Modern Circuit Board Design and Prototyping

Gas Detector

12/19/19

Final Report

Prof. David Westerfeld

Student: Roni Das

Objective: The goal of this project is to utilize acquired knowledge from Modern Circuit Board Design and Prototyping in servicing real-world application. The project is divided into several stages. Each stage reflects on the specification required for this project. The procedure for prototyping a Gas Sensor described below.

Concept: Many housefires are caused by natural gas leakage used for cooking. The goal is to detect the leak and notify the user about it. Natural Gas is naturally occurring compound primarily composed of Methane(CH_4), has small percentage of Butane and other combustible elements. Sensor MC105 used in this project is a gas sensor capable of sensing all the elements present in Natural Gas with precision. This project has potential to save property from house fires and save lives.

Model		MC105
Sensor Type		Catalytic Type
Standard Encapsulation		Plastic
Working voltage(V)		2.5±0.1
Working current(mA)		150±10
Sensitivity (mV)	1% CH4	20～50
	1% C3H8	30～70
Linearity		≤5%
Measuring range(%LEL)		0～100
Response Time (90%)		≤10s
Recovery Time (90%)		≤30s
Working Environment		-40～+70℃ <95%RH
Storage Environment		-20～+70℃ <95%RH

Figure 1: a) MC105 Catalytic Gas Sensor. b) Parameters and electrical characteristic of MC105.

Engineering Specification: There are few design requirements for this project needs to be considered for prototyping. Gas sensor, MC105, provides an analog output voltage in response to presence of natural gas on its beads. A Microcontroller unit is therefore used to convert analog output to digital values and display these values based on the concentration level of gas.

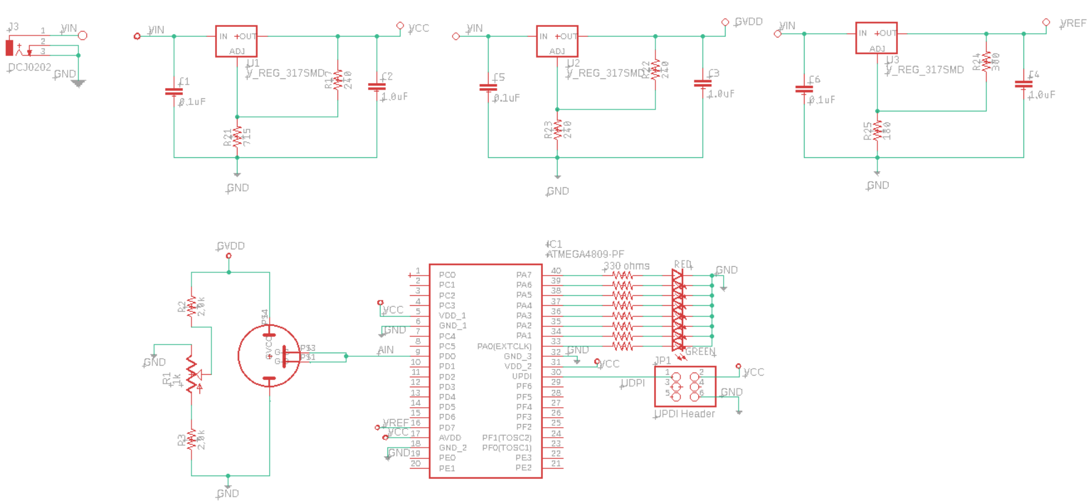

Figure 2: Schematic of Top-Level.

Schematic : Top-Level schematic describes the design elements required to implement this project. MC105 requires an accompanying resistive network in bridge configuration, depicted on bottom left in Fig.2. Output of MC105 is connected to ATMega4890(MCU); which shows results in PORTA using corresponding LEDS. Three adjustable linear voltage regulators, on top of Fig.2, were used to power corresponding devices. Power Jack, top left of schematic, is used to add battery support to the system. Footprints for all components but one, MC105, were available in Eagle. Therefore, a new library part is created for gas sensor.

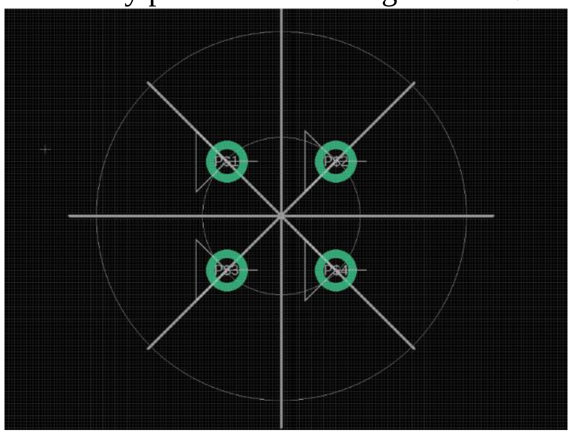

Figure 3: Footprint for Gas Sensor created using Eagle Library Files.

Board: Using Eagle's Switch to Board command, schematic in Fig.2 is converted to CAD board. The dimension of this board is set to 90mm x 90mm, allowing ample room for all the component in design. There are a total of 38 components, 33 of which is SMD and rest is through hole parts.

Figure 4: Designed Board for the system.

Board shown in Fig.4, is 2-layers CAD design showing positing of all necessary components. Top layer is used to route signals and power. Bottom layer is used as ground plane for easier routing options. On bottom right corner in Fig.4 depicts 6-pin header, which is used to program MCU, is optional.

Case: All the necessary measurements of PCB are considered before modeling the enclosure. The enclosure size is 98mm x 98mm, with 3mm of wall-thickness. Since the PCB is 90mm x 90mm in size, which allows 1mm of space (PCB to inner wall) inside the enclosure to be used in fillet, increasing support to system.

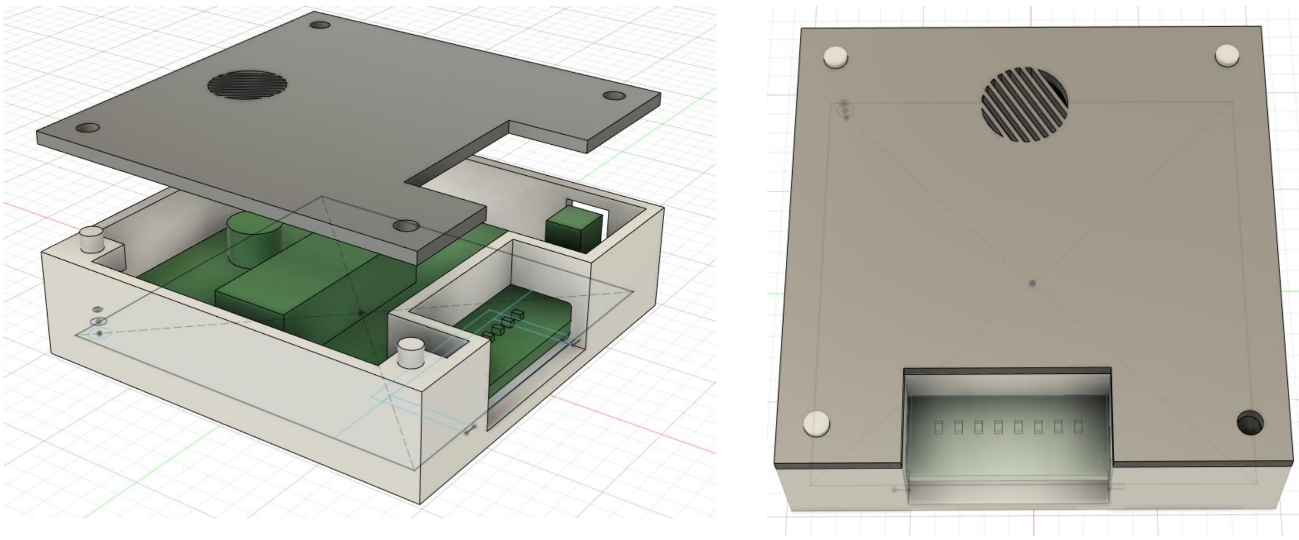

Figure 5: 3D model of enclosure designed using Fusion360.

Prototype: All the SMD and through hole components were hand soldered in lab. The standard 0805 package were used for passive elements such as resistors and capacitors. LEDs Package size is 0603.

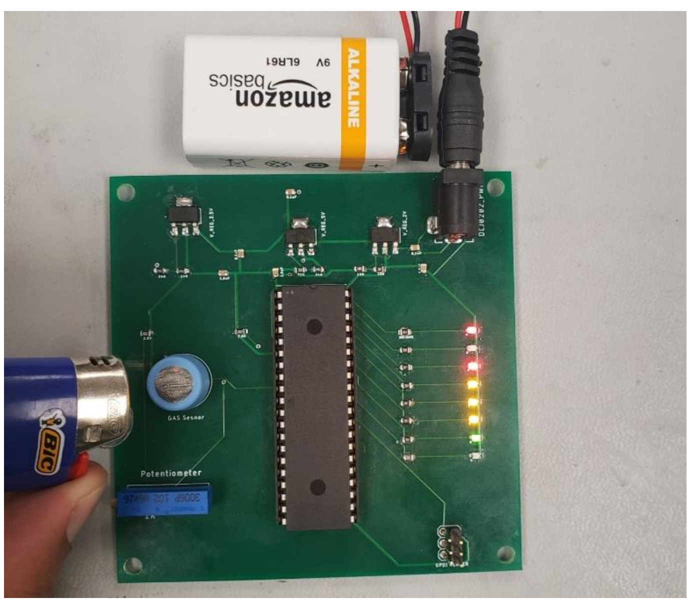

Figure 6: Final prototype of Board.

Figure above shows the completed board. Stimulus for gas sensor, MC105, is provided using generic gas lighter. The system response is displayed using LEDs bank on the right. Figure below shows enclosure printed for this device. The process of printing enclosure, Fig.7, took 3 hours for Base and 1 hour for Top unit.

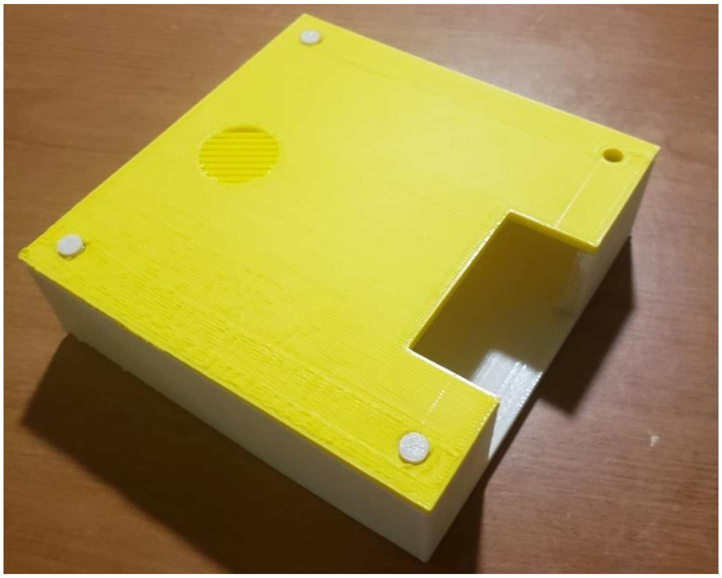

Figure 7: Enclosure for device printed using 3d technology.

The process of printing enclosure, Fig.7, took 3 hours for Base and 1 hour for Top unit.

Supplier: The board was ordered from PCBway. The process of placing an FR4 board order is simple and straight forward from company's website. Initially, board is scheduled to arrive within 7 days' time period using DHL delivery option. Extra shipping cost is charged for fast delivery service. However, it took extra 4 days from original delivery dates to receive this board from PCBway.

Results : Prototyping a complete system is fun and enjoyable experience. However, no engineering job is completely error free. During hand soldering of LEDs on the board, too much heat was placed on the pads of the bottom LED. This caused the pad to lose free from the board. Although printing enclosure was an error free process; due to design constraints the board did not fit in this case. Additional carving after print service was needed to accomplish this task.

Suggestions: This class serves as an example for how all engineering classes should be designed. The complete process of designing and prototyping a real-world system was fun and enjoyable experience. At no time students were overburdened with paper works and pressured to get it right every time. All the components for circuit design were also available in lab. There are no changes needed in improving this class.