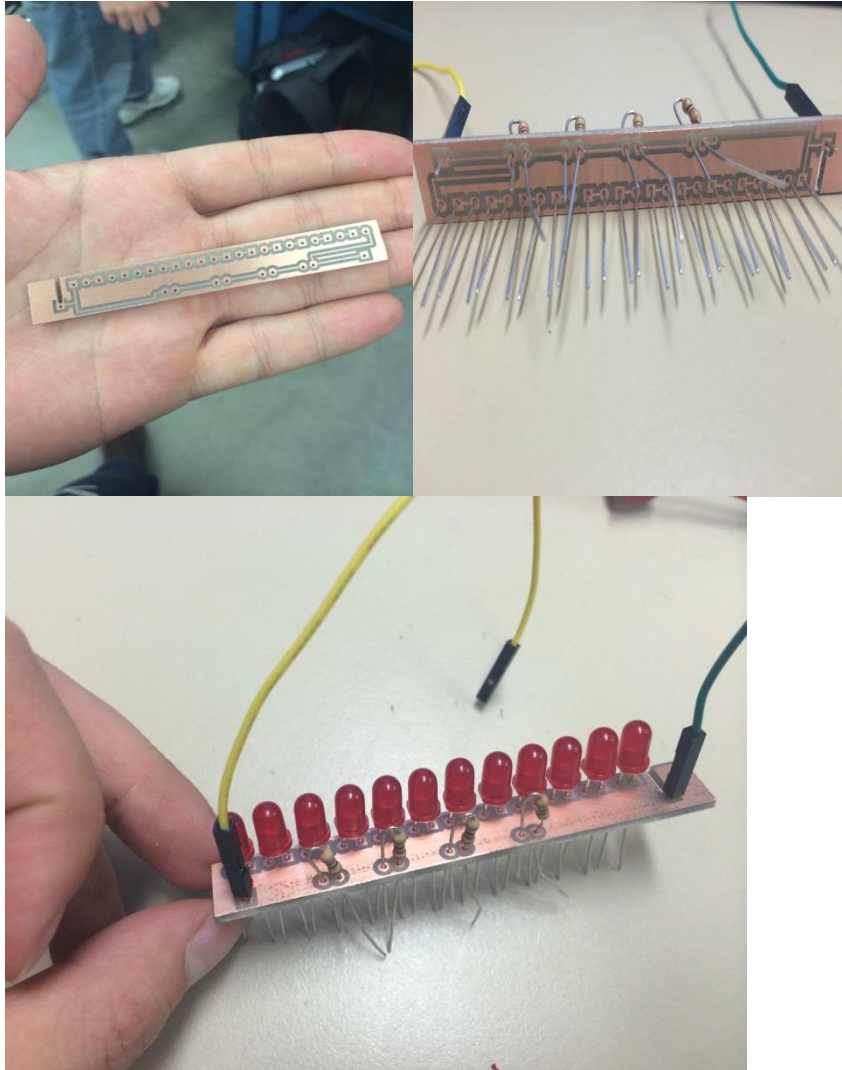
EXAMPLE: 1cm of 0.25 mm PC track has an inductance of 9.59 nH
(H = 0.038mm, W = 0.25mm, L = 1cm)

Figure 12: Inductance equations and samples

Conclusion:

This document briefly introduces our team design project sensor system concept and demonstrates the circuit. Also it is presenting specific steps of PCB design procedures. Also, the semiconductor conduction band will help the reader understand the semiconductor characters, and will show the structure of the LEDs with different methods to utilizing its characters.

Sample PCB of our project design:



References

1. Michgias state university (n.d.). *ECE Technical Services*. Retrieved March 26, 2014, from https://www.egr.msu.edu/eceshop/pcb/eagle_tutorial.php
2. Cheng zhang/Michigan State University (2012, December 15). Retrieved from <https://www.egr.msu.edu/eceshop/pcb/Application%20note%20-%20Chen%20Zhang-EAGLE-tutorial.pdf>

Michigan state university (n.d.). *ECE Technical Services*. Retrieved March 26, 2014, from https://www.egr.msu.edu/eceshop/pcb/eagle_tutorial.php
3. 6.27 *Optical Applications*. (n.d.). Retrieved from http://www.eng.fsu.edu/~dommelen/quantum/style_a/semicopt.html
4. Analog devices Retrieved from <http://www.analog.com/library/analogdialogue/archives/43-09/edch%2012%20pc%20issues.pdf>