Embedded Cleaning Robot: from design to implementation

Adrien DOS SANTOS
School of CSE, University of Salford, Manchester M5 4WT, UK
a.dossantos@edu.salford.ac.uk

***Abstract* - This paper is a proposal of PCBA (Printed Circuit Board Assembly) design for a cleaning autonomous robot, this groups the electronical architecture of the robot, the schematics, the justifications and choices of components and the rules followed, it is driven by an ATMega128, is powered by 12V and let the possibility to be connected to a battery. This proposes a layout designed for industrial purposes and let the possibility to develop further on the research side. The PCBA answers two main issues of embedded designs and propose some technologies to better fit with current industry. The purpose of this board is in the end to be implemented in the robot and to drive an embedded AI. This PCBA is designed to answer a specification document for professional purposes. The next step of the work is the manufacturing of the board and the realization of tests.**

Index Terms – Electronics, PCB design, Industrial PCBA manufacturing.

I. Introduction

Nowadays, the development of embedded system covers wide fields from complex requirement systems to simple little devices. The main requirement is to interface sensors to an analytic pole in order to interact with an environment. This paper details the design of the electronical board. The board is driven by an ATMega128 and uses thermal sensor, IR sensors, and 9-axis IMU (Inertial Measurement Unit). The board controls 3 actuators as well, two motors and a servo motor. The goal was to design a PCB while respecting the rules of professional designers such as physics constraints but also industrial requirements. We worked helped by professionals at the AVNET Company and we used the setting of their suppliers and their libraries for some components. We profited of the requirements from the Assignment to better understand the issue, but we also added or modified some features or components in order to fit with the present technologies and knowledge.

A. Literature Review

During the work, the main goal was to understand the physic constraints in order to use the components in their best shape. Some of the technical aspect like the H-bridge, or the USB requirements needed to be developed and justified. This section details the literature:

- A Practical Approach to EMC [1], this paper depicts a new tool developed to help PCB designers improving the EMC design flow. This presents a procedural method of work to approach the design of a PCB, it helped me to organize my work from the Design Specification to the Layout Routing. This can be found at Figure 1 of the paper.
- Analysis and Application for Integrity of PCB Signal [2], this paper explain the importance of reduce the signal reflection on high-speed clock PCB, this brings solutions and assure the effectiveness of resistors or diode at the end of a signal line. However, the apparition conditions of the signal reflection seem to appear dramatically on long wires and at frequencies way higher than 400kHz I2C bus on a 200mm PCB. The parameters must still be calculated for further applications. Moreover, the paper presentation helps to organize mine which is a process with a lot of pictures.
- Design for testability of printed circuit board assemblies [3], this paper highlights the importance of reducing cycles in the creation of PCB and PCBA, in order to follow the market need, the tools must be more effective, and the application of rules globalized on the PCB just like the use of supplier data upstream the design. This also presents the project DICTA which have similarities with the actual Altium Designer software. This paper encouraged me to use industrial rules, and components proposed by the market in huge quantity (resistors E12 or E24 maximum for instance).
- Design Study for Ultraflat PCB [4], this paper proposes a way of optimizing power circuits by using new manufacturing processes, board improvements have been realized and the choice of 70um copper thickness is assumed. It motivated us to manage the power circuit with the same thickness.
- Control of Ground Plane Influence on Antenna Radiation Pattern [5], this paper exposes the risk of a ground plane in creating some standing waves while all the simulations calculate with the hypothesis of an infinite plane. It also explains that it can cause significant distortion in radiation pattern. This phenomenon can be greatly reduced by using choke tubes, inductances.
- Rule ontology for automatic design verification [6], this paper summarizes the main rules to follow when checking the PCB before manufacturing process. It also exposes the DFA, DFM, and DFT rules used in industry. This pointed important rules that we used in the design and in the choice of component libraries.
- I2C Specifications [7], this the NXP official documentation around the I2C protocol, it allows us to properly dimension the resistors and to choose the better mode configuration for our data bus.

- USB 2.0 Board Design Guidelines [8], this is the Texas Instrument documentation to design in the best way the USB connector to a chip, this allows us to improve the signal quality.
- Real-Time Face Tracking for Tele-Operated Mobile Robot [9], this paper presents an AI embedded robot which looks like the one we want to realize it mays help for the architecture of the robot and the fuse process.

B. *Paper Outline*

The work is composed of the schematics of each unit, their interaction and the PCB. It also precise the changes and the justifications of the mounting or the comparison of the components.

The document is separated in three sections, the Section II. METHODS highlight the design of the schematics, the architecture of the system and explain some features of the components used. The III. RESULTS Section proposes a solution of layout balancing development but also industrial routing. And finally, in the Section IV. CONCLUSION, we will explain our results and describe the future work to do.

II. METHODS

In this section, we will detail the work done step by step, firstly the schematic of the different units, the choice of components and the rules associated to the schematics, then the PCB design and the rules associated. The software used is Altium Designer 21. The procedure follows the method advised in the paper [1]:

- Design specification, mixing with the supplier data
- Design
- Schematic capture
- Component layout
- Routing Layout

A. *Model Description*

First, the system is an electronical board using an ATMega128. It interfaces 2 IR sensors, one thermal camera, a 9-axis IMU and use one servomotor and two motors as actuator. The transmission of the data flows through I2C bus and analogic bus from the sensors to the microcontroller, then the motors are driven by PWM signals. The board is supplied by 5V and the motors uses the 12V supply from the alimentation

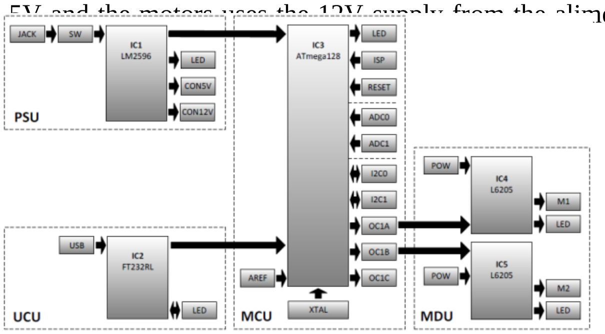

Figure 1: Block Diagram of the board

B. *Architecture*

The architecture is split in four main units:

- Power Supply Unit (PSU), the block in charge of supplying in 12V the motors and to convert 12V into 5V to supply the board, it is composed of a supply jack connector, a power switch and two 2-pins connectors.
- USB Control Unit (UCU), this block is composed of a USB A connector and a FT232 chip connected to the microcontroller, it is used to transmit the data by using UART protocol.
- Microcontroller Unit (MCU), this is the analytic block of the system, it regroups the I2C and analogic communication protocols but also the PWM channels to control the motors
- Motor Control Unit (MDU), this is the H-bridge block allowing the command of the motors

All these four units are detailed in depth in the Implementation Section.

To facilitate the implementation of these units, we created the interface schematic:

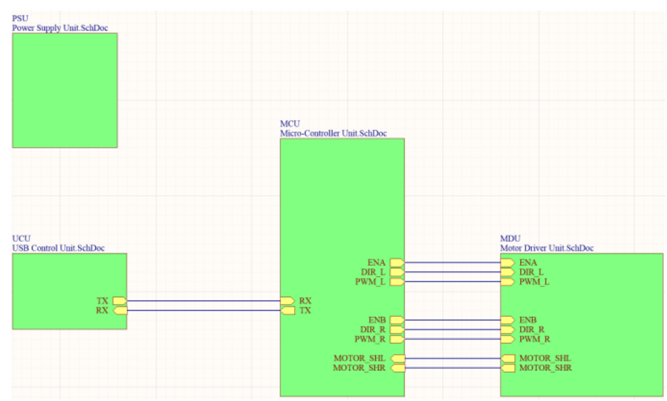

Figure 2: Interface of the cleaning robot board

Regarding the PCB, we have opted for a 4 layers board, the settings are those of the SAFE company, a PCB supplier of the AVNET Company, in this way we are sure the choice is relevant from physic side with the isolation of ground (GND) and phase 5V (VCC) but from the industrial side as well. Following the layer description:

	Layer Name	Type	Material	Thickness (mm)	Dielectric Material	Dielectric Constant	Pullba (mm)	Orientation	Coverlay Expansion
	Top Overlay	Overlay							
	Top Solder	Solder Mask/Coverlay	Surface Material	0.02	Solder Resist	4			0
1	Component Side	Signal	Copper	0.07				Top	
	Dielectric 1	Dielectric	Prepreg	0.13		4.45			
2	GND	Signal	Copper	0.0175				Not Allowed	
	Dielectric 3	Dielectric	Core	1.165		4.74			
3	VCC	Signal	Copper	0.0175				Not Allowed	
	Dielectric 2	Dielectric	Prepreg	0.13		4.45			
4	Solder Side	Signal	Copper	0.07				Bottom	
	Bottom Solder	Solder Mask/Coverlay	Surface Material	0.02	Solder Resist	4			0
	Bottom Overlay	Overlay							

Figure 3: Layout of the PCB

C. *Implementation*

The implementation began first by the creation of each unit, their description follows:

1. PSU:

   The PSU uses a LM2596 chip to convert 12V in 5V the schematic following shows to connector, one upstream the chip and a second downstream these are the link to the motor power board (not designed) and the ATMega128 (5V):

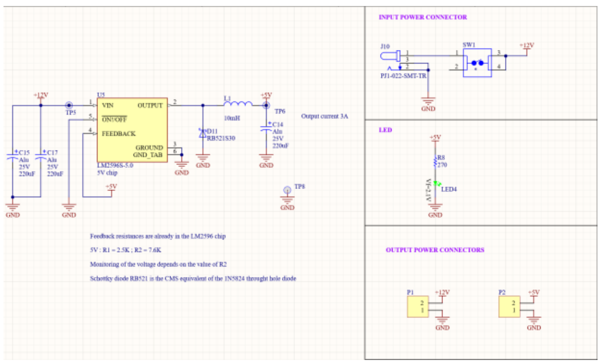

Figure 4: Power Supply Unit Schematic

One of the most important things to understand in this block is the function of the trio inductance, capacitance and Schottky diode. This is the heart of buck device, the current loops in this trio of components when the supply is off-state and the current loops in the whole system when the supply is on-state.

On-State	VL = Vi - Vo
Off-State	VL = -Vo

Figure 5: Voltage across the inductance

Regarding the feedback system, we can modify the 5V voltage by adding a resistor on the Feedback pin of the LM2596, however, two resistors are already in place in the chip: R1=2.5K (fixed) and R2 = 7.6K. In the design, we replaced the 1N5824 Schottky diode by the CMS equivalent RB521.

2. UCU:

   The UCU is a restrictive block on the schematic side as on the PCB side, it requires to be protected from high noise interferences but also to be protected from current peak. The most common component used to protect from electromagnetic interferences is the ferrite bead that we coupled with a capacitance so the voltage from the USB does not destroy the FT232 chip nor the data (According to the datasheet). For the D+ and D- routes, the rules are explained in the USB documentation [8], the differential impedance must be set at 90Ohm with plus or minus 10%. Moreover, the length of both lines must be equal. The Altium software let the possibility to do it easily and professionally from the schematic side but on the PCB side as well. The USB voltage is used to supply the FT232 which is the only component not linked to 5V. The ground is the same withal. Regarding the RESET feature, the datasheet let the choice of letting the pin unconnected or pulled-up to 5V. In order to reduce current consumption, we decided to let the pin unconnected. The Figure 6 shows the UCU schematic:

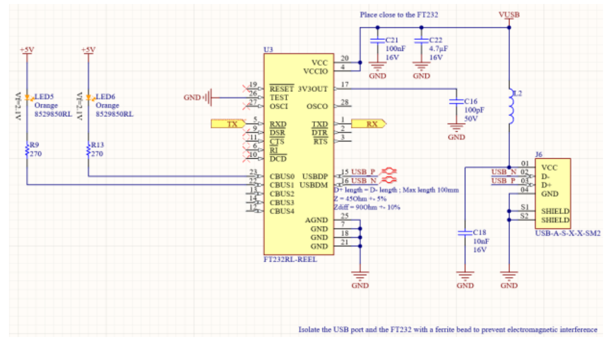

Figure 6: USB Control Unit Schematic

3. MCU

   Regarding the Microcontroller, the main thing is to appropriately choose the I2C specifications. The speed of the bus and the pull-up resistor values depends on the mode chosen, in our case, because of the length of the bus (approximated at 150mm) we must be careful concerning the bitrate. In total the length of the I2C bus is 230mm which represents a capacitance value of

   0.39 pF/cm * 23cm = 8.97pF. The calculation follows the formula:

$$C = 3.333\sqrt{er} \times dimension \times 120 \div \sqrt{er} \times \cosh^{-1}(s \div d) \qquad (1)$$

With s the distance between the two wires, d the diameter

$$d = 2\sqrt{w \times h}/PI \qquad (2)$$

w the width of the copper, h the height of the copper), and er the relative dielectric constant in mm (we used the value 1 in our case because of the 400 Kbit/s speed of the bus). This result can be neglected compared to the capacitance of 200-400 pF of the bus in fast mode. Considering these results and the I2C specifications, we chose the fast mode option to communicate on the I2C bus.

Regarding the pull-up resistor values, the formula follows, it comes from the ATMega128 Datasheet, but is also presented in the official I2C specifications:

$$\text{Freq} < 100\text{kHz} \implies R_{\min} = \frac{V_{cc} - 0.4\text{V}}{3\text{mA}}, R_{\max} = \frac{1000\text{ns}}{C_{\text{bus}}}$$

$$\text{Freq} > 100\text{kHz} \implies R_{\min} = \frac{V_{cc} - 0.4\text{V}}{3\text{mA}}, R_{\max} = \frac{300\text{ns}}{C_{\text{bus}}}$$

Figure 7: Formula - Pull-up Resistor values for high and low frequencies

With our calculations, we chose a 1.6K resistor.

Regarding the ISP block, we chose to use a JTAG protocol without RESET feature, the RESET button being nearby, we have opted for a 6 pins connector. To facilitate the test and maintenance actions, we have opted for a LED solution connected to the microcontroller and driven by the chip, the checking of motor rotation will occur by software protocol, a choice to avoid mixing signal and power voltages.

The last feature of the microcontroller is the shunt system in order to control the current value of the motors, it will be monitored by the software which will control the speed of the robot if the current is too high. The value of the shunt resistor and the schematics are detailed in the MDU section.

Finally, the microcontroller works with an external oscillator of 16MHz and two capacitances of 22pF.

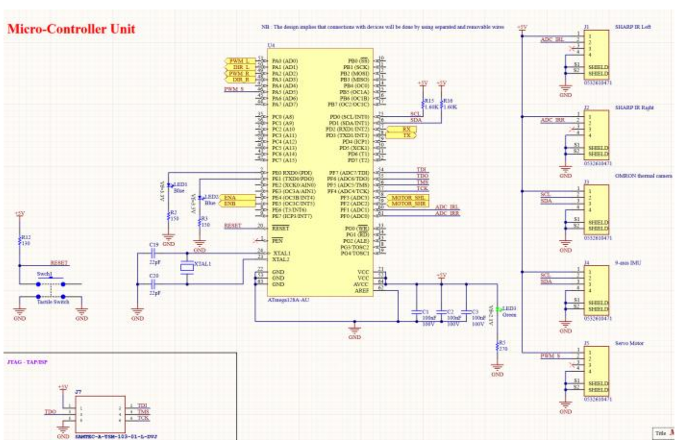

Figure 8: MCU Schematic, JTAG connector and microcontroller

4. MDU

The MDU is the H-bridge feature of the board, it takes order from the microcontroller and by hardware drive the motors to have different behaviors. These behaviors are explained in the datasheet and a summary follows with the truth table:

INPUTS			OUTPUTS	
EN	IN1	IN2	OUT1	OUT2
L	X	X	High Z	High Z
H	L	L	GND	GND
H	H	L	Vs	GND
H	L	H	GND	Vs
H	H	H	Vs	Vs

X = Don't care

High Z = High Impedance Output

Figure 9: H-Bridge Truth Table

The outputs can have 5 different states, the high impedance means a free-wheel behavior from the robot, the GND on both outputs and the Vs on both outputs mean a break behavior, and finally the output mixing GND and Vs are the rotation of the motor in one way or in the other.

Because of the output current of the motors, the layout chosen is the parallel connection, which allows the load to get 5.6A in peak.

Considering the 5.6A, it is important to protect the motor in case of high peak of current, to solve this problem, we added two shunt resistors.

The reference voltage of the ADC of the ATMega128 is 2.56V, considering this value, the shunt resistor value for the 5.6A peak of current is:

R = 2.56/5.6 = 0.46Ohm

The voltage covers full resolution of the ADC. However, the power to dissipate by this component is:

$$P = RI^2 \qquad\qquad U = RI \tag{3}$$

P = 0.45 * 5.6² = 14.336 W. This value is very high, and this will imply a resistor too big for our design. We decided to reduce the volume of this resistor and to lower the power to dissipate to 3W. This gives a value of 0.096Ohm but this implies a too low resolution for the ADC. We opted for two resistors of 0.15Ohm mounted in series. In this way, when a peak occurs, the resistors will heat up but will not be damaged, and the ADC's resolution is increased. Following a summary table of the different values calculated. The final layout will be composed of four resistors, two for each H-bridge, in this way the power to dissipate will be 4.7W but the ADC resolution will have a value of 336*2 = 672 which represents a ratio of 65% of the ADC capability.

Summary Table of the calculated values around the shunt resistors					
ADC Voltage (V)	Current Peak (A)	Continuous Current (A)	Resistor Value (Ohm)	Power to dissipate (W)	ADC Resolution
2.56	5.6	2.8	0.46	14.34	**1024**
0.54	5.6	2.8	0.10	**3**	214.29
0.75	5.6	2.8	0.13	4.2	**300**
0.84	**5.6**	**2.8**	**0.15**	**4.70**	**336**

Figure 10: Summary Table of values calculated

NB: The bold digits are those which were parameters of the line calculation.

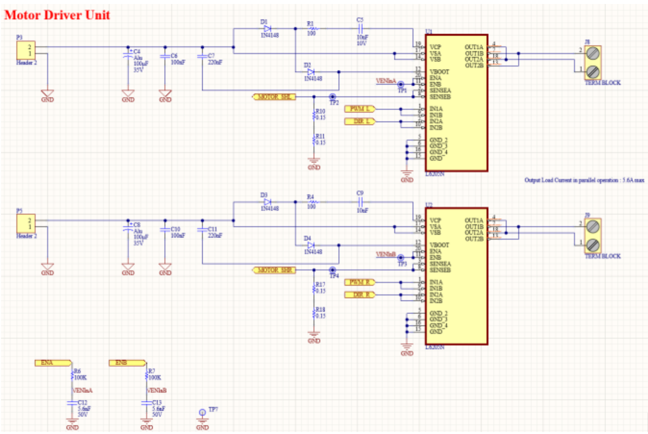

Figure 11: MDU Schematic

5. Dimensioning of Common Components
This section precise the components in detail and the potential changes between datasheet advice and components chosen. All the component casing values are 0603, this casing allow the person who needs to solder the components to do it manually, however this casing is quite small and is feasible in industrial process.
LED dimensioning for 10mA:

LED name	LED value (V)	Input voltage (V)	Resistor value (Ω)
LED1	3.3	5	150
LED2	3.3	5	150
LED3	2.1	5	270
LED4	2.1	5	270
LED5	2.1	5	270
LED6	2.1	5	270

Figure 12: Value of board resistors

Coupling capacitance:
The coupling capacitance are used to avoid noise transmission and to reduce the impedance of the supply routes. The values chosen are those advised by the constructor. The magnitude reduces categories of noise by slice of power of ten. Moreover, these capacitances must be placed as close as possible to the chip concerned

In the next section we will discuss about the PCB and the results associated to the mounting.

III. RESULTS

The first result is the power consumption of the board, we have collected the device and component operating supply currents and added 10mA for encompass the peripheral devices of the chips. The following table summarize these currents:

TABLE 1
POWER CONSUMPTION OF THE BOARD (mA)

PSU	50
UCU	53.92
MCU	104.71
MDU	4000
	4208.64

This allows us to proper dimension the battery needed, regarding the value founded and the standard battery proposed, we have several choices:

- Technology NiMH
- Technology Li-ion

Because of the NiMH technology age and the current supply capability of the Li-ion technology, we recommend using Li-ion. However, we must develop further regarding the battery charger.

By using the experience of the AVNET's industrial process, we managed to remove the single point Signal Ground of the MDU by using several planes in different layers of the PCB. It reduces the interferences and noise as well and improve global signal on the board. The four planes are:

- Signal 1
- GND (Ground recovering the whole board layer)
- VCC (Divided in 3 different polygons pour)
- Signal 2

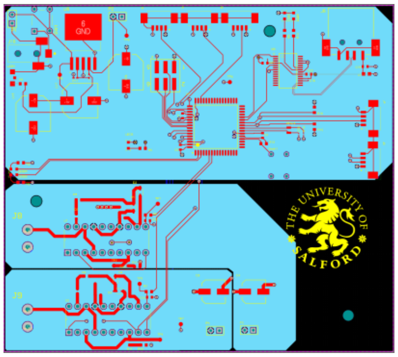

Figure 13: Supply planes

All the components are on the Signal 1 layer, and so we placed the ground plane right under the Signal 1 layer to better isolate the information signals.
All the polygons have 45° angle corners, this avoids some antenna effect.
We propose a solution of layout minimizing the length and width of the PCB and following are the characteristics of the board:

- 114mm x 110mm
- 70 um - Thickness of the copper layer
- 0.254mm/1mm - Width of copper trace (Signal/Power)
- E24 compliance on resistors, capacitance and inductance
- 3 fixing holes for mechanical convenience routed to the ground (GND)

On the layout, we propose a solution aiming to answer two issues, first the quality of the signal and second the mixing between development and research features and industrialization. To answer the first issue, we placed decoupling capacitance as close as possible to chips but we tried to create a barrier homogeneous to equilibrate the impedance of the routes. Moreover, we placed differential pairs to connect the USB to microcontroller to maximize the yield and reduce the loss of data. We also respect distances between the power and the signal blocks to avoid interferences. And particularly, the USB is at the opposite of the power H-bridge on the board. To answer the second issue, we placed some test points on the PCB in strategic places so the human who wants to test features or the machine for compliance industrial processes can both access the information. Moreover, the LEDs are placed strategically so even in a casing the main functioning LEDs still are visible. Finally, the routes and the components are spaced and big enough so they can be replace manually.

Regarding the temperature of the board, it should not exceed 35°Celcius over the ambient temperature according to the datasheet of the most solicited chip, the LM2596. This temperature is dissipated by the GND and VCC planes. Another precision about the width of the high current routes, they are designed to exceed of 10°Celcius the ambient temperature while current peak (5.6A max.) The shunt resistor brings information on it and the change of speed or strength of the motor will be monitored by the ATMega128

Finally, here is the result of the assembly in 2D and 3D:

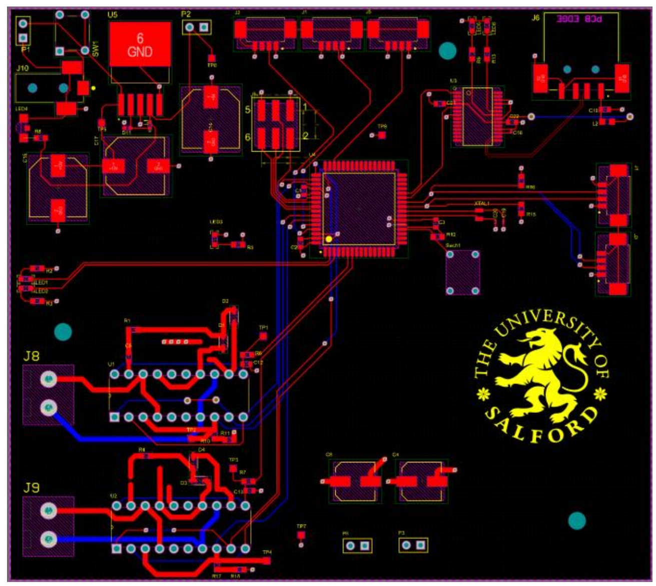

Figure 14: PCBA - Signal 1 and Signal 2 layers

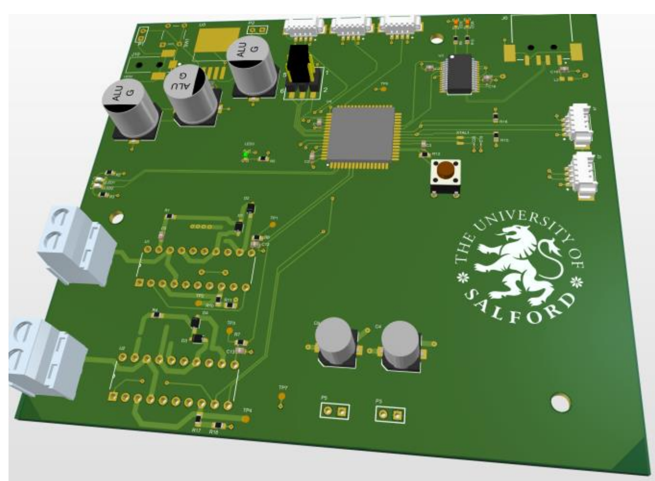

Figure 15: PCBA 3D rendering

IV. Conclusions

The PCBA proposal is complete and needs now to be manufactured in order to practice some tests. The process followed professional processes and design tools used for manufacturing. This gives an idea of the dimensions needed for the robot, and some measure to help dimension the battery and external connectors. Furthermore, feature have been added so the development on the software side is easier. The design is realized to be CE certification compliant and set up to be manufactured at the SAFE Company. Future works will optimize the length and width of the board and to place new components to replace the H-bridges. Adapted connectors will be implemented once the test and development phase will be over. The routing method will be improved as well by only using left-right or top-bottom routes respectively on Signal 1 and Signal 2 layers. This project make us learn a lot on the PCB design topic and the rules of design associated, but also in organizing the project with professional standards.

Acknowledgments

I would like to thank Josephine Piloquet and Kevin Hector for helping me in the organization of the report and the work to do, they also kept me motivated so I did not give up in these particular situations of work. Moreover, I would like to thank Jérôme Bioret from the AVNET Company, who helped me in the design of the board, who gave me piece of advice, and shared his experience with me to improve my design. This really transformed the mood of the project.

References

[1] John Berrie and Andy Slade, "A Practical Approach to EMC for Printed Circuit Board (PCB) and Multichip Module (MCM) Design" in *Proceedings of EURO-DAC 93 and EURO-VHDL 93- European Design Automation Conference*, sept.1993

[2] Xue-Ping Chen, "Analysis and Application for Integrity of PCB Signal" in *2010 2nd IEEE International Conference on Information and Financial Engineering*, sept.2010

[3] Therese Lawlor-Wright, "DESIGN FOR TESTABILITY OF PRINTED CIRCUIT BOARD ASSEMBLIES" in *IEE Colloquium on Intelligent Design Systems (Digest No. 1997/016),* in feb.1997

[4] Matthias Ludwig, Maeve Duffy, Terence O'Donnell, Paul McCloskey, and Sean Cian ÓMathùna, "Design study for ultraflat PCB-integrated inductors for low-power conversion applications" in *IEEE Transactions on Magnetics (Volume: 39, Issue: 5, Sept. 2003)*, sept.2003

[5] Peter Callaghan, Md. Hasanuzzaman Sagor, John C. Batchelor, "Control of ground plane influence on antenna radiation pattern for mobile handheld devices" in *2011 Loughborough Antennas & Propagation Conference*, nov.2011

[6] Rui Abrantes, Mário Basto, João Varajão, Luis Magalhães, Pedro Ribeiro, Luis César Freitas, "Rule ontology for automatic design verification" in *IECON 2017 - 43rd Annual Conference of the IEEE Industrial Electronics Society*, nov.2017

[7] NXP Company, "I2C-bus specification and user manual" apr.2014

[8] Texas Instrument, "USB 2.0 Board Design and Layout Guidelines" jan.2013