6.0 Breadboard/Independent PCB

6.1 MAIDS Schematics

Development of the MAIDS breadboard prototype and the custom-made PCB board began with the schematics diagram. The schematic will serve as a blueprint for laying out the traces and placing the components on the PCB board. The MAIDS schematic diagram (also called wiring diagrams or circuit diagrams) is a representation of the significant components of the system using standardized symbols and lines. That is to say, its schematics show how the different components (LED module, sound and motion sensors) of the circuit are connected. In the schematic diagram, lines represent connecting wires, while other elements like the LED module, and the motion and sound sensors are represented by standardized symbols called electrical schematic symbols. In addition, is it worth noting that the MAIDS schematic diagrams is useful to explain the general way that its electronic system works.

MAIDS' schematic diagram was developed using the University of Applied Sciences Potsdam, English language, Fritzing software version 0.9.4. Fritzing is an open-source CAD (Computer Aided Design) software used in the design of electronics hardware, breadboard prototypes and PCB board circuits. Fritzing installation requires one of the following operating systems:

1. Windows 10 (Windows 7 is reported to work, too)
2. Mac - OSX 10.14 and up, though 10.13 might work too.
3. Linux - a fairly recent Linux distro with libc >= 2.6

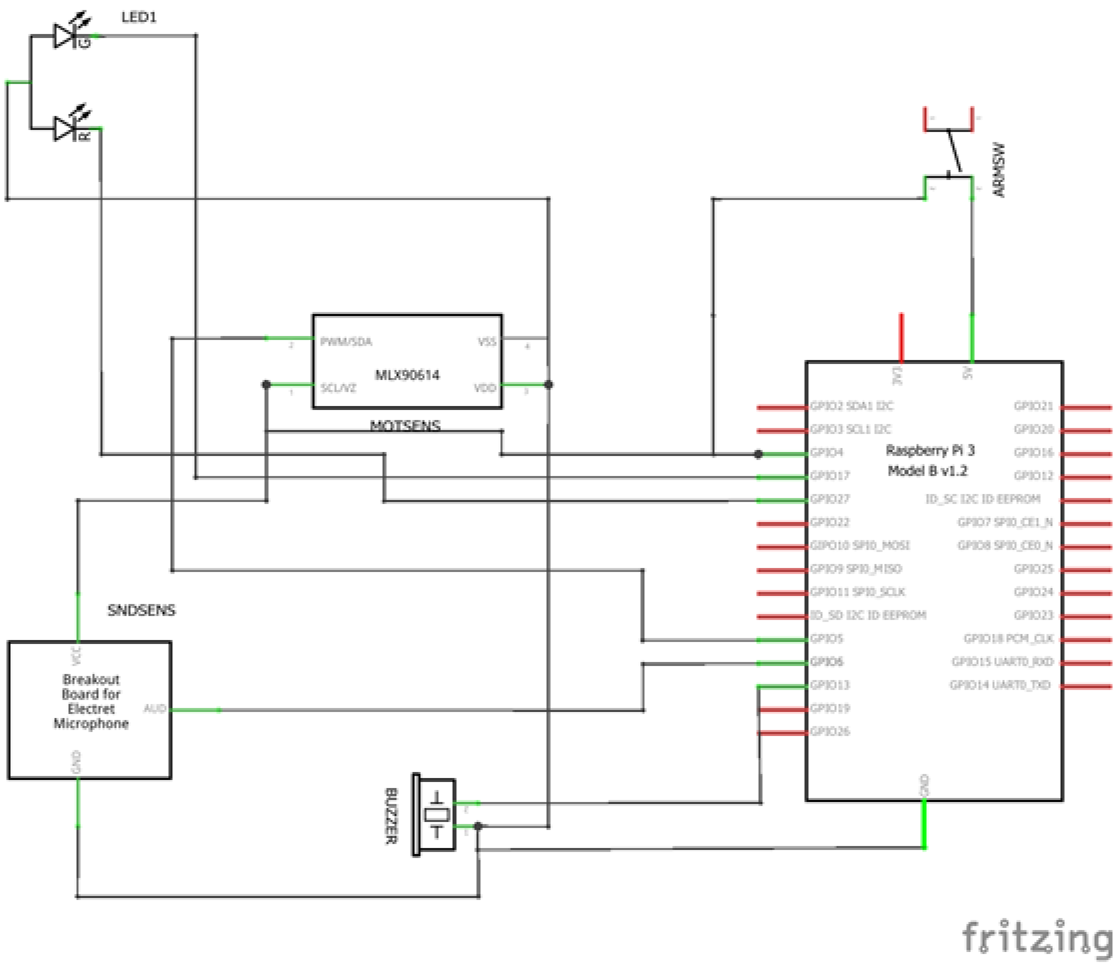

FIGURE 1 **MAIDS SCHEMATIC DIAGRAM.**

Essentially, the schematic design consisted of dragging and dropping the necessary components from the Fritzing built-in libraries onto the schematics. Once all of the symbols are placed on the schematic and the footprints to each symbol has been assigned, the wires are drawn connecting the components together.

6.2 MAIDS Breadboard

A solderless breadboard is an invaluable aid to prototyping a circuit for a project. The term breadboard originated during the vacuum tube era in the early 1920s. Tubes were plugged into sockets. The sockets and other large components were then screwed or nailed to wooden boards used for rolling dough. These breadboards made an ideal mounting platform for the components, and gave birth to the technique's name. Interconnections were made by soldering wires between appropriate pins on the tube sockets. Power and ground buses—made from heavy copper wire—were nailed or screwed to the wooden board. Early breadboards often used additional nails as connection points where wires could be wrapped and soldered. Terminal strips were also used for interconnection points. Nowadays, solderless (do not require soldering to make connections) breadboards are the norm.

The breadboard used in the MAIDS project is a full size board made from plastic and is rectangular in shape. In this typical solderless breadboard, the holes are designed to accept standard IC pins on 0.1" centers. Internally, the center part of the board is divided into two rows that are subdivided into a number of vertical columns having five pins connected together. The two horizontally connected rows at the top and bottom of the board make convenient buses for supply voltages and ground connections.

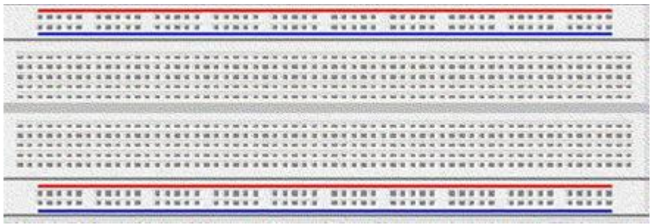

FIGURE 2 TYPICAL PROTOTYPING BREADBOARD (KESTER, 2016)

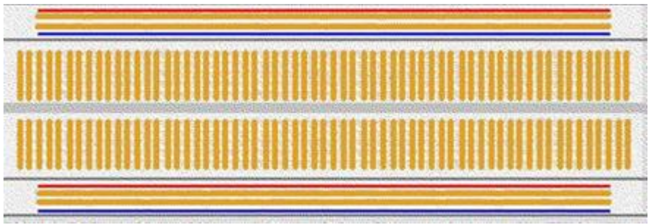

Figure 3 Breadboard Internal Connections (Kester, 2016)

The leads of components such as LEDS module, sound and motion sensors, are inserted into the holes. Each set of five holes connected by a metal strip underneath forms a node (a point in a circuit where two or more components are connected). Connections between the different MAIDS components are made by inserting their leads in a common node. The long top and bottom row of holes, indicated by the red and blue stripes are used for power supply connections (3.3 V DC and ground (GND)). The rest of the circuit was built by inserting components and connecting them together with jumper wires. It is worth noting that solid core wires rather than stranded wire are best to use with solderless breadboards. The connections are not permanent, so it is easy to remove components.

The resulting MAIDS circuit breadboard prototypes (Fritzing and actual) are shown below.

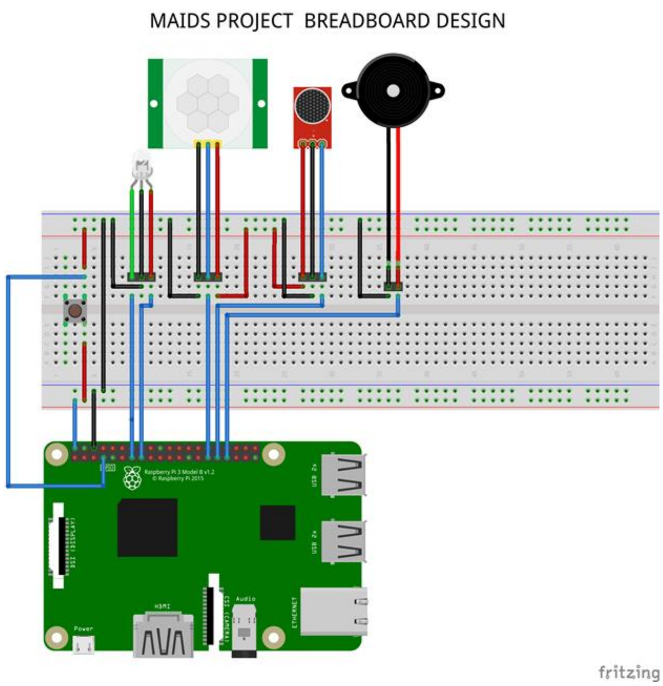

FIGURE 4 BREADBOARD CIRCUIT PROTOTYPE IN FRITZING.

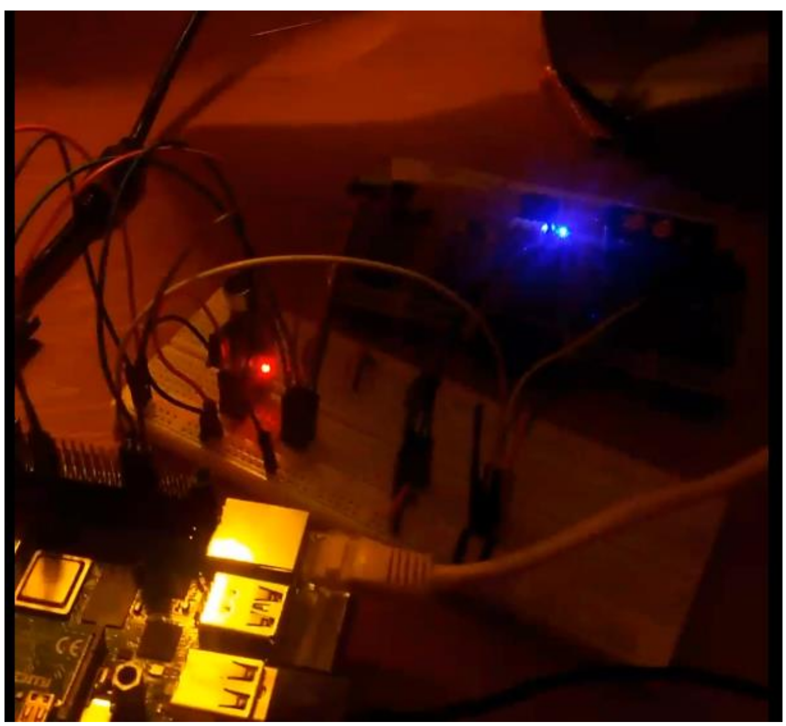

FIGURE 5 ACTUAL MAIDS BREADBOARD PROTOTYPE.

6.3 MAIDS PCB Board

Before the advent of PCB boards, electronic circuits were constructed using the point-to-point wiring process. Unfortunately, "…this approach led to frequent failures at wire junctions and short circuits when wire insulation began to age and crack." (Sparkfun, 2020) The creation of the PCB board resulted from the electronic industry's move towards the use of integrated circuits (smaller size and lower cost electronic components) and pressure on manufacturers to reduce the size and manufacturing costs.

PCB (an acronym for Printed Circuit Board) is the traditional name for the bare board which supplies the circuit layout and on which the components are mounted. A printed circuit board is used to mechanically support and electrically connect electronic components using conductive pathways, tracks, traces or vias etched from copper sheets laminated onto a non-conductive substrate. A PCB board permits signals and power to be routed between physical devices. The metal material used to make the electrical connections between the surface of the PCB and the electronic components is called solder. Solder, being a metal, serves as a strong mechanical adhesive for the components.

The MAIDS PCB board is composed of alternating layers of different materials which are laminated together resulting in a single object. The base material of the MAIDS PCB board is a solid fiberglass core designated as FR4 which provides the PCB board's rigidity and thickness. The thickness of the PCB board is the standard 1.6 mm (0.063"). A thin layer of copper foil is laminated onto the fiberglass board with heat and adhesive. MAIDS' PCB board contains one ounce of copper per square foot and is double sided board (copper is applied to both sides of the substrate). Each ounce per square foot translates to about 35 micrometers or 1.4 thousandths of an inch of thickness of copper.

The MAIDS PCB board did not have a layer on top of the copper foil called the solder mask layer (usually used to insulate the copper traces from accidental contact with other metal, solder, or conductive bits) nor did it incorporate a silkscreen (adds letters, numbers, and symbols to the PCB).

The final MAIDS PCB board design is shown below.

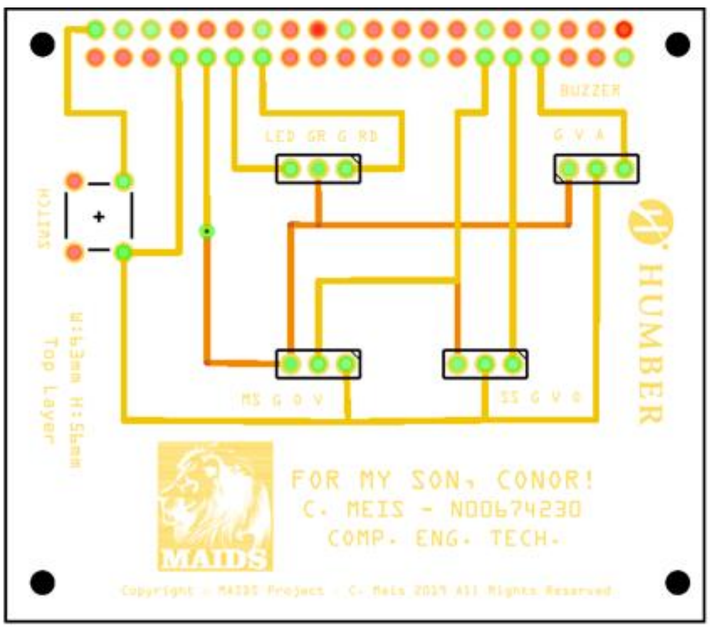

FIGURE 6 MAIDS PCB BOARD DESIGN DIAGRAM.

The PCB board requires the following Raspberry Pi 4 and custom-made PCB board GPIO pin connections:

TABLE 1 RASPBERRY PI 4 AND PCB BOARD CONNECTIONS.

Raspberry Pi and custom-made PCB Board pin connections						
Component	Sensor pin	RP4 pin	Sensor pin	RP4 pin	Sensor pin	RP4 pin
Motion Sensor	VCC	3.3V	Middle pin	Pin 29	GND	GND
LED Module	Green	Pin 9	Red	Pin 11	GND	GND
Sound Sensor	VCC	3.3V	GND	GND	Digital Input DO	Pin 31
Push Button	VCC	3.3V	Signal Pin	Pin 7, Motion Sensor and Sound Sensor		

The final MAIDS PCB board is shown below displaying the top layer with the via (a hole in a PCB board used to pass a signal from one layer to another) connecting the top and bottom layers of the PCB board, the traces for connecting the LED module, sound and motion sensors, and the bottom layer (ground (GND) layer).

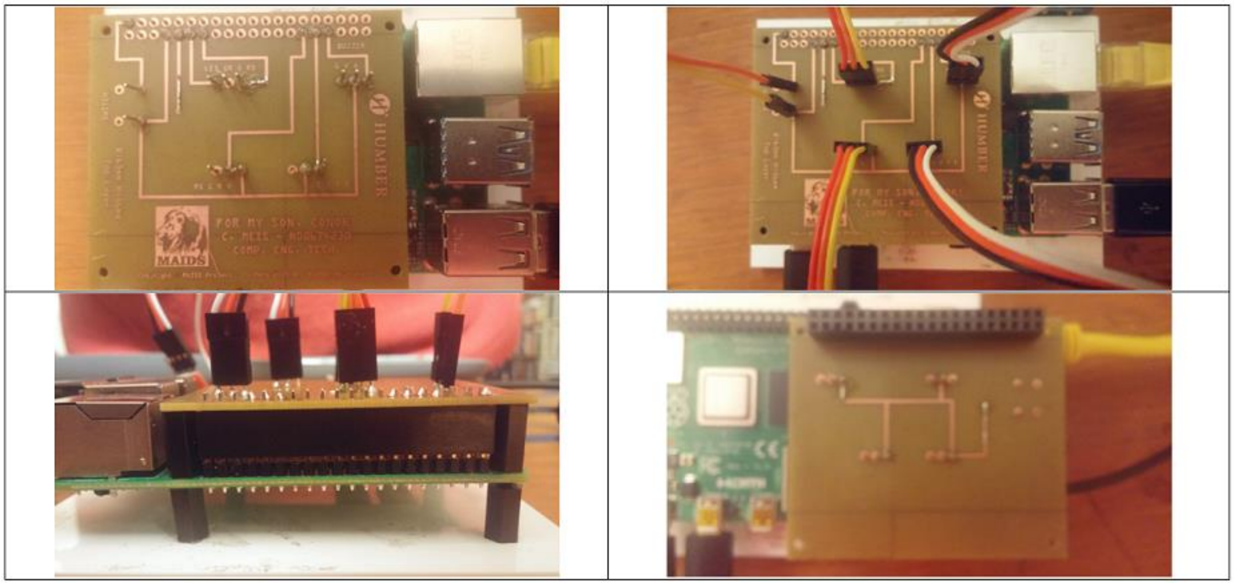

FIGURE 7 PCB BOARD BOTTOM AND TOP LAYERS.

6.4 Bill of Materials

The MAIDS' bill of materials for the design of the PCB board prototype is as follows:

TABLE 2 BILL OF MATERIALS FOR MAIDS PROJECT.

4	**MG Chemicals 5" x 3" Copper Clad Board, Double Sided, 1 oz Copper, 1/16" Thick, FR4**	**587**	**1**	**https://www.amazon.ca/MG-Chemicals-Prototyping-1-Ounce-16-Inch/dp/B008OAFOUO/ref=pd_sbs_328_2/132-5081513-4200133?_encoding=UTF8&pd_rd_i=B008OAFOUO&pd_rd_r=95480a50-419b-47d5-91b4-3b88102fc136&pd_rd_w=zfGQl&pd_rd_wg=FhDkE&pf_rd_p=dbebb38c-0e3d-4a67-ac15-432d7c7a2789&pf_rd_r=A5QQ0YC5SPEYDWN36ZA7&psc=1&refRID=A5QQ0YC5SPEYDWN36ZA7**
5	Elegoo 120pcs Multicolored Dupont Wire 40pin Male to Female, 40pin Male to Male, 40pin Female to Female Breadboard	EL-CP-004	1	https://www.amazon.ca/Elegoo-120pcs-Multicolored-Breadboard-arduino/dp/B01EV70C78/ref=sr_1_1_sspa?keywords=jumper+cables&qid=1579928593&s=industrial&sr=1-1-spons&psc=1&spLa=ZW5jcnlwdGVkUXVhbGlmaWVyPUEyVkZXSjU0U1RGSjFQJmVuY3J5c

	Jumper Wires Ribbon Cables Kit for Arduino			HRlZElkPUEwMDcxMTkxM0tSVVU5WVBKQVhKMiZlbmNyeXB0ZWRBZElkPUEwMzg1ODgyMUwxRTJBRE0zR1dIWCZ3aWRnZXROYW1lPXNwX2F0ZiZhY3Rpb249Y2xpY2tSZWRpcmVjdCZkb05vdExvZ0NsaWNrPXRydWU=
6	AUSTOR Lead Free Solder Wire with Rosin Core 0.6mm	AMA-17-532	1	https://www.amazon.ca/dp/B071XVPJVX/ref=sspa_dk_detail_0?psc=1&pd_rd_i=B071XVPJVX&pd_rd_w=apWS0&pf_rd_p=4b7c8c1c-293f-4b1e-a49a-8787dff31bcb&pd_rd_wg=s5ddm&pf_rd_r=Y6RJTAX1MR0X3E50MH9X&pd_rd_r=b093f5a4-5e04-4c4a-bf07-683669f8db02&spLa=ZW5jcnlwdGVkUXVhbGlmaWVyPUExWUMwVUFXVTBTMzUmZW5jcnlwdGVkSWQ9QTA0Nzk5MzAzSENLOTZDTEhKOTBBJmVuY3J5cHRlZEFkSWQ9QTAyMTk5NjJJTDlaVjVBQUtLRksmd2lkZ2V0TmFtZT1zcF9kZXRhaWwmYWN0aW9uPWNsaWNrUmVkaXJlY3QmZG9Ob3RMb2dDbGljaz10cnVl

6.5 Time commitment

The order lead time, the time from order received to customer order delivered was approximately 2 weeks. The order handling time, the time from customer order received to sales order created was less than 12 hours. The manufacturing lead time, Time from sales order created to production finished (ready for delivery) was approximately 2 weeks. The production Lead Time - Time from start of physical production of first submodule/part to production finished (ready for delivery) was approximately 2 weeks. The Delivery Lead Time - Time from production finished to customer order delivered was approximately 1 week due to testing and slight software modifications. (Rajaniemi, 2012, Measuring and Defining Lead Time in a Telecommunication Production)

The working time, time to design and fabricate the final MAIDS PCB board deliverable, was 1.5 hours per work day, five days per week (total of 7.5 hours per week), for a total of 15 hours, total.

6.6 Testing

The testing phase of the PCB board included the following:

- Applying the DRC rules in Fritzing to identify possible trace overlaps that might lead to short circuits which were corrected as before passing on to the manufacturing process.
- Once manufactured, the board was meticulously inspected visually for board discolorations, broken traces, correct connections and cracks on the board itself. By examining the board and the surface-mounted components, one can identify obviously damaged or disconnected parts before beginning testing. In addition, the boards was inspected for obvious signs of oxidation and corrosion such as rust. Wires were also inspected to make sure all of the components were connected. PCB boards was found to be physically sound and correctly connected from node to node.
- A magnifying glass was used to identify tin whiskers between pads and solder joints along with tin bridges. None was found.
- No cracks or blobs of solder were found.
- Opens tests were performed with a multimeter to make sure currents flowed between nodes by means of a LED-Resistor component placed between connecting points (nodes) to check for electrical conductivity. All traces and vias were found to carry currents properly.
- Shorts-test were performed using a multimeter were the resistance between neighboring traces and pads were measured on a PCB resulting in high resistance. No shorts.

6.7 PCB Board Final Status Report

Prepared by Claudio F. Meis, February 12, 2020.

The presentation of the MAIDS project at the Capstone Project EXPO at 1:00 – 4:00 p.m. on Thursday, April 9, 2020, is still on track.

The following work has been completed on the MAIDS PCB board:

- PCB Board schematic design.
- PCB Breadboard design (Fritzing and actual)
- PCB board fabrication (laser etching)
- PCB board testing.

Progress against Milestones

Schematic Design

Milestone 100%

Progress 100%

Breadboard Design

Milestone 100%

Progress 100%

PCB Board Fabrication

Milestone 100%

Progress 100%

PCB Board Testing

Milestone 100%

Progress 100%

Key Issues

No issues need to be resolved to meet Capstone Project EXPO deadline.

Action Steps

None.

7.0 Printed Circuit Board

The success of any project is often dependent on the foundations it is built upon. Much in the same way, the success of any electronic device depends on what it is built on. The PCB board of any electronics device relays electrical signal that performs some function for the equipment. Be it the communication signal between Raspberry Pi 4 and the custom-made PCB board, or a simple on-off signal from the switch, the effectiveness of the design is a function of the capabilities offered by the PCB board itself. A Printed Circuit Board (PCB) does not just connect electrical components using etched copper pathways, but also provides mechanical strength to it.

This section of the report does not concentrate on the actual fabrication process but on the specification, guidelines, considerations and recommendations required to produce an error-free PCB board. The actual fabrication process is delineated in sections 3.1.2.1 PCB Board Manufacturing Process and 3.1.3.2 PCB Board Cutting and Etching of this report.

7.1 PCB board Design Flow

In order to design a successful MAIDS' PCB board careful thought was given to its design flow. The design flow for the MAIDS project consisted of six major procedures:

1. Logic design((section 3.2.11.1 MAIDS Schematics)
2. Design verification by circuit simulation (section 3.2.11.1 MAIDS Schematics)
3. Schematic design (section 3.2.11.1 MAIDS Schematics)
4. PCB design (section 3.2.11.1 MAIDS Schematics)
5. Fabrication Specifications of the PCB board (This section)
6. Testing of PCB board (section 3.2.11.1 MAIDS Schematics)

Each section stated above will provide information on the particular part of the design flow process.

7.2 PCB Fabrication Specifications

PCB board manufacturing begins with the user-generated artwork that is then sent to the manufacturing facility in a particular format (RS-274X Gerber file) to be a laser etched. MAIDS used the following three standard technologies during the manufacturing of the PCB board: Machining, Imaging and Etching.

7.2.1 Machining

Machining includes drilling, punching holes and routing on a PCB with laser cutting. The strength of the board needs to be taken into account while machining hole-diameters accurately. Small holes were avoided so that plating was easily accomplished.

7.2.2 Imaging

Imaging transfers the circuit artwork onto individual layers. MAIDS' double sided PCB board design used direct laser imaging for creating the patterns on a print-and-etch basis.

7.2.3 Etching

Etching refers to the removal of unwanted metal and dielectric from the board that takes place by either dry or wet processes. MAIDS used a dry process. The uniformity of etching is the prime concern in this stage.

7.3 PCB Board Specifications

The specifications used in the design and fabrication of the MAIDS PCB board are listed in the table below.

TABLE 3 PCB BOARDS DESIGN SPECIFICATIONS.

Parameter	Standard
Annular ring: Internal Minimum Pad Size	.014" larger than finished hole size
Annular ring: External Minimum Pad Size	.014" larger than finished hole size
Plane Layer Clearance	
Plane Layer Clearance - PTH & NPTH Hole to Inner Layer Trace	.015" Spacing
Inner Layer Clearances:	0.010”
Copper to Edge of PCB:	0.010” for outer layers, 0.015” for inner layers, 0.020” is preferred.
Pad Size/Annular Ring:	Pad size should be +0.010” over the finished hole size for Vias +0.014” over the finished hole size for Component holes
Hole Size:	0,008” minimum finished hole size, 0.015” or larger hole size recommended
Copper Trace Width/Spacing:	0.005”) trace widths Recommended minimum spacing: 8 mils.

Inner Layer Line width on 1 ounce copper	Standard-.005"

7.3.1 PCB Board Component Placement

The component placement stage of the PCB layout process is very important. How the designer of the PCB board places the electronic components determine how easy the board is to manufacture, as well as how well it meets the original PCB design requirements. The MAIDS project used the following general board layout guidelines to place the components on the PCB board:

1. Orientation: All similar components were placed in the same direction. This helps the operative routing of the PCB board design, as well as, to help ensure a well-organized soldering process during assembly.
2. Placement: Placing components on the solder side of a board that would rest behind plated through-hole components should be avoided.
3. Organization: All through-hole (TH) components should be placed on the top side of the PCB board to minimize the number of assembly steps.

7.3.2 PCB Board Copper Thickness

Copper thickness of PCB boards can be specified directly or as the weight of copper per area (in ounce per square foot). One ounce per square foot is 1.344 mils or 34 micrometers thickness. MAIDS uses the common FR-4 substrate with one ounce copper per ft^2 (35 µm) which is the most common thickness;

7.3.4 PCB Board RoHS Compliance

Manufacturers, retailers and suppliers of electrical and electronic products in Canada need to comply with regulations stipulated in the Restriction of Hazardous Substances directive (RoHS2 Directive, 2011/65/EU). The directive bans the use of lead (among other heavy metals) in consumer items. MAIDS' PCB board is RoHS-compliant, meaning that all manufacturing processes did not involve the use of lead, all solder used was lead-free, and all components mounted on the board were free of lead, mercury, cadmium, and other heavy metals.

7.3.5 PCB Board Laminate

FR-4 is by far the most common material used today. The board stock with un-etched copper on it is called: copper-clad laminate. FR-4, is a woven fiberglass cloth impregnated with an epoxy resin (also known as polyepoxides, are a class of reactive polymers which contain epoxide groups).It provides low water absorption (up to about 0.15%), good insulation properties and good arc resistance. Several grades with somewhat different properties are available and is typically rated to 130 °C. The MAIDS' PCB board uses an FR-4 laminate in its fabrication.

7.3.6 PCB Board Trough-Hole Technology

Through-hole technology (also spelled "thru-hole"), refers to the mounting scheme used for electronic components that involves the use of leads (wire or a metal pad designed to connect two locations electrically) on the components that are inserted into holes drilled in printed circuit boards (PCB) and soldered to pads on the opposite side by manual assembly (hand placement). MAIDS' double-sided PCB board used through-hole technology. It is worth noting that Through-hole manufacturing adds to board cost by requiring many holes to be drilled accurately. Through-hole technology used holes and vias with a diameter of 0.008”.

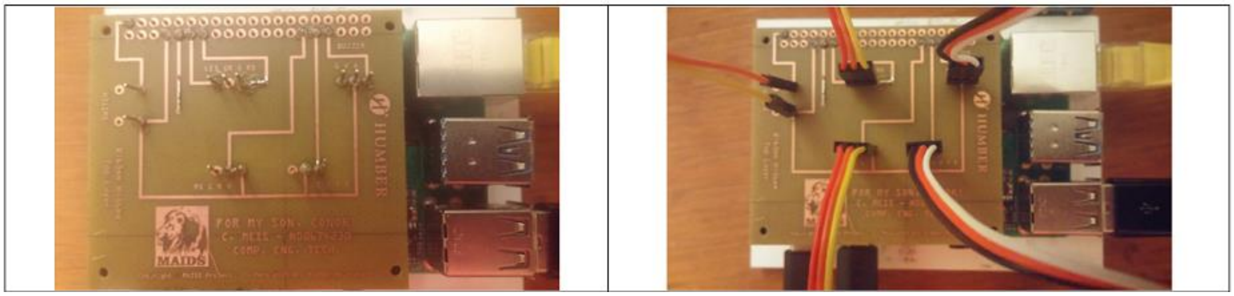

Figure 10: Final MAIDS PCB board.

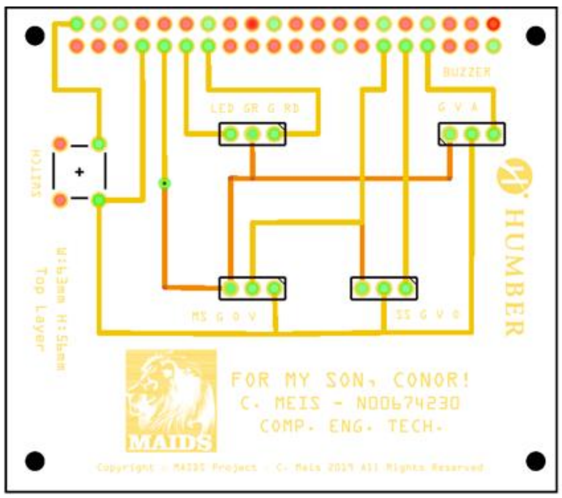

Figure 11: Final PCB Board Design Using Fritzing Software.

7.3.7 Routing Guidelines for PCB Layouts

MAIDS’ PCB layout followed the recommended best practices to achieve a trouble-free layout. In the MAIDS project, traces were placed as directly as possible between components, as well as, providing the shortest path between them. Is it also worth noting that if component placement forces horizontal trace routing on one side of the board, then the designer should route traces vertically on the opposite side.

7.3.8 PCB Board Size and Shape

MAIDS' PCB board was designed to be rectangular in shape and as small as possible. The board's size of 63 mm x 56 mm assured that the PCB board did not overlap the Raspberry PI 4 and therefore needlessly increase the size of the projects enclosure. Also, the chosen shape and size design made sure that larger Through-hole components had enough space to make them easier to solder onto the board.

7.3.9 PCB Board Trace Angles and Widths

Professionally designed PCB boards, have most of the copper traces bend at 45° angles. One reason for this is that 45° angles shorten the electrical path between components compared to 90° angles. Another reason is that high speed logic signals can get reflected off the back of the angle, causing interference. Unfortunately, CENG317 course directives stated that traces between components should bend at 90° angles. This directive should be changed in the future to mitigate the problems stated above and produce professional PCB board designs.

Like layer thickness, the width of the PCB board traces affects how much current can flow through the circuit without damaging the circuit. The proximity of traces to components and adjacent traces will also determine how wide your traces can be. MAIDS' small PCB board design did not have many traces and components. The MAIDS trace width was calculated (0.51 mil) using the PCB Trace Calculator (Bittele Electronics, 2020) and employing the following parameters:

TABLE 4 TRACE CALCULATION PARAMETERS.

Field	Value	Units
Current (max. 35A)	50	mA
Copper Thickness	1	oz/ft2
Temperature Rise (max. 100°C)	10	°C
Ambient Temperature	25	°C
Conductor Length	1	inch
Peak Voltage	3.3	Volts
Trace width required	0.51	mil

However, having designed the PCB board in Fritzing software, the minimum trace width was set to 8 mils.

7.3.10 PCB Board Verification

MAIDS' verification process included Fritzing's Design Rule Check (DRC). DRC imposes limitations on the PCB board layout in order to ensure its successful manufacturing. The common design rules applied to MAIDS were: minimum trace spacing, minimum trace width, minimum drill diameter, and trace overlapping.

7.3.11 PCB Board Final Status Report

Prepared by Claudio F. Meis, February 15, 2020.

The presentation of the MAIDS project at the Capstone Project EXPO at 1:00 – 4:00 p.m. on Thursday, April 9, 2020, is still on track.

The following work has been completed on the MAIDS PCB board:

- PCB Board design flow.
- PCB Board Fabrication Specifications
- PCB Board Specifications
- PCB Board testing.

Progress against Milestones

PCB Board Design Flow

Milestone 100%

Progress 100%

PCB Board Fabrication Specifications

Milestone 100%

Progress 100%

PCB Board Specifications

Milestone 100%

Progress 100%

PCB Board Testing

Milestone 100%

Progress 100%

Key Issues

No issues need to be resolved to meet Capstone Project EXPO deadline.

Action Steps

None.