UNIVERSITÀ DEGLI STUDI DI PADOVA

DIPARTIMENTO DI INGEGNERIA DELL'INFORMAZIONE

Information Engineering Department

Master's Degree in ICT for Internet and Multimedia

Design of a Control Board for The Realization of a Quadcopter

Author

OZUMCEM COSKUN

2004972

Supervisor

FEDERICA BATTISTI

Academic Year

2022/2023

Graduation Date: 23/10/2023

DEDICATION PAGE

I would like to dedicate the page for those who have always supported me during my academical journey.

Thanks to…

my family who always supported me no matter what,

my friends who made the life more colorful to me,

my professor, Federica Battisti, who was always ready to guide me and support me during my thesis period,

Tommaso Manna, the kindest company mentor that any intern can ever has,

Maria Vittoria Pelosi, my colleague, who always supported and guided me during my internship...

"The biggest battle is a battle against the ignorance."

Mustafa Kemal Atatürk

ABSTRACT

The PCB, printed-circuit board, is a surface where the resistors, capacitors and other microelectronic components are mounted. Printed-circuit board can also be counted as a platform that includes the electronic components upon its body. When the design of a printed-circuit board is paid attention, there comes up some steps, challenges and main criteria that can create the most proper printed-circuit board whereas, with the wrong preference, the least useful printed-circuit board can also be obtained. The thesis aims to design a four-layer printed-circuit board for a specific quadcopter that will include three different communication protocols, UART, SPI and I2C respectively, by reducing the possible negative challenges of routing, the tracks, the mounting style and the via size to be able to have the proper printed-circuit board with as much efficiency as possible. While the challenges are explained, the design project with ALTEN Italia will be explained with all details and steps from beginning until the end of design process with illustrations made during the internship. The 3D illustration of the designed board will also be shown at the end of the thesis. Since the company software preference is Altium Designer, during the entire design and simulations, Altium Designer software has been used. Therefore, all the shown figures are taken from the software itself. In brief, with the light of this thesis, the design of a multi-layer printed-circuit board that works properly with communication protocols will be examined in an efficient way. While doing that, the main challenges during the design will be reduced or completely solved.

Key words: *Printed-Circuit Board, PCB Design, Multi-Layer Board, Altium Designer Software, Control Board of The Quadcopter, Routing, Communication Protocols*

CONTENTS

LIST OF FIGURES

LIST OF TABLES

I. INTRODUCTION

1.1. Printed-Circuit Board

The printed-circuit board is an artificial mechanical surface that has different types of microelectronic components to be able to serve a main conclusion. These boards are the mostly preferred interconnection technology in an electronic product. Printed-circuit board functionality, shape and duties are varied according to requirements of the given case or a project. The mechanical board where all the design is planned to be mounted is shaped according to designer’s preference. Although in general, the printed-circuit boards are square shaped since it provides easier connection between the components and less risk for the signal transportation, in this thesis, all the research and design will be introduced on a very specific board shape. Although at first, having not a square/rectangular board shape can be thought as having a less efficient, less functional board, during the thesis, the placement of the components according to board shape will be expressed detailly and it will be observed that having not the standard board shape for this design will not provide a less efficient PCB for the project.

As discussed in the research [1], printed-circuit board design had always been one of the main interests during its own history. During the pre-computer days, printed-circuit boards were designed by hand using adhesive tapes on the drafting films. To be able to achieve this, many hours were sacrificed over the fluorescent light box. The general technique was cutting, then placing and ripping up. At the end of the process, tracks were routed by the power of hands. However, this technique did not last for a long time. After having the computer-based printed-circuit board design, the potential of having more accurate conclusions for the required design had increased. Also, it provided flexibility and editing abilities in the board design. Therefore, what was taking many hours in that time for the board functionality takes only seconds now with the help of computer-based PCBs. In today’s PCB design, to be able to have an efficient PCB, different design softwares are invented. The software that has been used is called Altium Designer. The steps of designing a printed-circuit board with all the simulations and settings had completed in Altium Designer and every step will be explained in the coming parts of the thesis detailly.

1.2. PCB in The Industry

In today’s world, printed-circuit boards are counted as one of the most oftenly chosen interconnection technological products. Printed-circuit board requirements have been improved with the packaging density increment of both the electronical and mechanical components. Today, manufacturers prefer printed-circuit boards with finer conductor tracks but also with the thinner laminates. In addition to physical structure, the components used such as integrated circuits become more sophisticated, especially if it is compared with the last decade. While having this development, naturally, the new design rules and mounting techniques has been existed. In other words, in the business industry, rather than having insertion with the dual inline package technology, the new mounting style which is called surface mounting technology grows faster and becomes frequently preferred. In addition to this development in the business sector in PCB field, through-hole method gets more popular and the amount of conductor between the through holes rises. Also, line dimensions get diminished and hole radiuses start decreasing. With the light of these improvements in the printed-circuit board lifetime, it is naturally assumed that in the future, printed-circuit boards are going to have higher functionality, higher density, better reliability and become less costly. In addition to physical development of the board during the years in the business, the industries will tend to prefer more environmentally friendly printed-circuit boards.

1.3. Advantages of Using Printed-Circuit Board in Electronics Industry

Rather than using interconnection wiring methods, or using different techniques, there are some important reasons to select printed-circuit boards when it comes to activities in fields. The advantages of using printed-circuit board are as below as also discussed in study [3]:

- Assembly component size is decreased.
- According to the decrement is size, the weight is reduced too.
- From the financial perspective, the total unit cost is decreased during the quantity production.
- The component wiring is mechanized during the PCB design.
- Rather than changing the intercircuit capacitance, the characteristics of the circuits of each sections are able to get maintained
- From assembly to assembly, printed-circuit boards assure high level repeatability

- In addition to repeatability, printed circuit boards presents uniformity
- With the help of the repair ability for the part locations, identified electronic components and electronic systems can be obtained.
- Since the printed circuit board gets rid of the probability of error, the inspection time get decreased.
- With the help of the printed circuit board, the probability of mis-wiring or having a short circuited wiring gets decreased.

1.4. Outline of the Paper

The introductory chapter presents printed-circuit board's importance, its place in the industry and the advantages these boards provide. The second unit introduces the background information about the design order of a printed-circuit board and the critical steps during the design procedure. Third unit presents the relevant published studies focused on PCBs. The fourth unit is a case study where the whole design and connections are completed and expressed step by step. With the last unit, conclusion, the thesis will be completed.

II. BACKGROUND

2.1. Classification of Printed-Circuit Board

Printed-circuit boards are grouped according to different attributes. PCBs are classified into three groups with respect to the usage and their applications. These are consumer, the professional and also the high reliability board respectively. The first group, consumer PCBs, are preferred to be used for the products as TV, radio and for the other measuring equipment. The consumer PCBs uses cheap base material if compared with the other groups. The second group, professional PCBs, consists of higher qualified materials to obtain tighter electrical specifications by using the fabrication techniques. Higher reliability boards are preferred for the specific applications and supplies the best electrical properties when compared with the others as discussed in research [2].

In today's PCB technology, more easy to understand ways are used which depends on the plane numbers or wiring the layers. Classification of the boards provided an advantage of becoming directly related to board requirements. Here is the most essential construction of printed-circuit boards.

2.1.1. Single-Sided Printed-Circuit Board

As understood from the name itself, in single sided PCB, the wiring is only available on one of the sides of substrate. While the side that has all the circuit path which is named as solder side, the other part is named as component side. Since these kinds of boards are not very complicated, they are mainly preferred for the simpler circuitry. Therefore, the manufacturing expenses gets less since the complexity is reduced. Although these boards seem quite basic, they occupy an important place for the printed circuit boards that are produced currently for both professional and the nonprofessional grades.

Most of the times, single-sided PCBs are produced by the print and etch, and die cut techniques. In these techniques, the die which carries an illustration of a wiring path is used. In the PCBs, the components are adjusted to jump above the conductor tracks. However, if this adjustment is not possible, the adjustment is done by using the jumper wires. Since the main purpose is having a less cost in such boards, the amount of the jumpers have to be limited and used wisely as indicated in [2].

2.1.2. Double-Sided Printed-Circuit Boards

As assumed from the title, in such PCB boards, the wiring patterns are created above the both sides of the material. For instance, while in the single-sided board, the circuitry is only above the one side, in the double-sided printed-circuit boards, the circuitry is placed above the both component side and also above the solder side. The other comparison is about component density. In the double-sided printed-circuit boards, the component density is much higher than a single-sided PCB. There are two types most commonly used double-sided PCBs in the industry, and these are called PTH and non-PTH. In other words, board with plated through-hole connection and the board without plated through-hole connection.

The PTH board's circuitry is observed above the both of the sides of the insulated substrate that is contacted by the metallization of the hole wall inside of the substrate. The substrate intersects with circuitry on both of the sides. This technique is also counted as the basis for the most of the produced printed-circuit boards. The other reason why this board style in double-sided PCBs is very famous is that with the help of this technique, when the circuit's complexity, and circuit density is high, this technique will provide much better solution than single-sided PCB not to cause serious damage and increase the efficiency at the end of manufacturing.

The non-PTH board is considered as the extension version of the single-sided PCB. Since the plating for these boards can be avoided, the cost becomes considerably less. During the layout process of such boards, the amount of solder joints above the side with the components must be held to the minimum to be able to facilitate the component reduction. However, most of the PCB designers recommend that it would be better to realize the conductors on the non-components side of the board. The component side should keep only the remainings. The interconnections in the double-sided PCBs are obtained by using jumper wires. An insulated solid wire is put through a hole and then, firstly clinched, lastly soldered to the pad above the each part of the board.

2.1.3. Multi-Layer Printed Circuit Boards

The multi-layer PCBs help engineers to use more than two PCBs with the very thin layer material which is also called as prepreg. This material is placed inside the each layers. In these boards, the electrical circuits are provided with the interconnection of different layers with the help of plated through holes. Multi-layer PCB has minimum three circuit layers. As a number, there is no any maximum limit. However, during the PCB research done by the engineers so far, even 50 layers have been used.

The areas that multi-layer printed circuit boards are preferred are the reason why this type of boards are actually intended to create. For instance, mostly in the military works and applications related to space, less volume and weight are required in the interconnections. Therefore, such boards would be the best option to use for these types of fields. In addition to this, in some cases, the interconnection complexity of the sub-systems desires more complex wiring styles. Also, in these cases, multi-layer PCBs are the best selection to be able to provide the demanded result as discussed in study [2]. In general, the multi-layer boards are separated into two groups as four-layers and eight-layers. The type of the printed-circuit board that is designed for the thesis is a multi-layer board with four layers.

2.1.4. Rigid and Flexible Printed-Circuit Board

The printed-circuit board may be classified according to the basis of used insulating material such as flexible and rigid. While in flexible boards, flexible substrates are used, in the rigid boards, different types of materials can be used. The substrate material which is used for the flexible boards are generally polyester and polyamide structures. The laminates preferred in the flexible type of boards have the copper above one or two sides of the rolls. However, when it comes to rigid-flex printed circuit boards, they are the combination of both the flexible and rigid boards. By saying that, these boards are bonded and have structures with three dimensions to be able to make the flexible parts connect to the rigid boards. The main purpose of this connection is to support components. With the help of this arrangement, the more efficient volumetrically packaging style becomes obtained. The other advantage of rigid and flexible printed-circuit boards is that this type of PCB can be seen in single-sided boards, double-sided boards which can be both PTH and non-PTH, or also in the multi-layer boards.

2.1.5. Silkscreen

Silkscreen layer, in other words component layer, is the layer on the top which includes the component designators, text of the components and component outlines. To be able to add silkscreen to the board, the designer should use silkscreening process. The standard colour for the silkscreen is white. If the designer wants to change or add other colours, this needs to be requested with extra cost. During the design process of the board, designer should check the text size of each component. All of the text sizes should be kept as standard or changed but the same size. In addition to this, the orientation must be done in the same direction. When the lay out section starts, it would be logical to add the component overlay which will show the component actual size. Also, all the polarization should be marked on the pin1.

Although silkscreen layer is an important layer, the designer should not rely only on this layer's accuracy because silkscreen layer is the least accurately aligned layer when compared with other ones. The silkcreen should be checked so that there becomes no overlap with the pad. Since there is no any specific recommendation for the width size of the component overlay line, designer can create the lines according to a way which fits the design the best. In addition to this, only the component designators should be added on the silkscreen, not the component values.

2.1.6. Solder Mask

Solder mask is a polymer coated above the board that covers the pads not to have a bridging possibility between the pins. Solder mask is quite needed for the surface mounting style and for the pitch devices. On the PCB, solder mask frames are everything except vias and the pads. That is why whichever software the designer uses, the software will remove the solder mask around the vias the pads and because of this, case is named as mask expansion. During the design, the measurement of the mask expansion should be done too. Generally, the mask expansion is adjusted according to the board structure. However, it should not be very big so that there becomes no mask between the pitch devices.

If the printed-circuit board design is under control, generally designer does not need to add anything above the solder mask. However, if the designer decides to make solder mask off in the some specific parts, fills and the tracks needed to be added in the layer. For the usage of solder mask, there are two kinds which are called silkscreen and the photo imageable. If these two kinds are compared, much better alignment and the resolution are provided in the photo imageable type. In this type the standard layer color is green. However, designer is able to choose an another one if needed to make the design more personal.

2.1.7. Mechanical Layer

Mechanical layer in the printed-circuit board is quite needed to be able to supply the board outline and the instructions for the manufacturing. Although mechanical layer is not counted as the main PCB design, it is a communication technique between designer and the manufacturer to tell the manufacturer how the designer desires the PCB assembled. The good part about this layer is that there is no specific tough and fast rules. It can be used however the designer prefers. The only important step is, whatever the designer prefers, it should be explained to the manufacturer.

2.1.8. Keep-Out Layer

Keep-out layer is very simple and with less complexity when compared with the other layers. Generally, this layer describes the areas on the PCB which the designer does not want the routing manually done or auto-done. The areas for the clearance around the pad and components with high voltages are some of the examples.

2.1.9. Layer Alignment

During the printed-circuit board manufacturing, there comes up the tolerances as a result of alignment above the artwork film for all the layers. The alignment tolerance includes silkscreen, drilling, track polygon planes and the solder masking. If the designer does not apply the layer alignment, it ends with a serious problem because of the tolerance unclarity. Therefore, the communication between the designer and the manufacturer should be well done for the information of the alignment tolerances that can be obtained for the design.

2.1.10. Netlists

Netlist is the general PCB name of the bunch of connections that is identified in the schematic of the printed-circuit board as done in this project. Netlists include all the component designators, each footprint, component lists and all the other related information for the schematic. The designer can create a file for the netlist by using the schematic packages. After that, the netlist files can be directly imported by the printed-circuit board package and all the needed components are added into the blank board. With the help of this, the auto routing can be done and it will save much time for the designer. Looking for the match of the component will be removed from the PCB design steps. Therefore, more safe connections will be made.

2.1.11. Rats Nest

Rats net is excessively needed for the moment of component placements. Especially, the designers need rats nest for the complex designs. Rats nest are actually hints for a designer to see the connection places between the component pads. The algorithm works according to the connection done in the schematic section of the design. The demonstration is done with straight lines. Whenever the designer moves the component, these straight lines will be moved too so the connection visibility will be even way easier to recognize.

2.1.12. Power Planes

Power planes are preferred by the designer to be able to distribute the power through the printed-circuit board. When power planes are used in the design, the wiring inductance of the power will be decreased as the impedance. This is quite important for the designs that have high speed. In addition to single sided boards, power planes can be preferred also for the double sided boards. However, designer should check the signal tracks if they are put on the top layer.

By saying power plane, it is actually indicated a layer made of a solid copper for the power rails or the ground rails. Sometimes, even for both of them. These plane reaches the middle layers of the printed-circuit board whose parts are next to more outer surfaces. For the complex four layer design such as a design done for this thesis project, it is a common beneficial solution to choose one layer for the ground planes and one layer for the all power planes. Since the ground is a signal line, ground planes are put firstly rather than starting with a power planes.

For the complex designs, distributing the power planes by laying the tracks down is an another way to make the power planes simpler. This is mostly preferred when the digital ground and the analog ground need to be separated. If the halves of the plane on the both sides are connected mistakenly, designer causes an undesired situation which is power loop. Therefore, the tracks are supposed to be placed all around the board's outer edges. It will be guaranteed that the power planes are not able to pass the outside of the edges. An alternative for a designer not to create a power plane if not desired is that designer can prefer a common signal layer with the lay down copper. After that, tracks can be used manually too.

2.1.13. Better Grounding

In PCB design, grounding is always counted as a fundamental action. With the good grounding style, the printed-circuit board design can be the most effective one. However, on the other hand, with the bad style, the PCB can even be useless. During the printed-circuit board design, there some extremely effective techniques that need to be paid attention. Here is the list of these techniques:

- The copper should be used as much as possible for the ground path. The more copper the designer prefers for the ground, the less the impedance becomes.
- For the better grounding, the designer should use as many polygon planes as possible.
- For the multi-layer printed-circuit board as the project for the thesis, one of the layers should be defined for the ground planes. Generally, this layer becomes the next to top layer.
- For the critical sections of the design project, designer should add distinguished ground paths. This is also named as a star grounding. In this way, the track of the ground runs out by starting from the central dot and it seems as a star. The name itself come from the appearance of the action.
- To be able to reduce the impedance of the trace for the ground, designer should prefer as much via as possible.

2.1.14. Better ByPassing

The bypassing should be applied for the components and for the points that creates switching current. To be able to apply bypassing, capacitors are used through the power rails. In general, when the designer wants to have a typical bypassing, the value for the capacitor is 100nF. However, it is also possible to have more than one different type of capacitors with different values by providing a parallel connection.

From the practical perspective, generally every integrated circuit has minimum one bypass capacitor. According to the frequency, the capacitor value may vary. The general preferred capacitor for the bypassing in high frequencies are 1nF and 10nF, while in less frequencies, these values are 10uF and 1uF.

2.1.15. Design Technique for The High Frequency

As discussed in the studies [1] and [4], some of the designs require high frequency and in this condition, the designer must pay attention to the parasitic capacitance's influence. In addition to capacitance, the inductance and the impedance will be watched out too. When the tracks are long and the signals are very fast, signal integrity issues can be obtained. To be able to prevent such undesired conclusions, the proper technique for the transmission line should be used or the proper track length should be selected.

For the designs that include high speed, to protect the signal integrity, ground planes are quite logical to use. With the help of it, the transmission lines for the impedances will be kept under control. However, rather than getting into the such details, there are two easier and more famous ways for the PCB design which are stripline and microstrip line. If there is a trace above the top layer and if there is a ground plane under the trace, this type is microstrip line. When the designer wants to calculate the characteristic information as impedance, it becomes quite complicated in the microstrip line because of the trace's thickness and width and the highness of the ground plane. PCB's structure material's relative permittivity is one of them too. These are the reasons why designer should consider to maintain all the ground planes as together and as close as possible to the first layer, in other words, to top layer. In addition to microstrip line, the stripline is very similar. However, in stripline, there is an extra ground plane above the trace. Therefore, the trace is supposed to be above the inner layer. When the comparison is made between these two lines, the stripline is more positive since the radiations will be consumed in the ground planes in the stripline.

For a designer, there are some rules that needs to be considered while designing a high frequency designs. Here is presented list for the rules:

- The tracks include high frequencies should be kept as short as possible.
- For each power pin, putting a decoupling capacitor is a mostly preferred technique.
- Not to have discontinuity, above the cutout in the ground planes, the high frequency tracks should not be run.
- If possible, according to the design, first of all, designer should add bypass capacitor to the integrated circuit power pin. After this, the switching noise in the plane will be decreased.
- Designer should always consider that the used vias are going to create discontinuity for the

transmission line’s characteristic impedance.

- To be able to reduce the crosstalk between traces on the ground, the location difference between the trace and the plane needs to be decreased and the trace distances should be increased.
- Since the via radius can be varied according to the track, designer should try to adjust vias as small as possible so that less parasitic inductance can be obtained in high frequency project.
- In the high frequency applications, the main power input connector should not be connected to the power plane. Via should be used for the filter capacitors.

2.1.16. Double-Sided Loading Selection

By saying as double-sided loading, it is actually meant placing the project components on the both sides of the printed-circuit board. There can comes up two advantageous. The first one is about the limited board size. When the board size and the shape has a limitation, the double-sided loading technique will work a lot. When the board size or the shape is not efficient for components for the placement, this technique will help designer to put the components on the both sides of the board. The second advantage for the technique is about the bypass capacitors. When the high speed surface is preferred, generally there becomes no space for the required bypass capacitors or the placement does not become efficient. Therefore, these issues will be solved by this technique.

2.1.17. Submitting The Design for The Manufacture

This step is the last section of the entire printed-circuit board design. Designer must let the manufacturer know about the specific information. Therefore, the first thing designer should send to manufacturer is Gerber files. In addition to gerber files, here are the list of information that every manufacturer has to know for the production process of the printed-circuit board.

- A specific reference code for the project
- Required board manufacturing time.
- The number of boards needed.
- Thickness of the board

- The board types.
- The layer number that designer created for the PCB.
- The surface finish.
- The colours of the overlay and solder mask on the board
- Weight of the copper
- If electrical testing is needed or not. As done in this thesis project, for the multi-layer boards, it is highly recommended.
- The clearance ratio for the space and the track
- The dimensions/shape of the board
- If the designer wants the board panelised or cut individually

2.2. Printed-Circuit Board Design with Altium Designer Software

Altium Designer is a software platform which helps required tools to get collected and work with a proper functionality to be able to provide an electronic product development. With the help of the Altium Designer software, different design tasks can be achieved such as circuit simulation, HDL design, printed-circuit board design and signal integrity analysis. During this thesis, the Altium Designer 21 version has been used. Although the steps of a printed-circuit board will be explained step by step in the next chapters, the PCB design in Altium Designer will be summarized in this section by separating into steps. The explanation made in the studies [9] and [10] will be summarized in this section too.

2.2.1. Schematic Library

The very first step of the printed-circuit board design is the step where every component is created and classified according to project requirement. For the thesis project, none of the components were given by the company. Therefore, every component is chosen according to the circuits of the each section of the project. In schematic section, designers generally illustrate the integrated circuits as a box whereas the passive elements are drawn by the designer's preference. After drawing the representation of the component, specific information of the components must be inserted into the software for the manufacturing part when the PCB is ready. The inserted component information are manufacturer, manufacturer number, supplier, supplier number and the mounting style respectively. It is quite important that these information must be filled in the Schematic Library section of the design so that when the circuits are ready to pass to PCB, manufacturer and the designer can find the correct components easily by using these information. In the PCB manufacturing industry in Italia, the most common websites used for the components are MOUSER, DIGI-KEY and SAMTEC. In these websites, the designer can have the component information and insert it into their design in the schematic library section of the PCB.

2.2.2. Schematic

The second part of the printed-circuit board is called schematic. In schematic section, all the created components in schematic library are placed. According to the circuits desired, the components can be connected to eachother with the help of wires. The important point in schematic section is making a connection between different schematics sections. For instance, in this thesis, there will be four different schematic parts according to the each section of the project. Providing a connection between these section matters and can be happened with the help of net labellings. However, it must be paid attention that the added net label to the pin must have the same name in other section's pin. That is how the software senses the pin's of the circuits and then, contacts the schematic of the other sections. Whenever the designer approves the created circuit diagrams, the entire schematic must be validated and annotated. Validation is simulating the circuit and seeing the errors if any. Annotation part is essential especially for the PCB part of the design. The duty of annotation is ordering the same components by adjusting the designators. For instance, in a circuit diagram, there becomes more than one same or different resistors. To be able to validate the design, the annotation should be completed. As a result of annotation, the resistor designators will be ordered as R1, R2, R3 etc. In brief, the schematic section of printed-circuit board design consists of component placement, wiring, validating and annotation.

2.2.3. PCB Library

Printed-circuit board library is the third step of the PCB design in Altium Designer software. PCB library can also be counted as a part where the research and hard work can prove itself. In this section of the design, the designer must know that every component that was chosen in the previous sections should have a official datasheet. All the components must have their own unique footprint dimensions in these datasheets. To have these measurements, the datasheets of the components must be searched from their manufacturer's or supplier's websites and the dimensions of the PCB layout must be found. Moreover, these dimensions need to be drawn in the Altium software.

During the design, there were different types of same category components with the same package types. In such cases, there is no any reason to draw footprints for every single components because as long as the components' package numbers are the same, the footprint will be the same for all of them. This is an important point that PCB designers should know before attempting to design a PCB.

The second essential point that has to be completed during the PCB design is that as soon as the footprints of each component are prepared in the PCB library, designer go back to first part which is Schematic Library and upload the footprints to the related component. For instance, in the thesis's PCB library, one of the created footprints belonged to 0603 packaged number resistor/capacitor. When the 0603 packaged numbered resistor or capacitor's footprint is ready, the same component should be found in the schematic library and the footprint must be added there. The reason why designer must complete this process is that this is the only way to place all the components that we used in our circuits into the last- section of the design which is called printed-circuit board section.

2.2.4. PCB

Printed-circuit board is the last section of the PCB design. In this section, every created component will be placed on a copper surface. As expressed, and illustrated in the next chapters in the thesis, every component in the PCB section is actually the footprint images of each component that PCB designer drawn according to the component's official datasheets in the PCB library section. When the PCB section is opened, the first step that must be done is placing the components on the board. In this part, much attention should be paid because the component placement can affect the board functionality and efficiency directly and, in this case, every component datasheets should be checked one more time to see all the limitations for the placements.

2.3. Communication Protocols

Hardware communication protocols play a huge role to provide a communication between the devices. This communication organization is designed in different styles by depending on the requirements. Since the main purpose is to have a successful communication, each protocol is preferred in specific fields. In this section of the thesis, the used communication protocols for the thesis will be discussed respectively.

2.3.1. Universal Asynchronous Receiver-Transmitter (UART)

As indicated in the research [5] and [6], universal asynchronous receiver-transmitter is a communication protocol type that is preferred as one of the most. UART is a device to device hardware protocol. The beneficial point about UART is that this protocol is able to work different kinds of serial protocols which includes receiving and transmitting data. During the serial communication, transfer of the data is obtained by having a single wire and this transfer happens bit by bit. During the two way communication style, to be able to obtain a certain data transfer, two wires are used. The structure of the wire and the circuitry depends on the application requirements. Therefore, if reducing the cost is the aim during the protocol application, less complicated circuitry is preferred.

UART is mostly used by the microcontrollers and embedded systems for device to device communication. Although this protocol is highly used, it does not become optimized completely all the time. As understood from the full name itself, to synchronize the bits of the output, there is no any clock signals. The UART's transmitting is connected to the controlled bus which is sending the data in the parallel way. After that, data is able to be transmitted serially through the wire bit by bit until it reaches to UART's receiver. The transmission mode of the protocol is in a form of packets. The small piece which provides a connection between the transmitter and the receiver has serial packet's creation. This packet includes start bit, stop bot, parity bit and the data frame. The communication is provided with the help of stages of these bits.

2.3.2. Serial Peripheral Interface (SPI)

As discussed in [6] and [8], SPI is serial and also synchronous interface. By saying synchronous, for the transfer and the receiver of the data, clock signal is needed. Between the master and the slave, clock signal becomes synchronized. Therefore, in this stage, SPI is differentiated from UART because UART is asynchronous communication protocol type. The clock signal checks when the data needs to be sent for the slave and when it is going to be read. Only, master device is able to check the clock. Therefore, only the master device is able to supply a clock signal for all of the slaves. Without clock, transfer of data can not be happening. During the protocol, both of the master and the slave are able to exchange the data between themselves. This exchange happens during the clock signal's rising and falling edge. In general, such communication protocol type includes three signals or four signals. The duty of the master device is data frame's initiation. This device also has a power to choose slave device too for which the data is going to be transferred to. The most common devices that can work with SPI protocol efficiently are RFID modules, SD cards and wireless transmitter-receiver modules that has mostly 2.4GHz.

2.3.3. Inter-Integrated Circuit (I2C)

I2C is one of the serial communication protocol types which provides a connection between the multiple devices and the single bus. I2C is mainly preferred in embedded system fields. Also, when the less amount of devices need a communication, this protocol is preferred firstly.

As explained in the research [7], the data transfer between the devices is obtained either parallel way or serial way through the medium. Therefore, I2C is a perfect choice for microcontrollers since these devices use serial protocols for the communication with the devices externally as displays and the sensors. I2C is preferred for the communications that requires only short distance. Since this protocol needs only two wires, it is highly recommended to have I2C in the projects that aims to have a communication between many devices, because for the entire communication, only two wires will be needed. It creates such an advantage for the complex connections.

.

III. LITERATURE

3.1. PCB Studies

Printed-circuit board has always been one of the main focus in the engineering field. During years, much research has been done about the printed-circuit board with different concepts. Some scientists pad attention the relation of the design of printed-circuit board with external components, some of them focused on the functionality of the board, some directly searched the relation of printed-circuit board with integrated circuits, some of them wanted to know more about the efficiency while some wanted to succeed reducing the possible issues that comes up by the PCB. In brief, from time to time, printed-circuit boards have always searched by the researchers with different concepts. Thanks to researchers who did an amazing job about the PCB and published their work for the public. In this section, with the light of this contribution, some of the important research done so far will be summarized before passing to the last part of the section.

As indicated in the research [4], one of the research projects done was a printed-circuit design with the high speed. Printed-circuit boards with high speed always require much more care because of the operation frequency and decreased rise time. The main key part of the working PCB with high speed is about signal integrity. However, obtaining the signal integrity can be a tricky part. For example, designer has to examine the communication of the data bus and interconnects when the bandwidth has an increased case. When the rise time is decreased, the frequency of the signal increases. This is a common knowledge also indicated in the Fourier analysis. When the frequency becomes high enough, the capacitance, resistance and the inductance will be delayed like it moves to load from the source. These parameters need to be controlled and designed carefully so that the signal does not fail completely.

To be able to have the motherboard, different types of integrated circuits are supposed to be interconnected with different functions. With the help of it, these will be communicated, and the data will be exchanged. When the integrated circuits are added above the printed-circuit board, they become connected by copper trace.

As indicated in the research [11], PCB design with the real world EMI control has been searched as an another research topic. Well-designed PCBs can create a difference between the emission requirement among the first cycle. EMC designs which are traditionally focused, are rule based designs. PCB designers implement the rule of thumb lists. When such lists are hard to implement by the designers, designers generally ignore them. When the product is manufactured, product will not provide the emission requirements and there will be lots of time-consuming cases. In addition to this, it will be a waste of money case too . If the designer wants a good EMC design, designer needs to pay attention to know about EMC emissions' potential causes and its basic understanding. Designer does not need deep mathematics for that. With the light of the EMC emissions' potential causes, decisions for the trade-off can be made to obtain the optimal EMC design. When the designer considers all of the potential sources, the design will be passed to requirements of the emission in a laboratory.

Another research was done in the paper [12] which indicate mainly the PCBs' environmental impact by considering the sustainability. According to the research, printed circuit boards are used for the temperature exchanging equipment as heat pumps, air-conditioners and the refrigerators. Also, these boards have an impact on equipment for the households like dishwashers, printing machines, and vacuums. The other impact of PCB on the environment is about the smaller households such as electronic toys for the kids, cameras, and the shavers. The last example impact of the printed-circuit board in the environment is for the ICT field. The effects of these boards in ICT is observed in laptops, calculators, smart phones, PCs and the telephones. The amount of the waste for devices increases because of the rising demand and the obsolescence. Here is the shared figure that shows the waste printed-circuit board amount according to the 2021 studies.

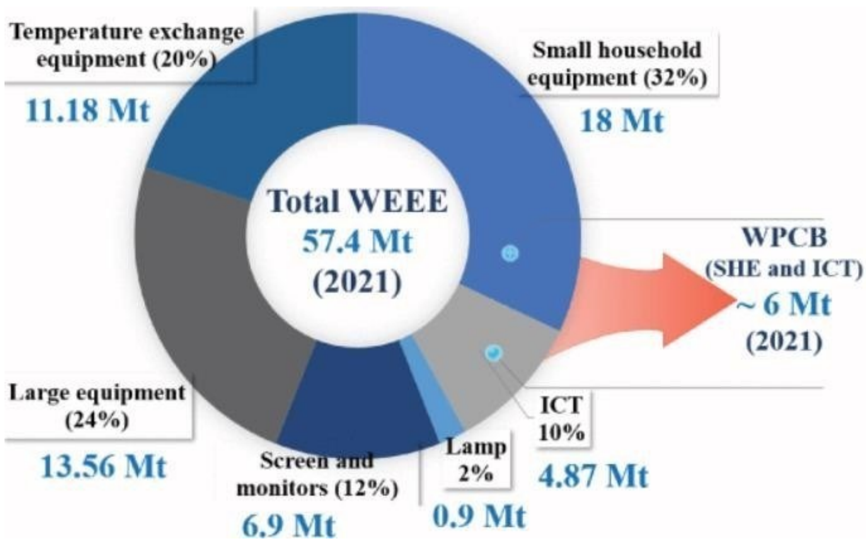

Figure 1: Waste PCB Amount [12]

The main materials and their effect in the environment are also discussed in [12] and the evaluation of the material selections was mentioned in the study. The case studies have been completed by applying the smartphone. In smartphones, the material compositions were showed to differentiate the manufacturer age and smartphone types. The data was analyzed from the material compositions by averaging. However, in brief, it can be summarized that in PCB, the material selection affects every stage of the environmental impact by considering every single element used.

In another research about the printed-circuit boards which is represented in the paper [13], the printed-circuit boards' effects are discussed from the sustainability perspective. In today's world, increased quality of modern life requires better equipment and more technological developments. When it comes to the PCB, which represents the main base of the electronics world, can be counted as technological waste since printed circuit boards are quite hard to be recycled and expensive. In addition to this, their materials' diversity range is very wide. On the contrary, the study supports that the regulations especially in Europe, support the reduction of negative impact of such devices on environment, and increase the chance of recycling. Therefore, in today's world, when the electronic device as PCB is designed and manufactured, the society wants to be sure that the sustainability condition will not be harmed during any of the steps. Printed-circuit board's fixing problem can be expressed as migrating from the alloys such as Pb-Sn to the free alloys. With the light of this replacement attempt, the toxicity of Pb is planned to be reduced. Here is the brief demonstration below that shows the PCB's life-cycle from the sustainability perspective.

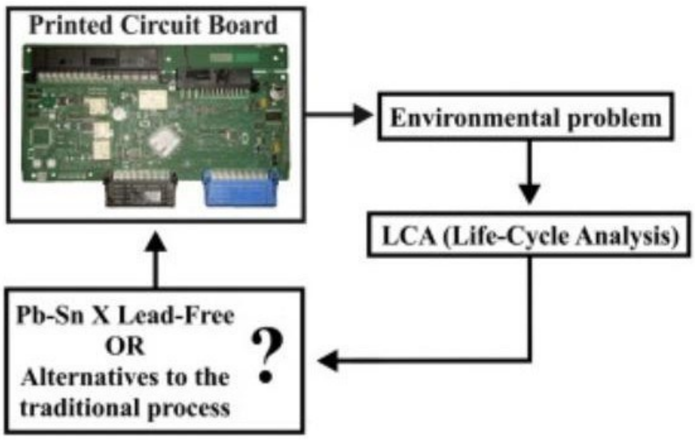

Figure 2: Life-Cycle of a PCB [13]

IV. CASE STUDY

The case study for the thesis is a design of a control board of the quadcopter that has to be multi-layer board with four-layers, needs to provide three different communication protocols and will reduce the most common challenges of PCB design during the process of routing and mounting.

Firstly, the mechanical design of the project is drawn and given below as a general understanding.

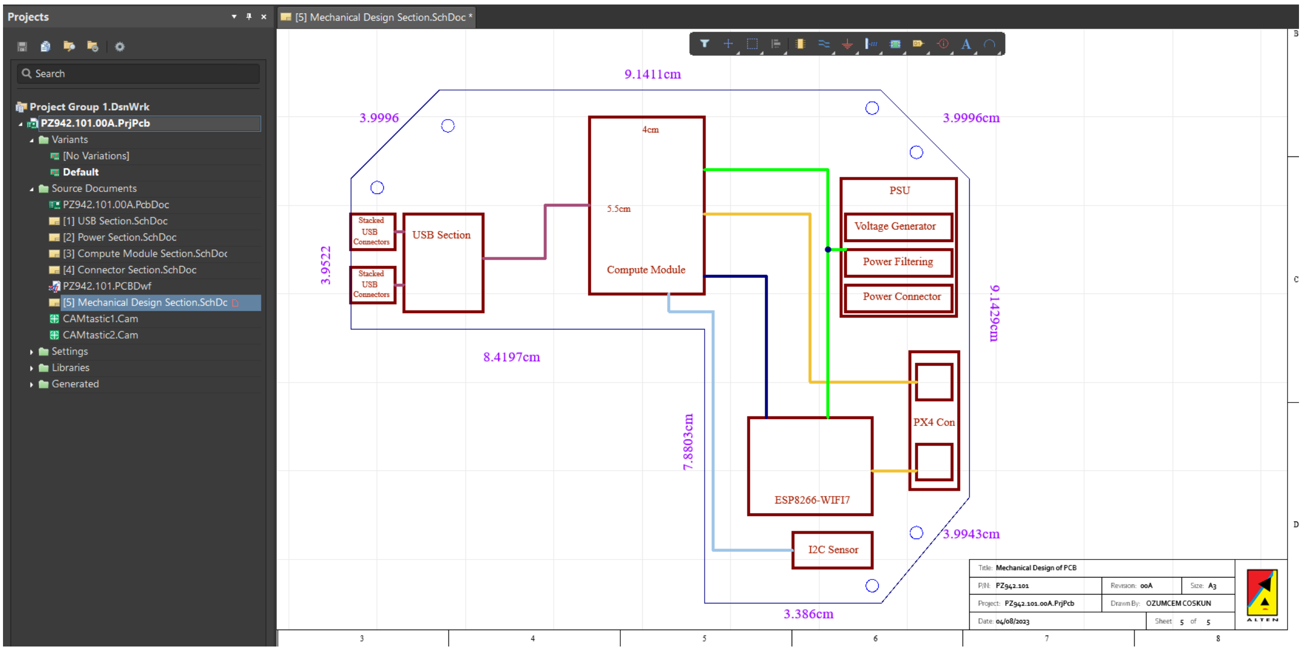

Figure 3: Printed-Circuit Board's Shape and Measurements

As seen in the figure, at first, the requirement given by the company ALTEN Italia was the board shape and the measurement of the edges. It is also known that the quadcopter will have an external autopilot out of the board while there will an external LED display, external stack USB ports and the wifi module on the board. Therefore, the PCB design should support all these connections by choosing the most proper components.

As a start, the steps for the PCB design are classified and ordered as:

- Power Supply Unit
- USB Section

- Connector Section
- Compute Module

4.1. Power Supply Unit (PSU)

Power supply unit is the main part of the entire design since all the power will be supplied from this section as expressed quite detailed in the study [14]. For this section, since the preferred compute module for the board is raspberry pi compute module 4, the official raspberry pi 4 and its compute module's datasheets are checked carefully. The main reason to do that is in such official files, the placement examples of the each sections are indicated for the designers. As a result of the official datasheets and ALTEN Italia's preferences, the following steps in the schematic are power supply part, filtering part, and the voltage regulator part. Here is the created the schematic of power supply unit according to the official datasheets and the company's preferences.

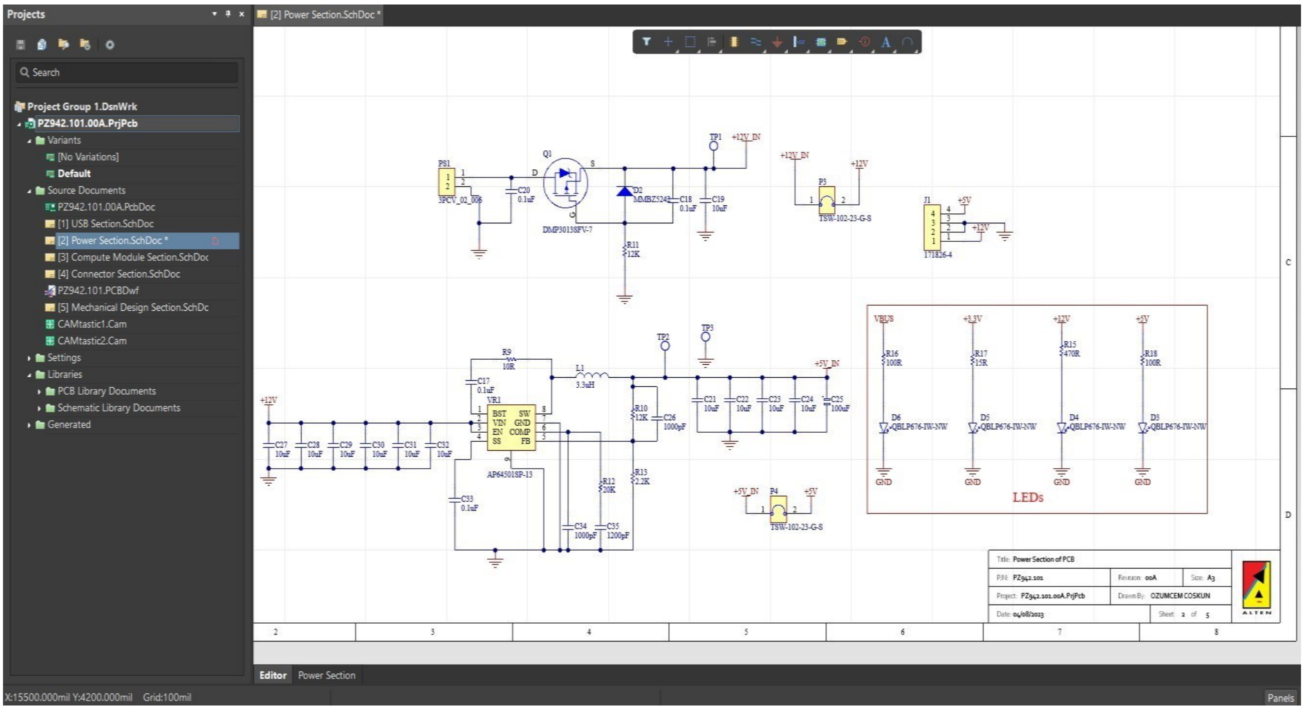

Figure 4: Power Supply Unit in Schematic Section

As seen in the figure, the first step is filtering with the MOSFET and other components. When the filtering is completed, a switch is added not to risk the rest of the system. The voltage regulator decreases the voltage amount and 5V output voltage for the power supply unit is obtained. After drawing the power supply unit on the official Altium Designer's schematic as in the figure above, the design has validated to see any came up error messages. Although the software can continue even if there are errors in the schematic, for the rest of the design, it can be a serious problem as having burns on the components. Therefore, all the errors in the message list should be fixed, validated again and then, should be moved on.

4.1.1. Schematic Library of Power Supply Unit

Schematic library is the first step of any PCB design. In this section, the components are selected according to their package number, their tolerances and their qualities. For instance, if the resistor is aimed to create, it is important to know the resistance value, the package number of the resistor and the tolerance percentage of the resistor. Although these three parts are the most important steps for the component selection, it is also essential to find the information of manufacturer, manufacturer number, supplier, supplier number and the mounting style for both the schematic library section of the design and also for the PCB manufacturing part which is the end of the procedure. When all the information is obtained by the designer, the schematic library section of the design starts.

While the passive components as resistors and capacitors are drawn by the PCB designers in their original shapes, the other components such as ICs are illustrated as square/rectangular boxes as below. Each component's official datasheet must be checked and the number of pins on the pads must be known. After the step, as seen below, in the property section of the Altium Designer software, the information should be filled to describe the component. For example, in below, the voltage regulator used in the power supply unit is drawn. In the datasheet of this regulator, it is indicated that regulator has nine pins with pin names and with each pin functionality. This process must be done for the every single chosen component by the designer that each section has respectively. Here are the two component examples of the power supply unit in schematic library section of this thesis project.

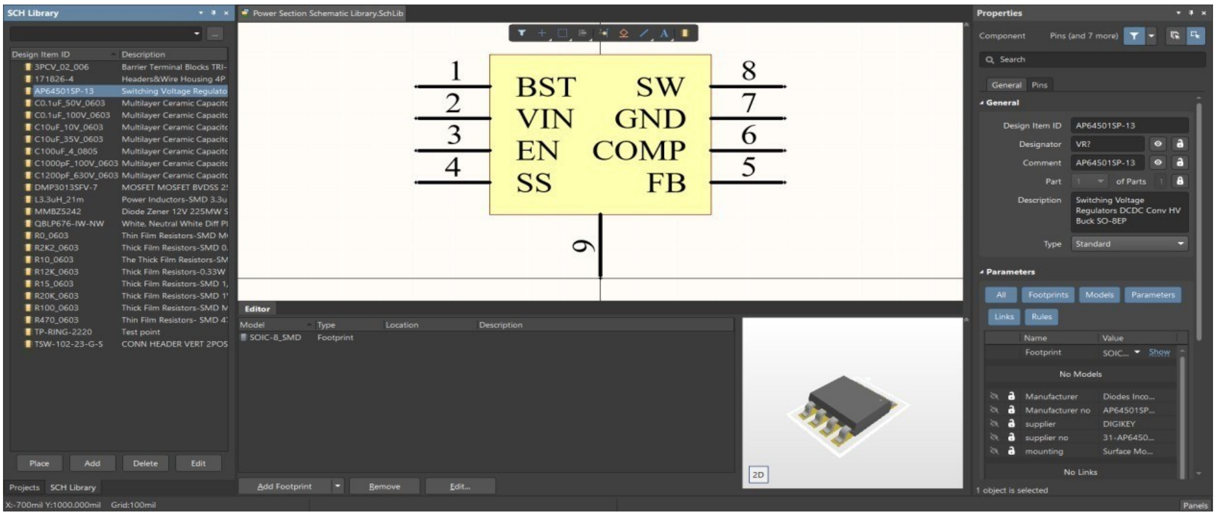

Figure 5: Voltage Regulator Design in Schematic Library

As seen in the figure, while filling the information of the components, it has to be known the global representation of each component as symbols. For instance, while the global representation of the resistor is R, the voltage regulator is represented as VR. However, in the designator section, when the representations are written, it is mandatory to add also a question mark next to the representations as in figure's designator part on the right side. The main reason for PCB designer to do it is when all of the components are ready to pass to the the last section of PCB, the software will order the same representations in a sequence. If there is no question mark, the designer has to number every single component manually and for the complex circuit systems like this project, it can be very chaotic and causes a high potential of risk for the PCB design since the integrated circuits don't understand which components to sense. Therefore, it is highly recommended that when the components are described, question mark with the global representation should be added into the property section as recommended in the study [3].

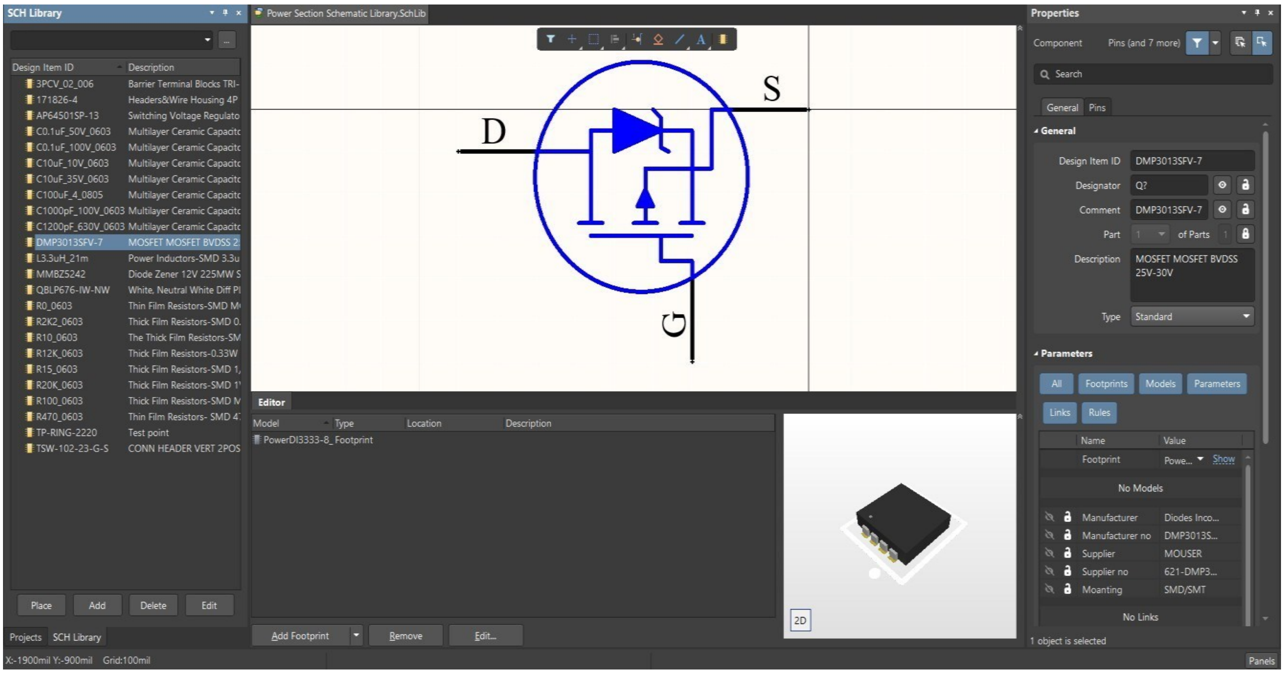

Figure 6: MOSFET Design in Schematic Library

The second drawn component example is MOSFET as shown above. Since the MOSFET has drain, gate and source parts, these parts are shown with three separated pins. The pin connection to the big circuit will be made according to component's datasheet recommendation. For instance, voltage regulator datasheet recommends designer to put the input capacitors as close as possible to the input section. Similarly, the MOSFET datasheets recommends to do the placements by considering the drain and source, not the gate.

When the components are selected and all the required information is filled in the related schematic library part, the schematic section must be validated so that this part can sense the inserted information of the components. When the validation is completed, the other step should be applied that is called annotation. To have a proper working printed-circuit board, annotation should be completed in this schematic section. By saying annotation, it is a step where multiple number of same categorized components have the same general name. To prevent this chaotic case for the PCB design, this step is mandatory. For instance, in our power supply unit section, we have lots of resistors. Software registers the resistors with their global symbolic letter which is R. However, since there are many resistors, there has to be order. After annotation, the resistors are order automatically as R1, R2 etc. Otherwise, the routing and the PCB design is not possible to achieve.

The other essential part is having changes in this step. It is very common for a designer to change the component for the upcoming times during the project. Changing the component means having changes in schematic library and replacing all the component information with the new one's. To let our PCB understand the changes, the schematic library should be updated and the schematic section has to be validated because after the validation process, message panel indicates every possible error that can damage the proper PCB design.

The way how the components will be mounted to the board in the upcoming parts is also indicated in this section. For instance, the mounting style for this MOSFET is SMD/SMT. Therefore, the manufacturer can understand that the mounting will not be a through-hole style.

4.2. USB Section

One of the requirements for the project is having stacked USB connectors with USB Hub. In this section, the USB connections for the hub and stacked connectors are presented. As recognized, there are two components in the schematics called U1A and U1B. Normally, these two components are actually one USB 2.0 Hub. However, to be able to be more flexible on the PCB, the voltage section is separated from the USB's other pins. In that case, it is very essential to put the capacitor group that is fed from the voltage source part of the USB as close as possible so that in PCB routing section, the tracking will be easy and the signal transportation will not be risky.

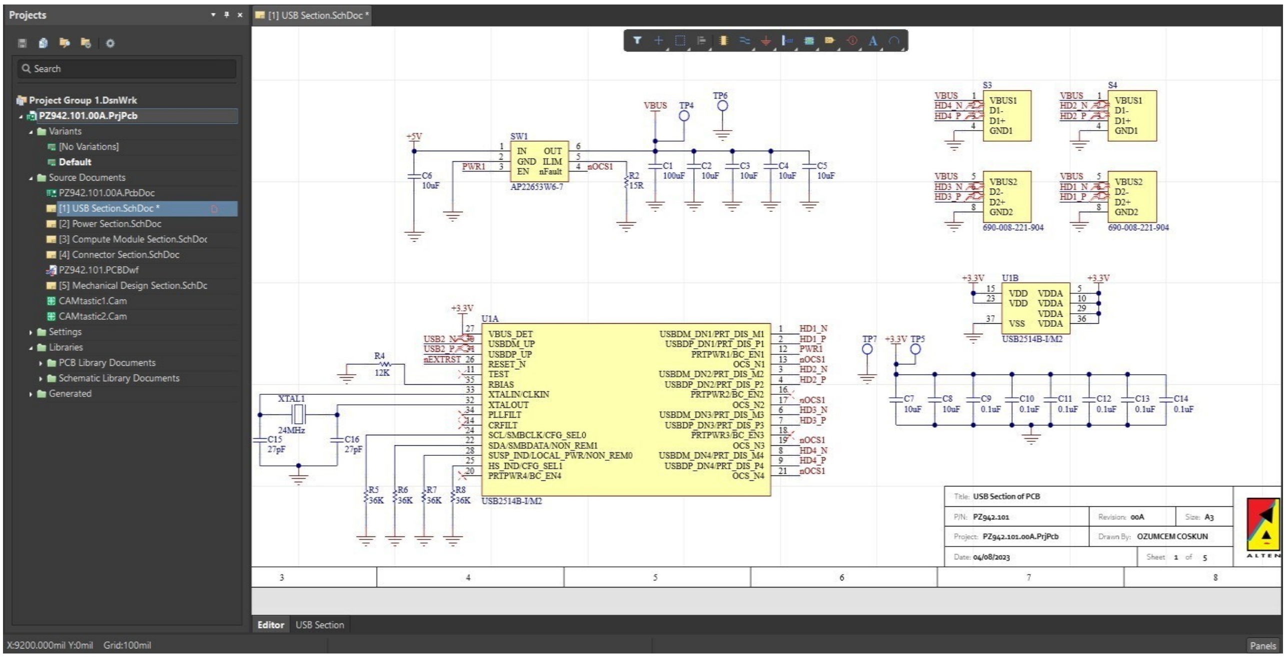

Figure 7: USB Part in Schematic Section

The other important case in USB design is to pay attention to differential signals. In PCB design, the tracks are generally divided as power tracks and signal tracks. In signal tracks, differential signals are the most important signal type. Therefore, in the schematic part, those differentials should be indicated and in the PCB section, the routings has to be started from the differential signals. If the placement in the PCB section does not provide a proper routing for the differential signals, the component placement must be changed and they should be repositioned until differential signals are routed directly in the selected layer.

In some part of the circuits, there are some test points are added as recognized in the schematic and they are represented as TP since the global representation of the test point is accepted like this. The reason of adding the test point is to prevent possible risks for the entire system. Since every section of the project is connected to each other with net labellings, the any possible error can directly affect the entire system. To be able to reduce the risk, when the circuitry is completed, before passing to the PCB design and manufacturing part, the connected components are needed to test in the laboratory environment by checking the test point pads. Since the ALTEN Italia research and development unit has its own hardware laboratory, the testings were made and moved on according to the results. It is also a good way for a PCB designer to understand if the components should be stayed the same or changed to have a better test result. In the section below, three different examples of component selection, component information and their representation are illustrated.

4.2.1. Schematic Library of USB Section

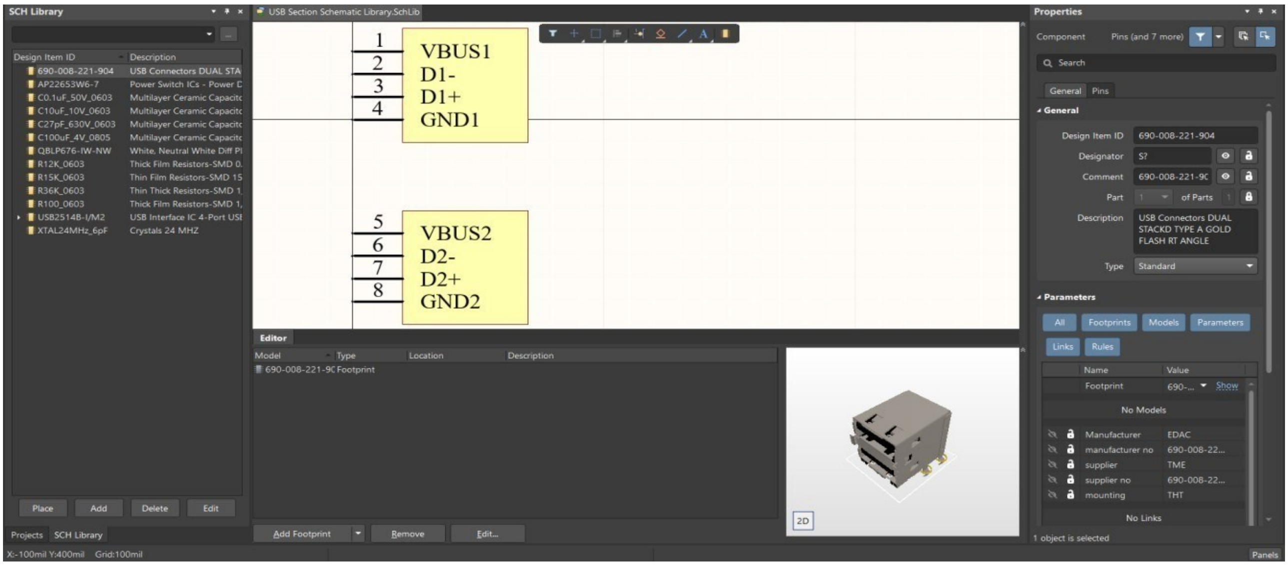

Figure 8: USB Stacked Connector Design in Schematic Library

The first component created for the design is stacked USB connectors. The official datasheet of stacked USB connector is checked and then, pin numbers, pin names and functionalities are obtained. As seen in the figure above, each USB connector has two USB entrance. Therefore, in the schematic library, each box represents one of the USB inputs. In every box, there is a voltage source and ground expectedly. In addition to this, there are pins called D1 and D2. These pins represents the differential signal in the USB section. Also, these pins are the ones which make a connection between stacked USB connectors and USB 2.0 Hub. Rather than using the direct wiring, net labelling is preferred. For the complicated circuits like this thesis design, it is more beneficial to show the connections with net labellings.

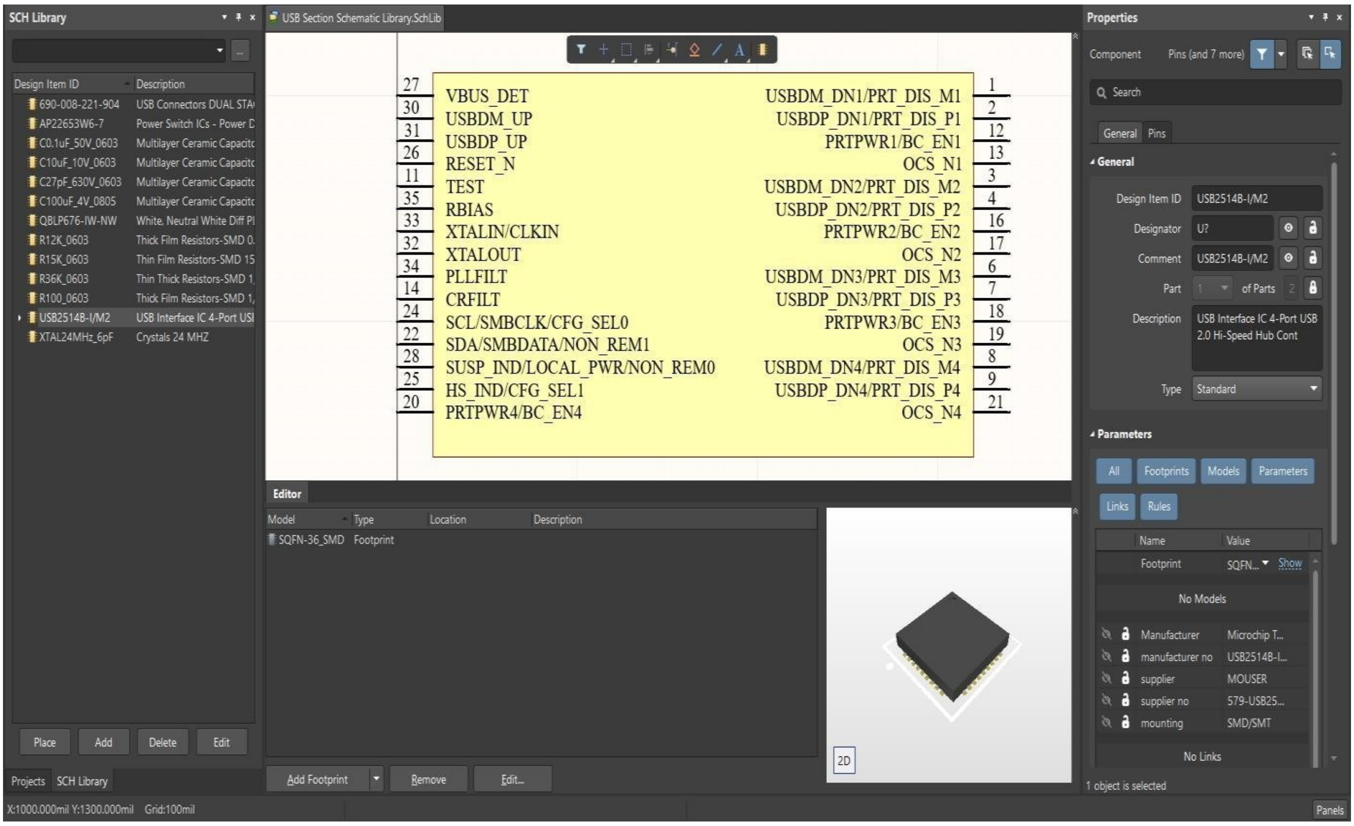

Figure 9: USB 2.0 Hub Design in Schematic Library

The figure above shows the USB 2.0 Hub representation after separating the voltage sources as another component. The component brand that is used as USB 2.0 Hub is USB2514B-I/M2. According to the official datasheet of the USB2514B-I/M2, the number of pins for the component is thirty. Each pin number and the pin names are adjusted according to the datasheet and its recommendation. With the light of such sources, the other components' connection with USB2514B-I/M2 becomes more solid and less risky. In PCB design, no matter how the components are perfectly suitable for the design, if the components are not placed well, the connection produces problems, and the results come with much less efficiency.

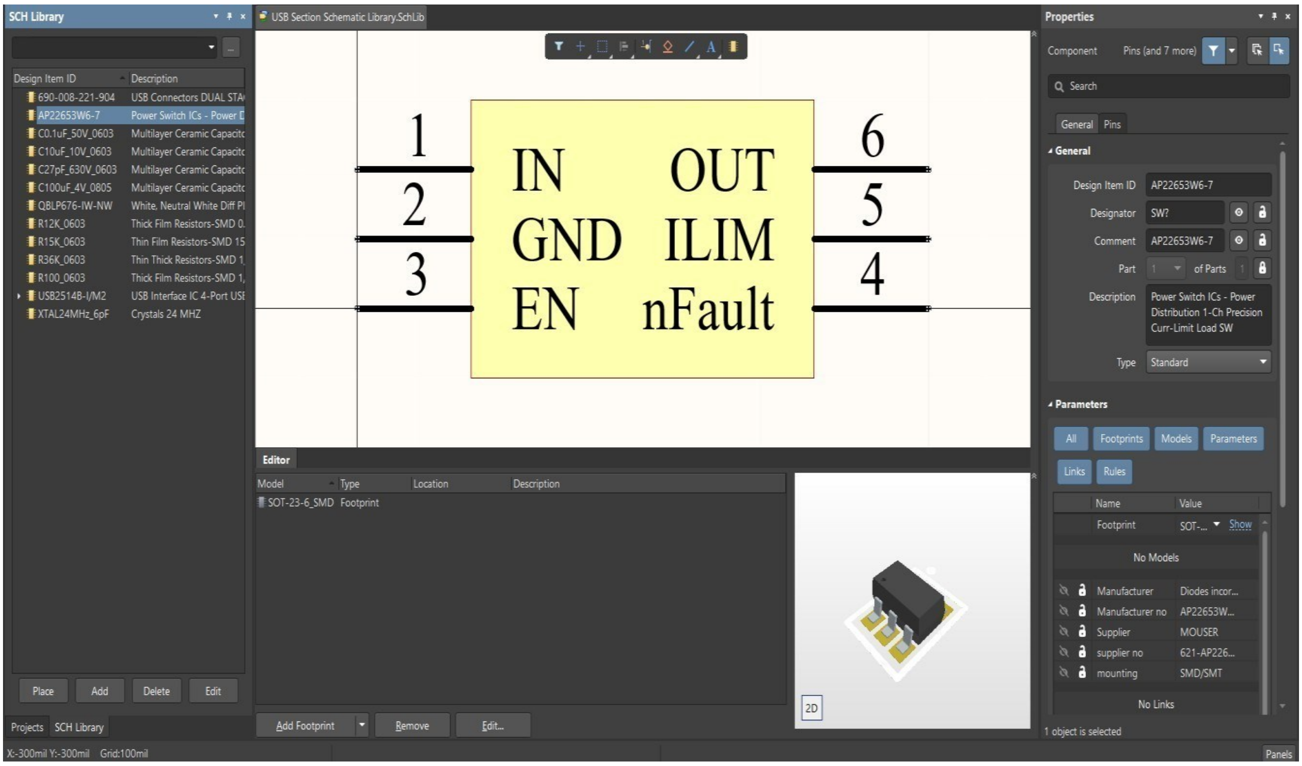

Figure 10: Integrated Circuit Design in the Schematic Library

The third example represents the Integrated circuit for the switch preferred for the USB section. As seen in the right side of the figure, when the manufacturer number is entered to the supplier's official website, the official datasheet of the product will be presented to the designers. In this datasheet, designer will have an idea about the pin names, number of pins, the product functionality and the placement of the product with the other components. In addition to this, test point is added just after the voltage source called VBUS as seen in the schematic. The main reason of this action is to see the voltage changes during the hardware test so that designer can see how the voltage division is happening through the output capacitors. Also, as seen in the schematic, the Enable input of the switch has a net labelling with a name called PWR1. As realized, the same label name is added to the 12^{th} pin of the USB2514B-I/M2 component. That means, the enable pin is directly connected to the USB2514B-I/M2 Hub's 12^{th} pin with a label called PWR1. The good part about net labelling method in the design is that if the components are in different sections rather than in the same one as in this example, that would still let the different components connect with each other. On the contrary, it would not be possible to connect such components by wire from schematic to schematic in the standard templates. Therefore, with the help of net labelling method, it does not matter how complicated the project is, the connection can be done in the simplest way. Net labelling method is set during the project between:

- USB section-Compute Module
- Compute Module-Power Supply Unit
- Compute Module-Wi-Fi Module
- Power Supply Unit-Wi-Fi Module
- Compute Module-PX4
- Wi-Fi Module-PX4
- Compute Module-LED Display

4.3. Compute Module Section of The PCB Design

The usage of a compute module was a request by the company for this research activity. Therefore, according to the project's purpose, especially to support the three different communication protocols, the best choice as a type of compute module is the Raspberry Pi 4B Compute Module. The Raspberry Pi 4B Compute Module properties are presented below:

- RAM with 4 GB
- Gigabit (Ethernet)
- 2.0 USB
- Interfaces for the 2 cameras,
- 2.4 GHz wireless LAN (dual band) and 5.0 Bluetooth
- eMMC (32 GB)
- 55 mm × 40 mm form factor
- GB memory
- 5V DC input power
- 0 to +80°C (for the operating temperature)

4.3.1. Schematic Library of the Compute Module Section

The designed connector for the raspberry pi compute module is illustrated in schematic library below. According to the datasheet of the compute module, there are 200 pins with different functionalities and names. Therefore, the designed connector must have the same pin number as the compute module to provide the surface mounting. The connector is represented as a rectangular box. However, as seen, there are two sections belong to module. There are two reasons to divide the one connector into two parts in the schematic library. The first reason is about the pin numbers. According to the compute module's mechanical design, there are two separated connectors under the compute module with certain distance. Therefore, to mount the module's connectors onto PCB, the connector pin numbers and the distance between the connectors must be the same. Otherwise, there would be no connection and the board would not serve the compute module. The second reason is that after focusing on the pin functions of the compute module in the datasheet, the pin numbers 1-100 are chosen for the main focus of the project which is supporting three different communication protocols. The second part's pin numbers which are 101-200 are selected for the differential signal connection between the compute module and the USB section. Although the communication protocols will be explained very detailly in the connector section of the thesis, the net labelling used for the communication protocols on the compute module connector are MOSI, MISO, CE, SCL1, SDA1, SCLK, RX_MODULE and TX_MODULE. These net labels can be adjusted with different names too. However, keeping the original protocol pin names is always a better idea for the upcoming parts of the project.

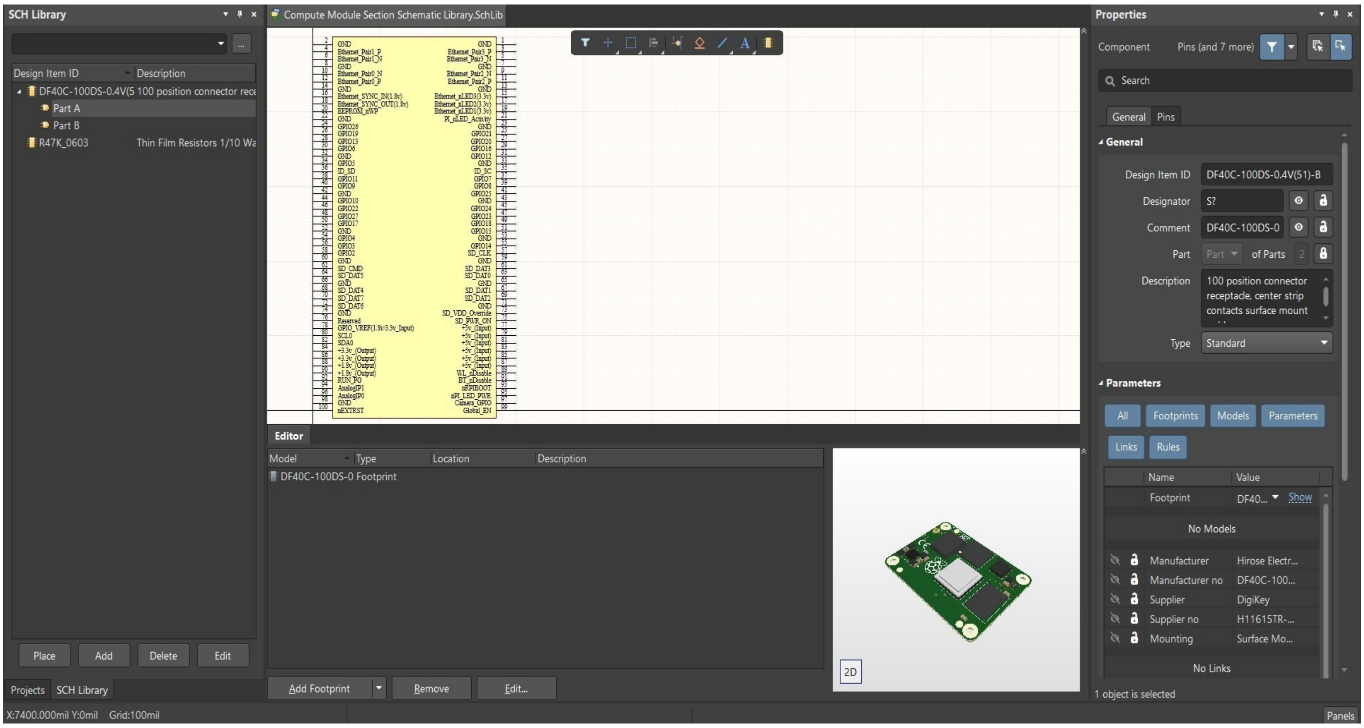

Figure 11: Design of a First Half of the Compute Module Connector

The connector part number that is planned to be mounted on the PCB for the compute module placement is DF40C-100DS-0.4V(51). To see the full details of the connectors, searching for the manufacturer number online would be more than enough.

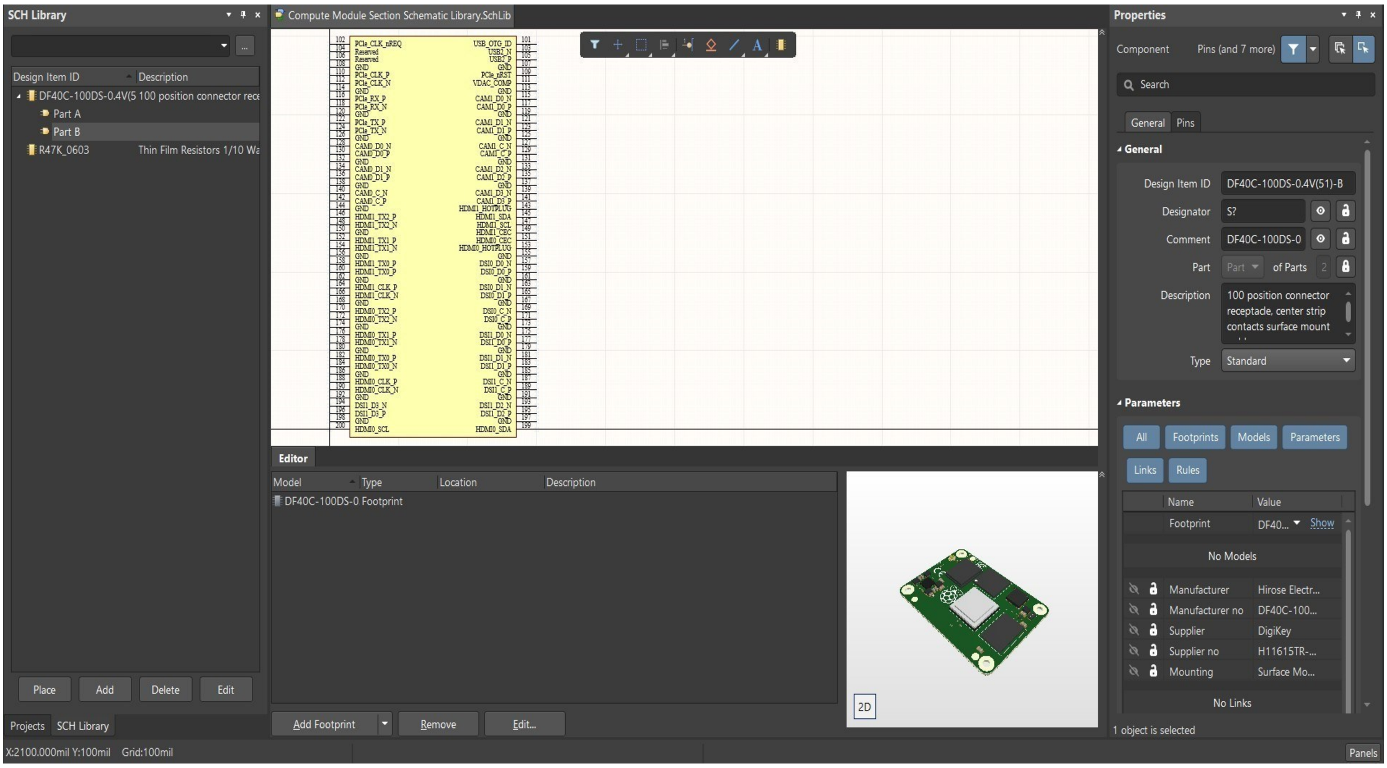

Figure 12: Design of a Second Half of the Compute Module Connector

4.4. Connector Section of the PCB

4.4.1. Schematic Library of the Connector Section

As a main aim of this thesis project, the desired control board for a quadcopter has to have a base that can provide three different communication protocols. These communication protocols are UART, SPI and I2C respectively. Each of these protocols will be used for a specific reason. Although the communication protocols have explained earlier in the thesis, they will be summarized specifically with these three different connectors in this section.

First communication protocol that was decided to be provided for the quadcopter is UART. UART is decided to be used specifically for the PX4 autopilot section of the device. However, as realized in the figure, there are two same PX4 connectors created for the design. While one of the connectors is created for the Wifi module, the other connector will be used for the raspberry pi compute module.

Therefore, it can be understood that PX4 required a wifi module and it has to have a connection with the compute module for the communication.

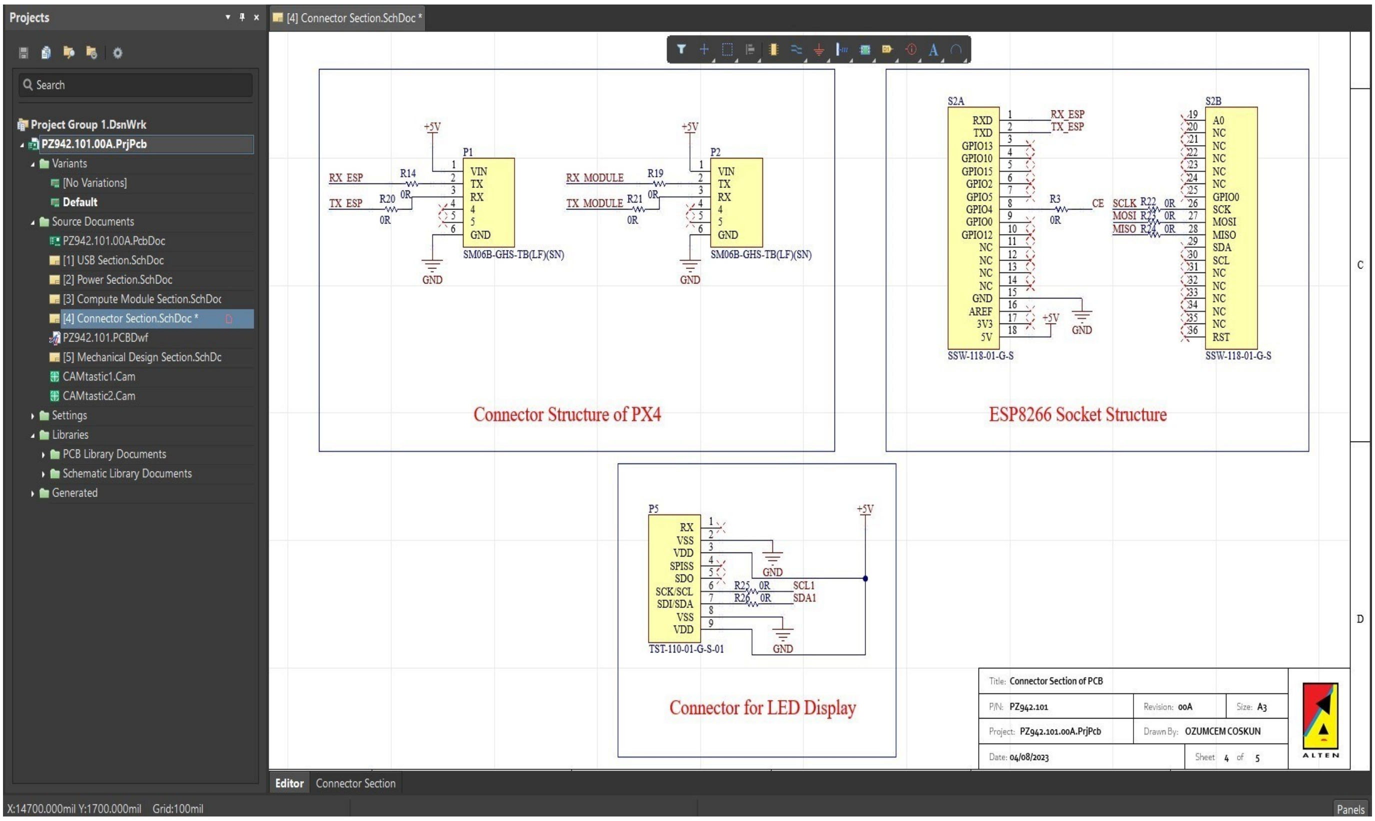

Figure 13: Entire Connector Part Design in Schematic Section

The methology is directly used for the connection of a PX4 connector with the compute module. The part that is represented in the schematic above and called as "Connector structure of PX4" has its pin names adjusted. The second TX pin of the connector is labelled with a name RX_MODULE while the third RX pin of the connector is labelled with a name TX_MODULE. At the same time, to provide the communication between these two components, the same labels must be added to compute module itself too. The real challenge appears in that part. To find the correct pin in the compute module that will support UART communication protocol for the PX4, entire raspberry pi and raspberry pi compute module official datasheet must be paid attention by the designer. In the raspberry pi 4B compute module datasheet, the alternative function assignments are indicated with pin names, and five different available alternative options. The designer can also see the general case of the pull for each pin.

After doing research for every alternative pin that raspberry pi 4 compute module has, TX_MODULE net is combined with the GPIO14 alternative pin of the raspberry pi compute module. With this action, the receiver pin of the PX4 connector is directly connected to compute module and UART's transmission is provided. The similar action is taken for the receiver pin. The most proper pin of the compute module for the PX4's transmitter pin is selected as GPIO15. Therefore, GPIO15 pin is attached with a label called RX_MODULE. By doing that, the connection between PX4's transmitter and compute module is provided. It is important to check the other pins of the PX4 connector. For this project, the chosen connector is SM06B-GHS-TB(LF)(SN) and it has six pins. The pins are Vin, TX, RX, Pin4, Pin5 and GND respectively. The second TX pin and third RX pin connections are completed with this way. To check the Vin, both compute module's and SM06B-GHS-TB(LF)(SN) connector's official datasheet must be focused on. As a result it, 5V is the proper voltage for the input voltage pin of the connector. The compute module's related UART pins are shown below.

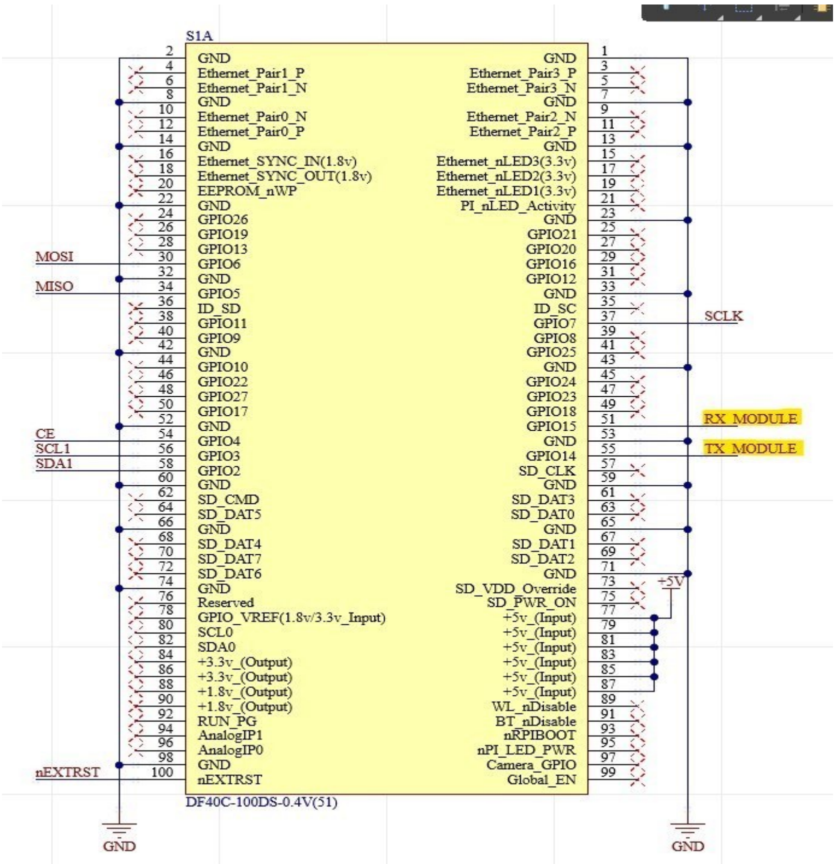

Figure 14: Raspberry Pi Compute Module Connector Pins for the UART Protocol

All the specific information related to connector was also filled in the properties section of the component. Therefore, for a designer and the manufacturer, it will be easier to find the component inside of hundred of tools.

The second selected communication protocol for the printed-circuit board is SPI. This communication protocol is desired to be used to provide the communication between raspberry pi 4 compute module and Wi-Fi module. The Wi-Fi module in the project is called ESP8266. Since the module has two connector parts to connect the board, on the PCB, there needed to be designed two 18-pins connectors with a specific distance between them. For all the information, the ESP8266 module's official datasheet must be checked carefully.

Here is the schematic library of the Wi-Fi module connectors that will be used for the SPI protocol.

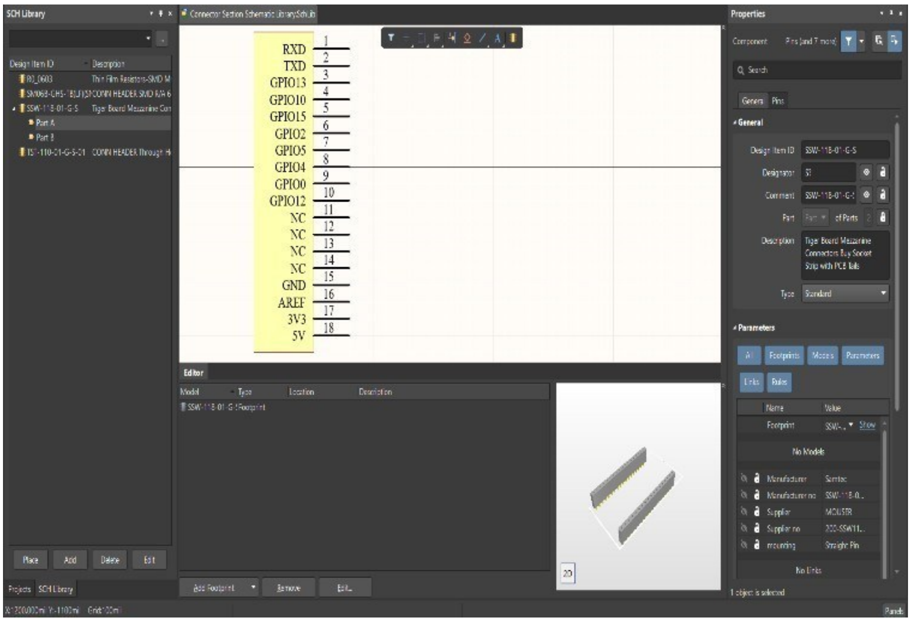

Figure 15: Wi-Fi Module's First Connector Design in Schematic Library

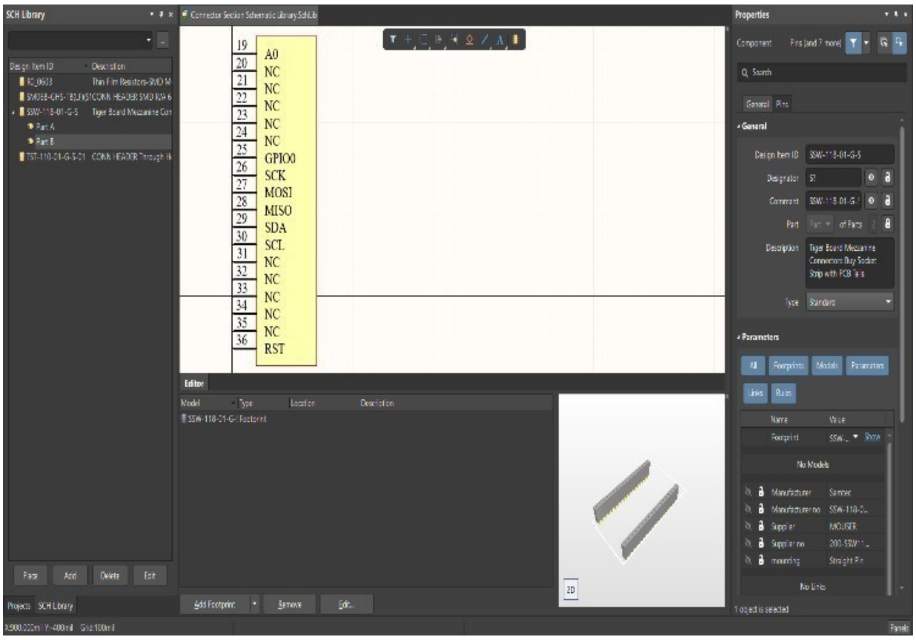

Figure 16: Wi-Fi Module's Second Connector Design in Schematic Library

As shown in the figures above, SPI communication protocol's main pins are SCK, MOSI, MISO, and CE. However as done earlier for the UART protocol, for the SPI protocol between the ESP8266 Wi-Fi module and the raspberry pi compute module, compute module's pins are supposed to be read carefully because the connection will be happening according to the pin functionality. As a last decision of completed research, for the raspberry pi module used for this thesis, the pins that will support SPI protocol are pin30-GPIO6, pin34-GPIO5, pin54-GPIO4 and pin37-GPIO7. To be able to make a connection between compute module and Wi-Fi module by using these pins, the net labelling method used. Therefore, the related pin's label in the Wi-Fi module connector must be the same with the net label of the compute module pin. In the below, the related SPI protocol pins are highlighted on the compute module connector.

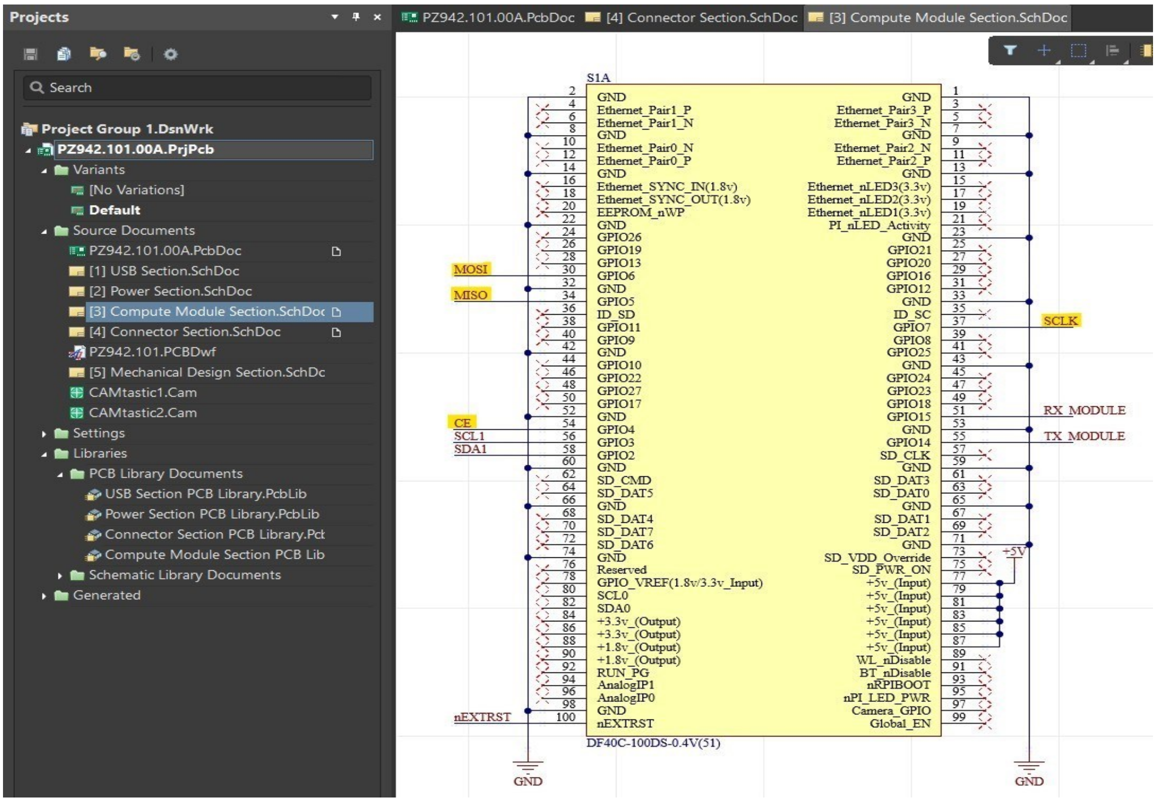

Figure 17: Raspberry Pi Compute Module Connector Pins for the SPI Protocol

The last communication protocol used during the project is I2C. This protocol was aimed to be used for the communication between the LED display and the raspberry pi compute module. The selected LED display module is Serial Liquid Crystal Display Module with NHD-0216K3Z-FL-GBW-V3 part number. To be able to mount this module above the PCB, the official datasheet of the board is examined as always done. The module requires 9 pins with different names and functionalities. In addition to understanding the function of each pin, designer must understand how to provide a connection according to these pins. Therefore, the compute module's I2C connection pins are searched detailly and the proper pins are chosen. The important step is that all the chosen alternative pins of the compute module should be different than each other so that each communication protocol can provide its communication most efficiently. Especially, after figuring out the communication pins for the UART and the SPI on the compute module, the remained pins of the module needs to be searched more since the number of alternative pins are decreased and for the I2C, different alternative compute module pins must be used.

As discussed in the communication protocol section of the thesis, the I2C protocol requires mainly SCL/SCK and SDI/SDA pins to provide a communication. Therefore, these pins on the LED display connector are used to communicate with the compute module. In addition to this, here again, the net label method is used rather than trying to use direct wiring which is not quite possible since the connector section and compute module section are two different PCB sections. To bring two separated sections together, the related pins must use net labelling. Here is presented the main pins for the I2C protocol on LED display connector, on compute module and the net labelling of the pins respectively.

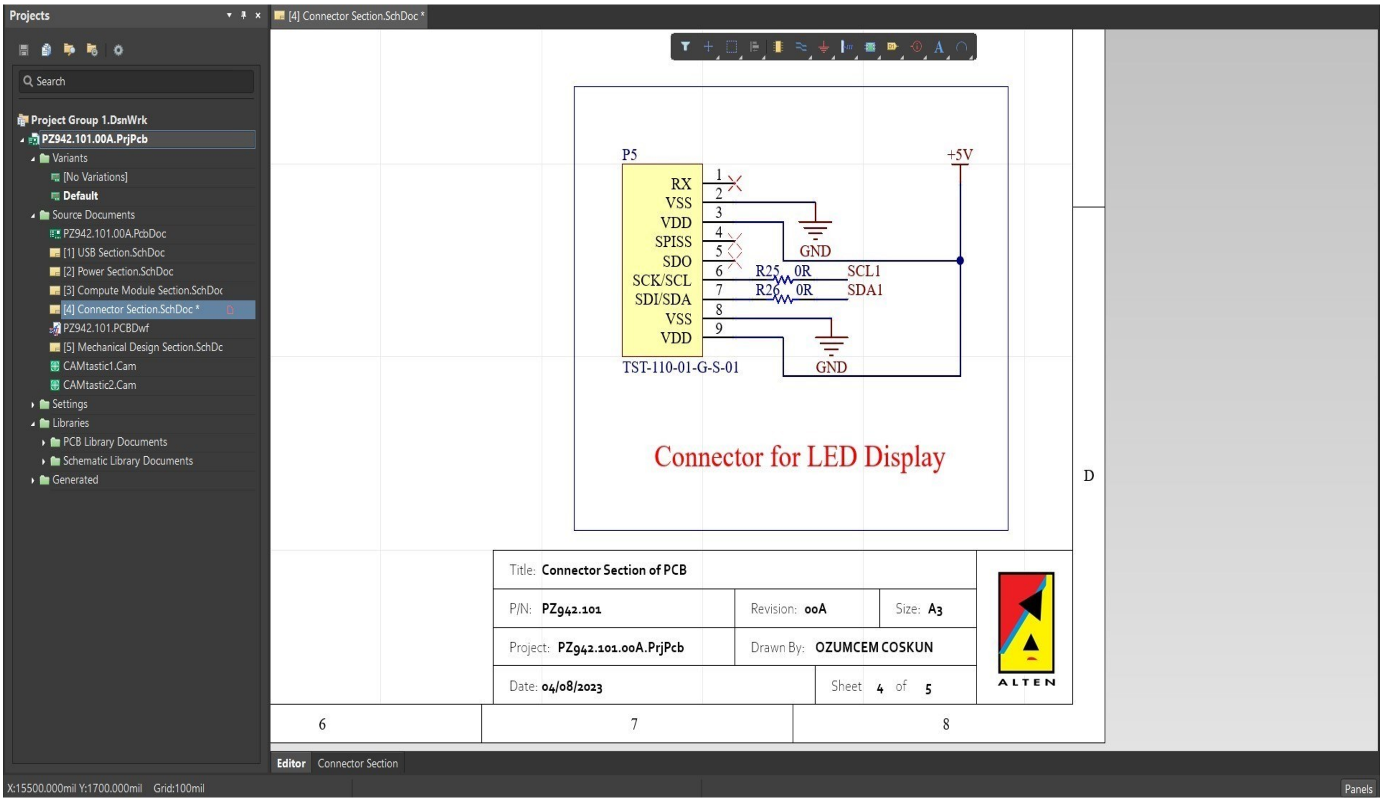

Figure 18: I2C Communication Protocol Pins on the LED Display Connector

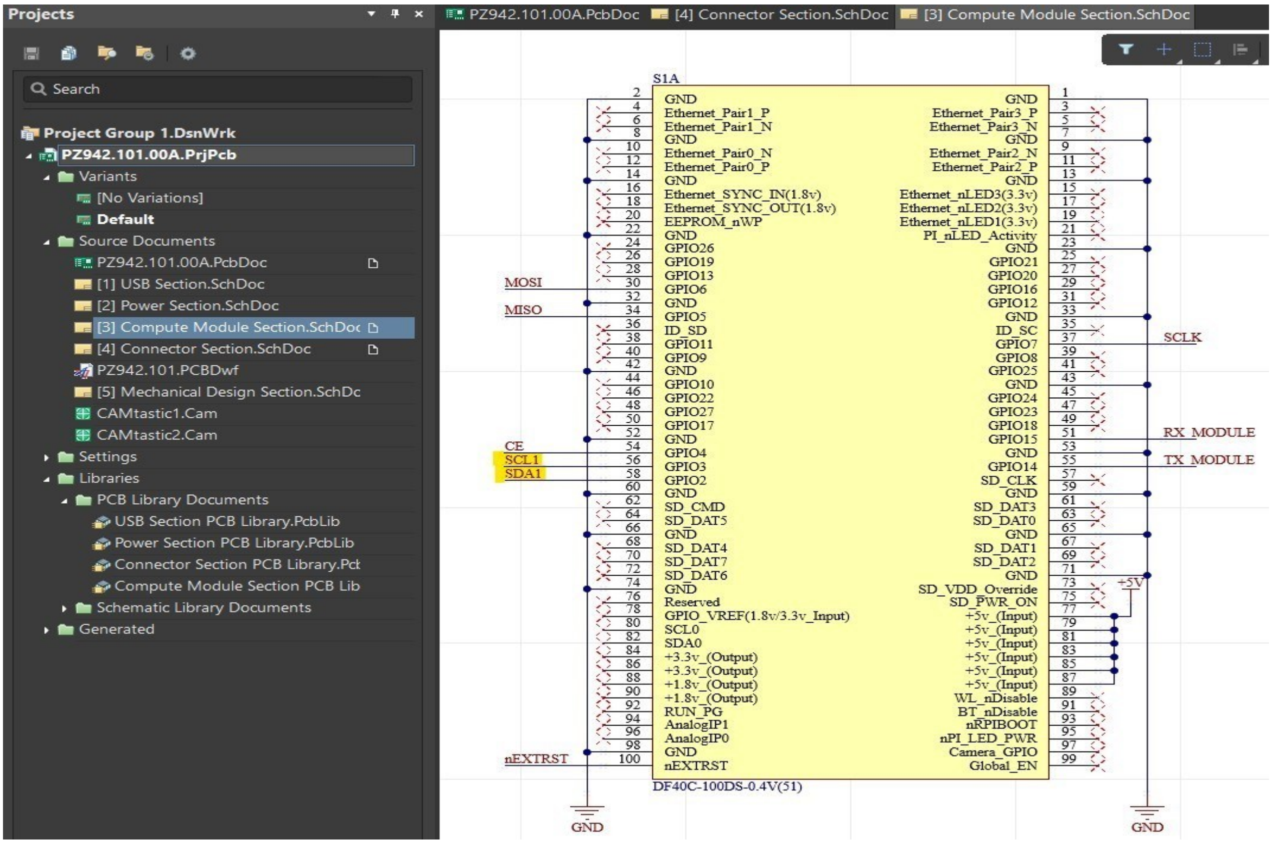

Figure 19: I2C Communication Protocol Pins on the Raspberry Pi Compute Module Connector

4.5. Footprint Creation for the PCB Design

The footprint is one of the most important sections of the entire PCB design process. After creating the components in schematic library and connecting the connection in schematic, the next step is setting up all the footprints for each of the created components. By saying footprint, the pad number, pad size, the distance between pads, the shape, the type of the pads are described. This information is the key element of PCB design just before passing to the last section of PCB design. Every footprint is special to its own component. Therefore, to be able to create a footprint, designer must find the official datasheet of the component by searching from the manufacturer number and check the recommended pad layout section. In this section, information is given with inch or mm units. According to the design unit, any of these can be preferred. However, most of the times, not all the size lengths are given. In such cases, by considering the middle point of each component layout, the calculations must be done. To do the double check, in the PCB library section of the Altium Designer's software, the center coordinates can be set as Pin1, and designer can see if the size between the pads matches with the value calculated manually. At the end of each footprint creation, the center coordinates needs to be adjusted as center, (0,0). This process should be repeated until every component has its own unique footprint. In addition to these, there is another step need to be paid attention. While creating the footprints, the designer has to check the package number of each component. It is not expected to see for integrated circuit and resistor have the same package numbers. However, passive elements such as resistors and capacitors may have the same package numbers. In such cases, rather than creating a footprint for resistor and another footprint for the capacitor which have the same package numbers, one footprint can be used for different components as long as they are in the same package size family. For instance, for this thesis, there are two different package sizes are used which are 0603 and 0805 package sizes. The main reason is doing that is to reduce the number of footprints since same package sized components can share the same footprint.

When the footprint creation is completed, the next step is adding the 3D body of each component. The main logic is to understand how all these components are going to be seen on the manufactured board. In addition to this, it is also a guaranteed way to see if the drawn footprints are correct or not. When the 3D body is added to component, the pad size, the distance between pads in the 3D body have to match with the ones that are created by the designer. Since the 3D body of the component has the certain size, the wrong calculation can be detected and corrected. Also, during the project, all 3D bodies of the components are added a layer called mechanical layer. Since each layer has its own duty, mechanical layers are for the 3D bodies, for the board shape or for the arcs that indicates the first pin of the footprints which make the manufacturer understand how to mount the component. The other key point always to consider is that when the footprints are prepared for each component in the PCB library section of the software, in the schematic library section, all these footprints have to be uploaded to components because in the last section of the project which is called PCB, only the footprints will be visible on the board and only footprint will be placed. The board will be manufactured according to component's footprints. In addition to this, every footprint has its own frame. These frames are collected in both the mechanical layer and top-overlayer. When the single-mode is activated, the mechanical layer is seen as a purple layer whereas the top-overlayer is a yellow layer. These frames are added for a reason around the component's footprint. When the designer places all the components on the board, the area that components occupy will be visible because of the top-overlayer yellow colour and there becomes no conflict between the components. Whenever any changes are done in the PCB library on the footprints, the entire project must be validated. Without validation, the other sub-parts of the projects such as schematic library and schematic are not able to sense the change. Therefore, when the components are placed on the board, these may be the wrong old version of footprints and sizes.

As done for the footprints, the added 3D bodies for each component should also be added into the schematic library part of the project and the entire change should be validated to get the components newest version on the PCB section. Although adding 3D body is quite beneficial for the PCB design, actually it is not a must. However, with the help of 3D bodies, the components placements, their areas and the last version of the board in 3D will be quite clear and more certain. Here are the footprint examples of the components belong to each section respectively.

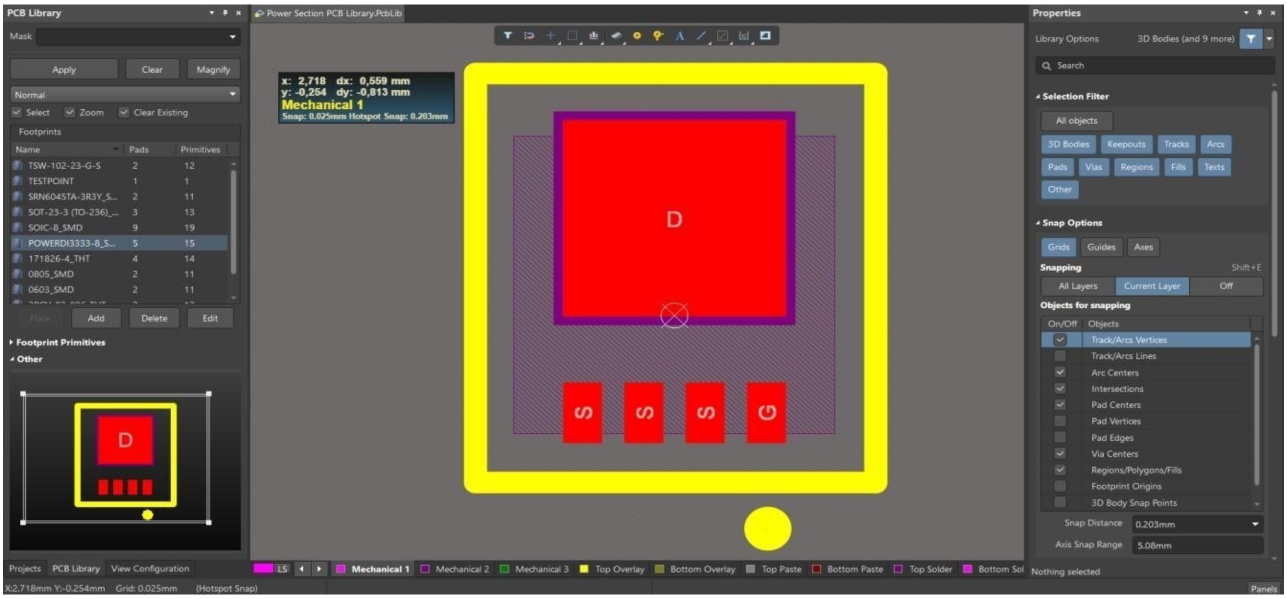

Figure 20: MOSFET Footprint Design in PSU

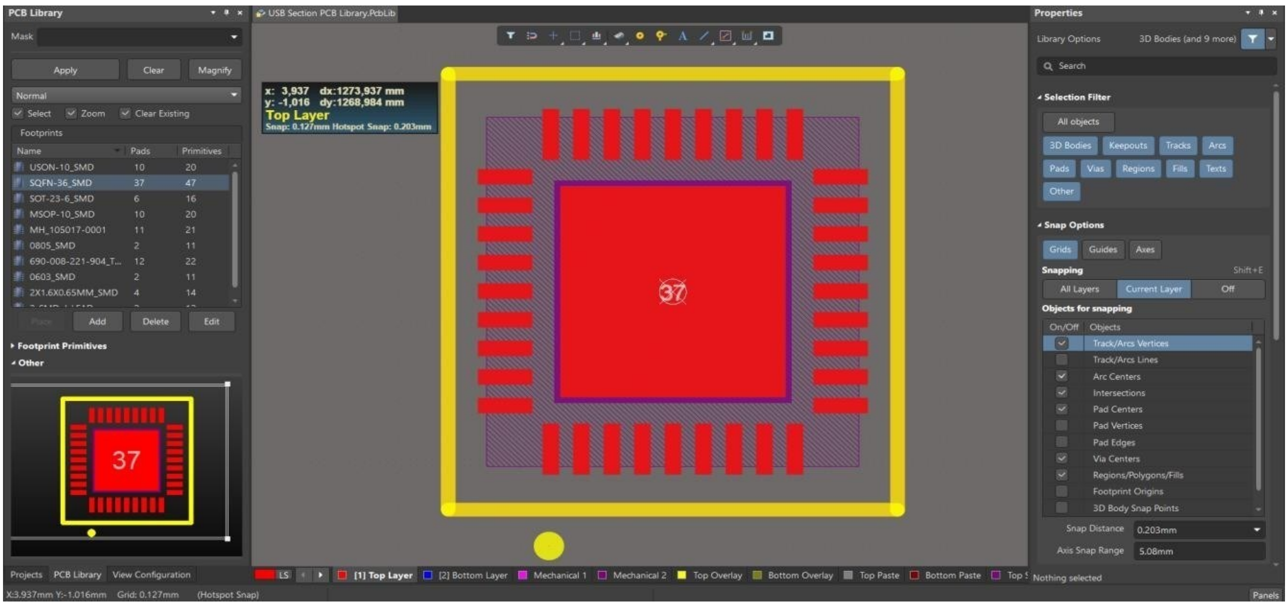

Figure 21: USB Footprint Design

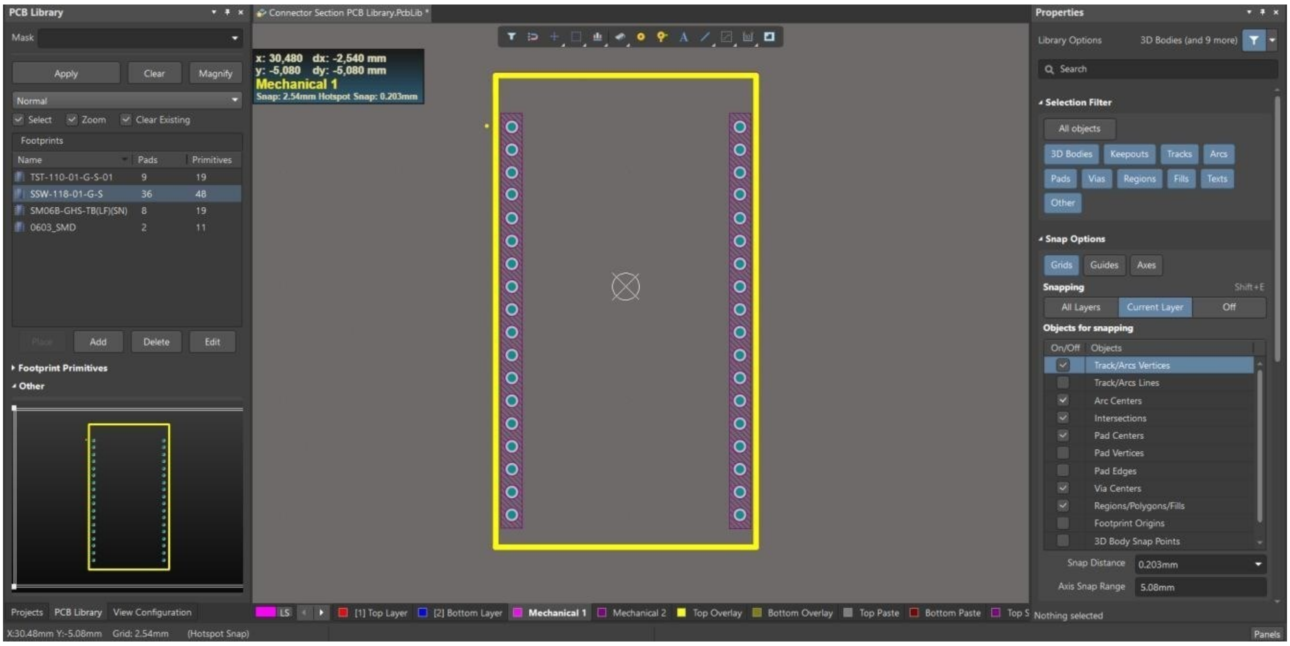

Figure 22: ESP8266 Connector Footprint Design for SPI Protocol

The ESP8266 Wi-Fi module connector footprint is a good example to realize the pad difference. If recognized, while the other footprints have red rectangular pads, this one has circular pads. In PCB design, the circular pads have inner and outer radiuses, and this also means these pads are multi-layer pads. By saying multi-layer pads, these pads' mounting on the board will not be as usual SMD/SMT mounting. Multi-layer pads are through-hole pads, and board has to have the same exact sized hole as in the connector.

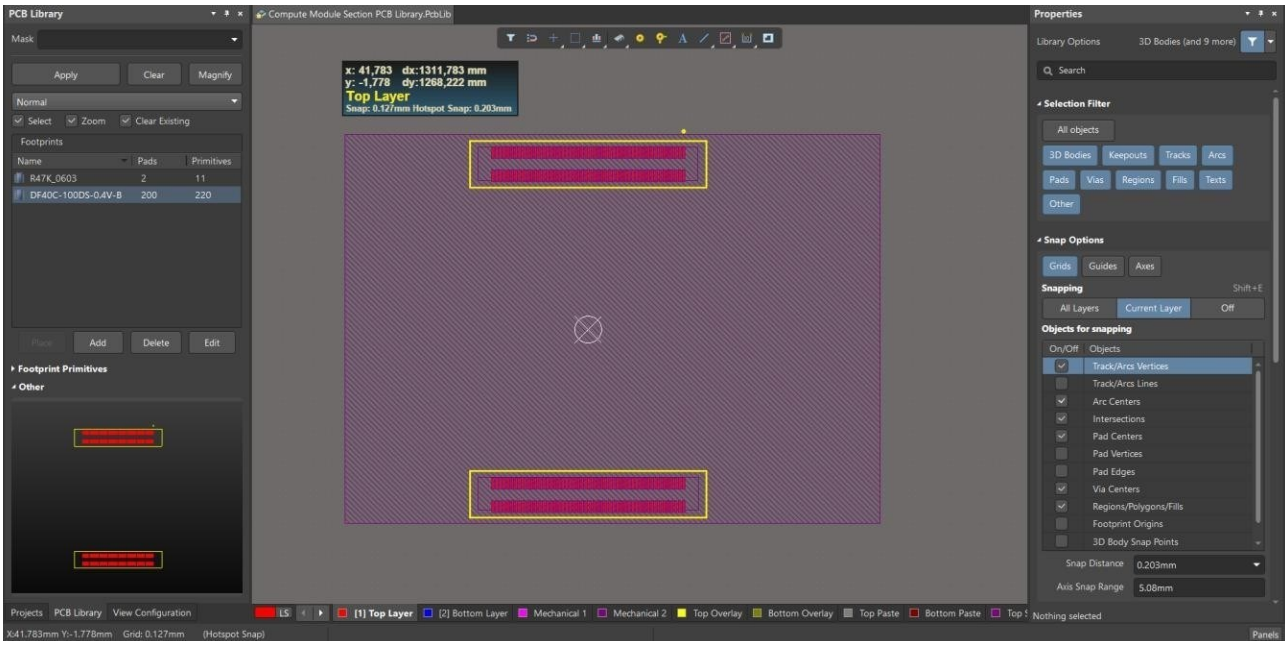

Figure 23: Raspberry Pi 4B Compute Module Connectors Footprint Design

As a summary, as seen in the all figures, the red rectangles (or red circles according to the footprint type) are the pads in the top-layer, the purple surface is a 3D body of the components in the mechanical layer, the yellow frames around the components are top-overlayer, the yellow point on the corners represent the pin1 of the components, the circle in the middle indicates the center coordinates.

4.6. Printed-Circuit Board Section

4.6.1. Design Procedure of the PCB

After creating the components and filling every information of the products in the schematic library section, completing all the connection between the PSU, USB, connector and compute module sections by providing three different communication protocols, UART, SPI and I2C, and creating all the footprints for every single selected components, the last stage of the PCB design control board project is placing all these designed components on the board and providing the connections by using the method called routing. Since in this section, the actual printed-circuit board is designed, the section is also called PCB. There are many key points in PCB section to apply during the design. They are ordered as below step by step.

- The board is supposed to be shaped according to the desired measurements and the placement of the components must be completed according to all recommendations that official component datasheets present.
- The routing must be started from the differential signals since these signals are quite sensitive. The routing is supposed to be done very carefully because the corner of the routing should not be sharp not to cause any damage.
- The components should not conflict with each other. Therefore, the yellow top-overlayer frames should be followed carefully on the board.
- The top layer should be the main layer and the signals like differential signals should be transported in this layer. Every layer needs to be adjusted for specific duties.
- Vias must be added when needed for the nets, voltage sources or grounds.
- When the placement is completed, the polygons are supposed to be created and placed in the suitable layers.
- The keep-out layer should be indicated so that the components are not allowed to move outside

of the board during the design.

- Before starting the routing, the layer stack manager adjustment must be done according to the current and voltage values of the components. With the help of layer stack manager, each via and track width will be specific for the component's current value so that the components does not get destroyed.
- The clearance of the tracks and vias should be adjusted so that there becomes no conflict between the tracks, vias and components.
- The design rule check should be applied so that the designer can see the errors on the board.
- When there is no error in the design rule check, the Gerber file, NC Drill, Assembly file should be printed out so that the manufacturer can use the information.
- When every step is completed and approved by the project manager, specific project number will be given to design by the company.

After talking about the key points, before passing to the PCB section, it is important to know how to do routing. Routing can be counted as the most important action of the entire PCB design since all the connections are completed there, according to the components placements and their electrical properties. Since routing is the heart of this section, there are some routing rules that every designer should follow during the printed-circuit board design. In this thesis, each of these rules are followed respectively. Here is the list of the routing rules listed below:

- Designer should keep the nets as short as possible. If the designer keeps the track length long, the track's inductance, capacitance and the resistance will be greater which means the undesirable factors will be obtained.
- All of the tracks are supposed to have only forty-five degrees at each corner. Designer should avoid using the right angles and no matter what kind of design the designer has, using ninety degrees should be always avoided since the track will be the sharpest version of itself. PCB package has a mode only to bear forty-five degree moves. In addition to this, when the corner of the tracks are ninety degrees, the tracks can not produce EMI that are measurable. Also, as a design, the sharp tracks never seem good in the PCB.
- Designer should not use rounded track corners since these are much slower and harder for the placing.
- The tracking should be completed with a sneak method which means tracking from point to point should be avoided. Although point to point tracking method seems more efficient than

sneak tracking method in the beginning, there are two main reasons why to avoid it. The first reason is about the esthetic. PCB design should have an esthetic look when the board is completed so that the user can have a chance to investigate the connections. Point to point method seems not esthetical in the design. The second reason is that when the designer wants to combine more layers in the design just like how it is done in this thesis project, point to point method is not space efficient.

- Designer should enable the design's electrical grid. The software should be able to find the pad center and end the tracks automatically for the designer. When the pads and tracks are not lined up to the given snap grid, enabling the electrical grid will be a great opportunity for the designer.
- When the routing is started, designer should always get the tracks to the pad center. Only touching the tracks to the pads should be avoided during the routing. If this rule is not applied, the program does not sense that the track is having an electrical connection with the pad. Therefore, having a proper snap and the electrical grid get rid of the many problems that designer can have in this step.
- During the routing, rather than using the multiple tracks, a single track should be used. Although as a look, it does not show a certain difference, for the upcoming edits in the PCB, it can turn to a serious problem. According to the connection, the tracks size can be wider. In such cases, the best option is to delete the previous one and using the current track size. Although adjusting it manually may take more time for the designer, during the design, it will be worth it.
- While routing, the designer must be sure that tracks are moving from directly center to center of the pads. If it goes only to the one side, although it seems like there is a connection, connection would not be completed in such case.
- Unless the design is very large, the designer should avoid using the two tracks. Only one track between the pads should be the main focus. The main for that is considering the tolerances of the components and the connections.
- As done many times during the project, for the high currents, the designer should use as many vias as possible while passing through the layers. With the help of it, the track impedance will be decreased, and the reliability will be increased. Whenever the power planes' or tracks' impedance should be lowered, this technique is supposed to be used.
- If the designer realizes that ground and the power tracks seem critical, these steps should be lay down firstly. After that, the power tracks should become as big as possible. It can be realized in the routing of this project, the power tracks are much bigger than the signal tracks

because of the current.

- Designer should hold ground and power tracks very close. Rather than sending these two types of tracks in the opposite ways, the directions should be the same. With the help of it, the power system's loop inductance will be reduced, and more efficient bypassing will be obtained.
- From the general perspective, the placement, the routing would be better in the symmetrical positions. It is quite important for the professional point of view.
- Designer should never leave the copper fills unconnected. They should be grounded or gotten rid of as done during the thesis project.
- If the designer's board is non-plated double-sided board, the link should be soldered on the both top layer and the bottom layer through the boards.
- If the designer's board is non-plated double-sided board, vias should not be added below the components.
- If the designer's board is non-plated double-sided board, the designer should try to use the components with through hole rather than SMD. With the help of it, the number of vias used will be reduced. Also, since whenever the via is added to the board, two solder joints will be added too. Therefore, the reliability will be decreased.

When the routing is completed, there are some other last touches should be done. Here is the list of the actions that designers take right after the routing:

- When the design has thin tracks, the chamfer needs to be added on the T junctions to be able to get rid of the ninety degree angle. With the help of it, the track becomes more robust and the possible etching problems during the manufacturing will be removed completely. From the esthetical perspective, the board look more detailed and nicer.
- The designer needs to check the board structure if there is any hole required on the board. If yes, designer should do the mounting holes clearly.
- The hole sizes need to be checked by the designer. When the hole size is bigger, the extra cost can be obtained. When the hole sizes difference in the board, it will cause again the extra cost since the drilling will be varied.
- Designer makes sure that there is always a certain distance between the placed components. Otherwise, there becomes conflict and the disconnection.

After all the important steps are listed, here is the identified board shape and the completed component placement of the project given below.

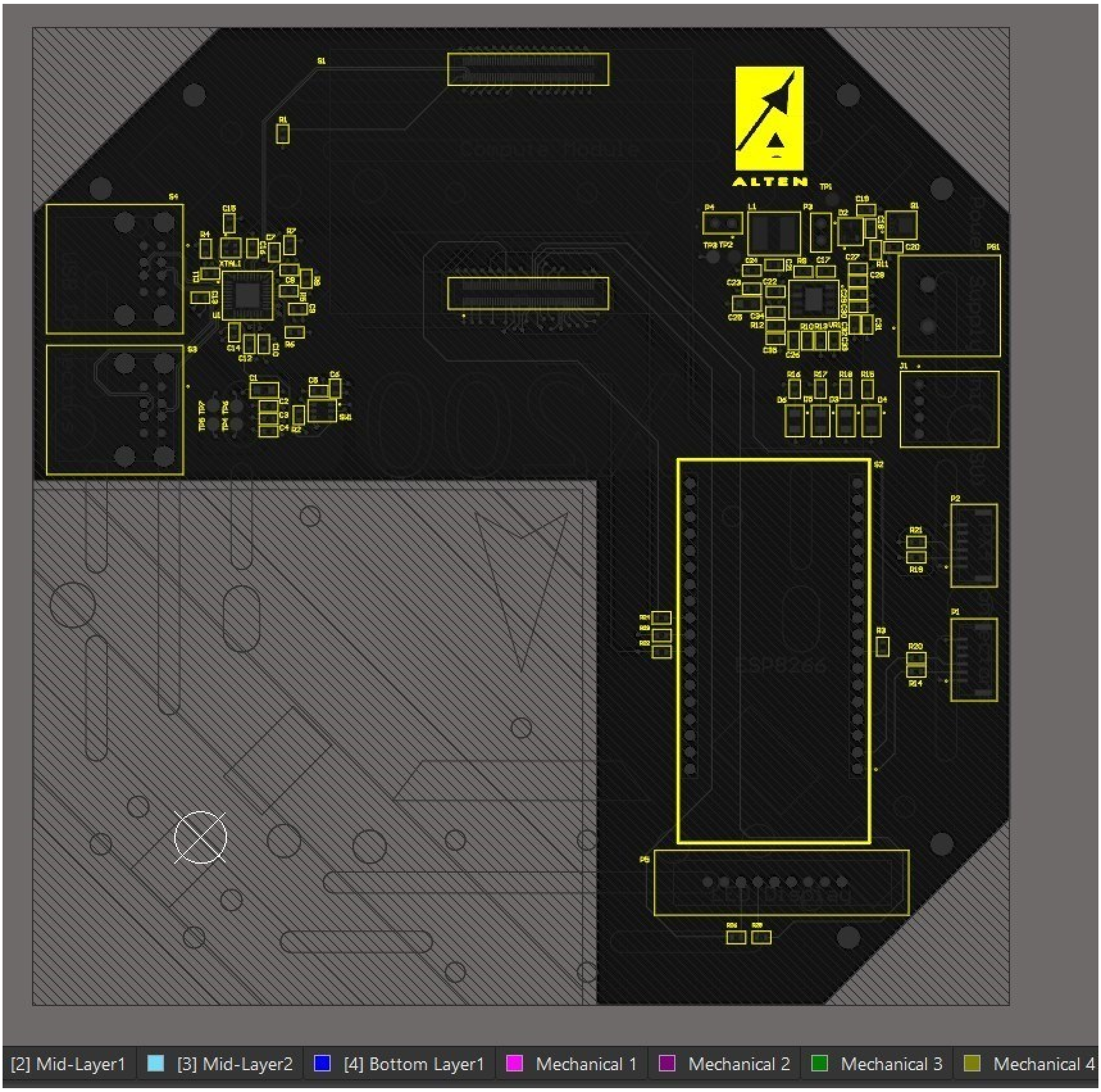

Figure 24: PCB's Top-Overlayer Demonstration

The USB section is placed on the left side. Since the stacked USB connectors will be used externally, two USB connectors are placed as close as possible on the edge. Since power supply unit (PSU) and the USB section does not have a direct connection, PSU is placed on the right side of the board across the USB section. In the middle, raspberry pi compute module is placed because both the USB and PSU section is directly connected to the compute module. On the right side, just below the PSU, Wi-Fi module connectors and PX4 autopilot connectors are placed. Since the PX4 connector connection will be provided with an external device, it is more logical to put these connectors as close as possible to the edge. Below the UART and SPI protocol connectors, I2C protocol connector is placed which is also the LED display connector. While doing the placements, the board mounting points should be considered as well. These are the circles that will be used for the mounting of designed PCB onto the surface. Therefore, if there is any conflict of these points with a component, during the mounting, the components may be destroyed.

During the placement process, it is realized that voltage regulator in the PSU section is more different than the others. While other components do not require a specific limitation, voltage regulator presents some certain limits for the PCB Layout. Here are the steps that indicate how the voltage regulator and the other components are placed during the placement.

- The voltage regulator (AP64501) works with 5 amper load current and the heat dissipation can be a serious problem for the layout. Therefore, for both the top and the bottom layers, the 2oz copper is suggested.
- The input capacitors are supposed to be as close as possible to the Vin and GND pins of the voltage regulator.
- The inductor connected to the voltage regulator needs to be connected to SW pin of the regulator.
- The output capacitors are supposed to be placed to GND pin of the voltage regulator as close as possible.
- Entire feedback components should be placed closely to FB pin of the regulator.
- Since the designed PCB has four layers (top layer, mid-layer1, mid-layer2, bottom-layer), the second or the third layer should be used as ground to be able to increase the thermal performance.
- The ground layer should have as many vias as possible for the heat dissipation as it is done for the Vin plane.

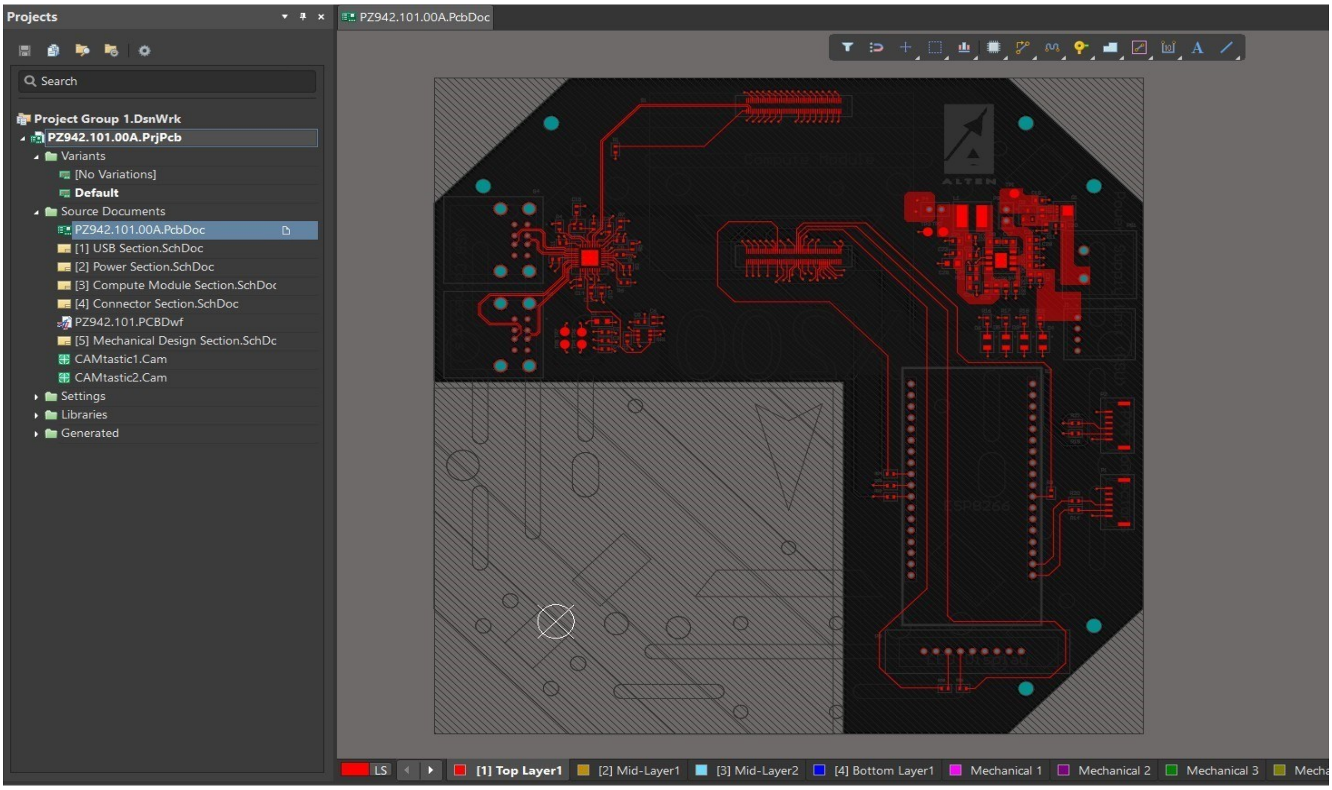

Figure 25: Top-Layer Demonstration of the PCB

After completing the placements of each section and the components, the routing of the nets is started. The first routing is done for the connection of differential signals between the compute module and the USB module and also between the stacked USB connectors and USB module. However, with this way, there becomes many errors since the track and via sizes are not adjusted. The adjustment is set according to the rated current that every component's datasheet. To be able to do the correct routing, information table to set the sizes for the routing is created and listed below.

Component Description	Component Part Number	Voltage(V)	Current(A)
Voltage Source		+3.3V	1.5A
Voltage Source		+5V	3.5A
Voltage Source		VBUS	2.1A
Voltage Source		+12V	2A
Power Inductor	SRN6045TA-3R3Y	5A (Maximum DC Current)	
Barrier-terminal block	3PCV-02-006	300V (Rated Voltage)	10A (Rated Curr.)
Power Switching Integrated-Circuit	AP22653W6-7	3 V to 5.5 V	2.1A Output 2A Input Current

BVDSS-MOSFET: 25V-30V	DMP3013SFV-7		35A (Continuous Drain Current)
Zener Diode	MMBZ5249	12V	1 µA (Reversed)
DC-DC Conv Buck Switching Regulators	AP64501SP-13	3.8V (Input-Voltage Min) 40V (Input-Voltage Max)	5A (Output Current)
LED-PLCC2 SMD	QBLP676-IW-NW	3V (Forward Volt)	20mA (Test Curr.)
USB, USB 2.0 Hi-Speed Integrated-Circuit	USB2514B-I/M2	3V-Min supply volt 3.6V-Max supply volt)	70mA (Supply Current)
Conn-Header	SM06B-GHS-TB(LF)(SN)		1A (Rated Current)
Tiger-Board Connector	SSW-118-01-G-S	465 VAC, 655 VDC (Rated Voltage)	4.7A (Rated Current)
Conn-Header	TST-110-01-G-S-01	425 VAC/600 VDC	3.4A (Rated Current)

Table 1: Information Table for the Routing Process

By using the current values represented in the table above, the software called Saturn_PCB shows the width of the tracks and vias. Since the tracks and vias are varied according to the power nets, signals nets and ground nates, the table for the values for each section is shown below:

	Power Tracks	Signal Tracks	Ground Tracks
Preferred width:	0.254mm	0.18mm	0.254mm
Minimum width:	0.2mm	0.18mm	0.15mm
Maximum width:	0.254mm	0.25mm	0.254mm

Table 2: Track Size vs Width Table

	Power Via	Signal Via	Ground Via
Minimum hole size:	0.3mm	0.3mm	0.3mm
Maximum hole size:	0.5mm	0.3mm	0.5mm
Preferred hole size:	0.3mm	0.3mm	0.3mm

Table 3: Via Size vs Hole Table

When the via and tracks sizes are decided by using the Saturn_PCB software, routing of the PCB is started. Again, the first routing is done on the differential pairs. Since the PCB can sense the connection of the nodes in the circuits, during the routing, designer can make a connection without searching the correct node for a long time. Direct routing is fine if the components are close to each other. In some complicated designs as this design for the thesis, the track is not able to reach the component because of the number of tracks used and no tracks are allowed to pass the other one in the same layer. In the cases where the tracks may have conflict, vias are used. This is also the purpose of the vias. In the thesis project, the vias are mainly used for the voltage sources and the grounds. Since each component has its ground pad and its not easy to collect all the grounds in the top layer because except the ground and voltages, all the routing is completed in the top layer, for the connection of ground pads, the second layer of the PCB is used which is mid-layer1. For the VBUS and +3.3V voltage connections, the third layer of the PCB is preferred which is mid-layer2. After completing all the signal routings and setting the routing for +12V in the top-layer, for all the ground routings in the mid-layer1, for VBUS and +3.3V routings in the mid-layer2, the only left pad was +5V. Normally, the +5V routing is demanded to add into the mid-layer2 so that all voltages could be collected in the same layer. However, the intensity of VBUS and +3.3V tracks did not allow +5V tracks to be collected. Therefore, +5V routing is completed in the last layer of the printed circuit board which is bottom layer.

For the PCB design, its important to remember that although the routing can be done by using many tracks, this is not the preferred technique. Since one of the thesis requirements was letting the PCB work as much efficient as possible, rather than using lots of tracks to combine pads, another method is preferred which is called polygon. For the designs that has multi-layers, complex and has many circuitry sections just like the current project, polygon method is the best way for the routing for the PCB's efficiency. Here is the created polygon for the project in the top layer:

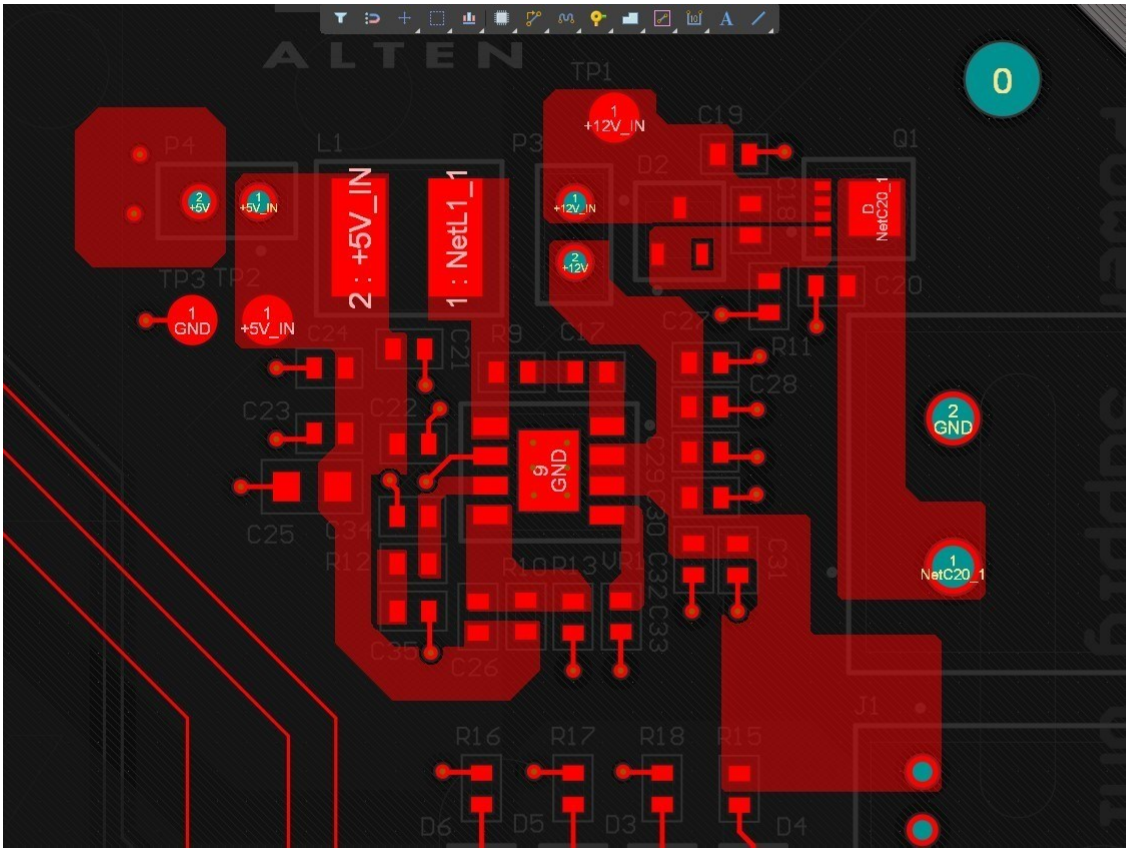

Figure 26: Polygon Structure in the Top-Layer

Polygon is the copper area that is added above the existing objects. These existing objects may be tracks or pads. Polygons are able to be placed to define surrounded shape. Polygon may be hatched or a solid and also can be placed above the other layers. However, they only pour around the objects in the signal layers. The signal layer polygons keep clearances, and the clearance is defined in the section of clearance design rules. For the thesis project, the clearance is adjusted as 0.19mm. However, the clearance for the ground layer and the bottom layer is adjusted differently because if the designer defines only one clearance rule for the all layers, since the track and pad sizes are different according to the signal type, there becomes conflict between the tracks and vias. Therefore, design rule check method gives many errors that indicates the PCB would not be an efficient one which is against the project's purpose. The clearance set for the bottom layer is 0.5mm and for the ground is 0.18-0.19. After setting the clearance rule, all the rules are adjusted so polygon creation can be started. In top layer, polygons are created for the nets which are +12V_IN, +12V, NetC20_1, NetC18_2, NetL1_1, NetC17_2, NetC34_1, NetC26_2, NetC33_1, +5V, 5V_IN, NetC17_1. In the mid-layer1, the polygon is created for GROUND. That means all of the ground vias are collected in the second layer which is mid-layer1. In the third layer of the PCB which is mid-layer2, all the VBUS vias are gathered together and the polygon is created. In addition to VBUS, an another polygon is created in this layer for the +3.3V vias. Therefore, the third layer of the PCB is separated only for the voltage sources. Lastly, the last layer of the PCB which is bottom layer is used for the +5V vias.

According to the given information, although each polygon for the net is created in only one section, for the +5V vias polygons, there are two layers are used separately. One+5V polygon is created in the top-layer, the other +5V polygon is created in the bottom layer. The main reason of such design is that the created component for the +5V has also the highest current amount which is +3.5A, Therefore, rather than collecting the vias only in one layer, the polygon creation is separated into two different layers to make sure that design will not get affected negatively because of the current amount. It is very necessary to complete the pouring whenever there is any change done in the polygon creation. When the pouring is completed, the board will be shaped and have the via size holes on it according to the place of vias. The top layer polygons are represented above. Since the related nets are close to each other, it is easy to collect them in the polygon plane. However, in the mid-layer1 layer, the design is full of ground vias and there is no any proper order. Therefore, the entire board of this layer is turned to a ground polygon. In the third layer, since the layer is aimed to use for the two voltage nets, it was impossible to choose the whole board as one polygon. Therefore, two different polygons are created. During the action, it is important to be careful that the polygons should not be conflicted with each other. Here is the mid-layer1, mid-layer2 and bottom layer polygons of the design are shown:

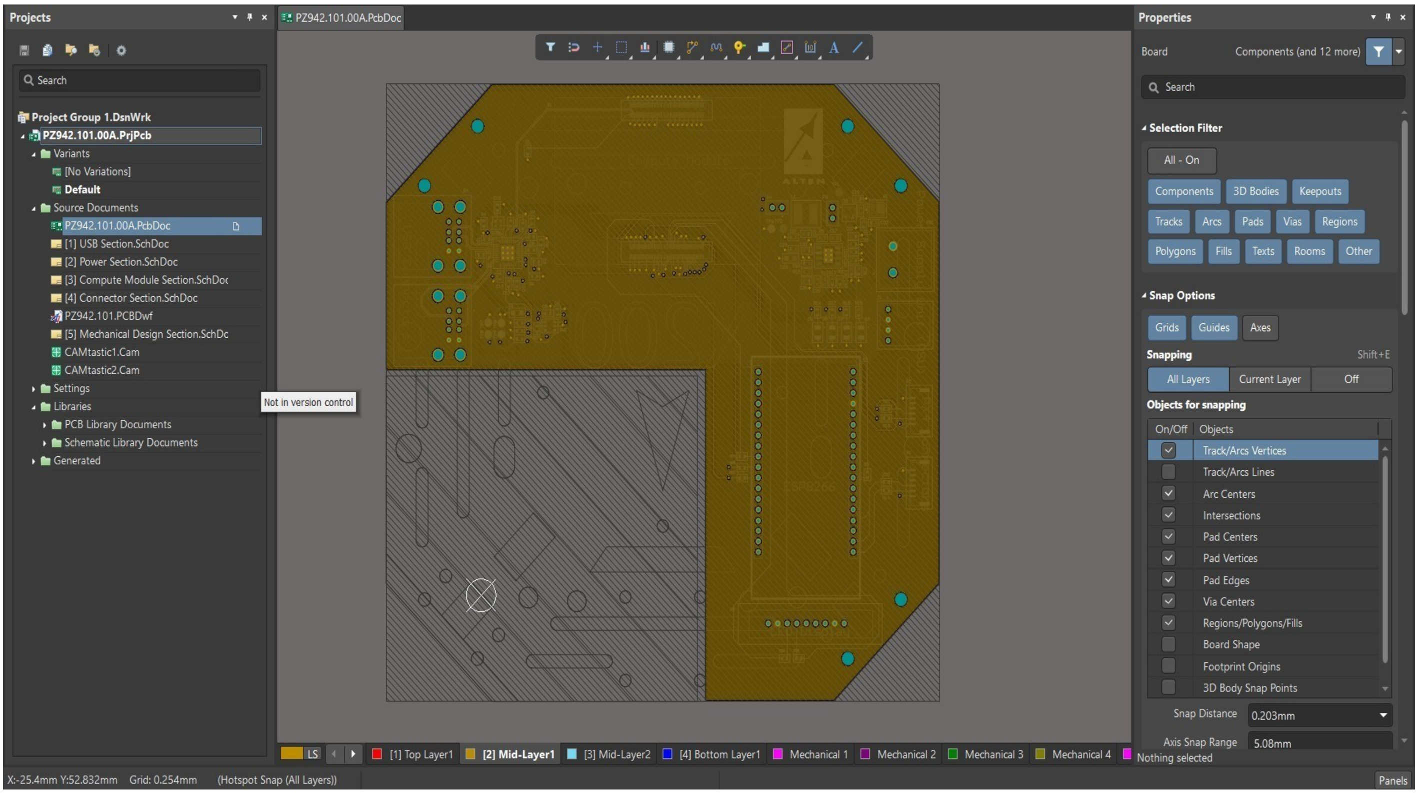

Figure 27: Mid-layer 1 Polygon Structure for the Ground

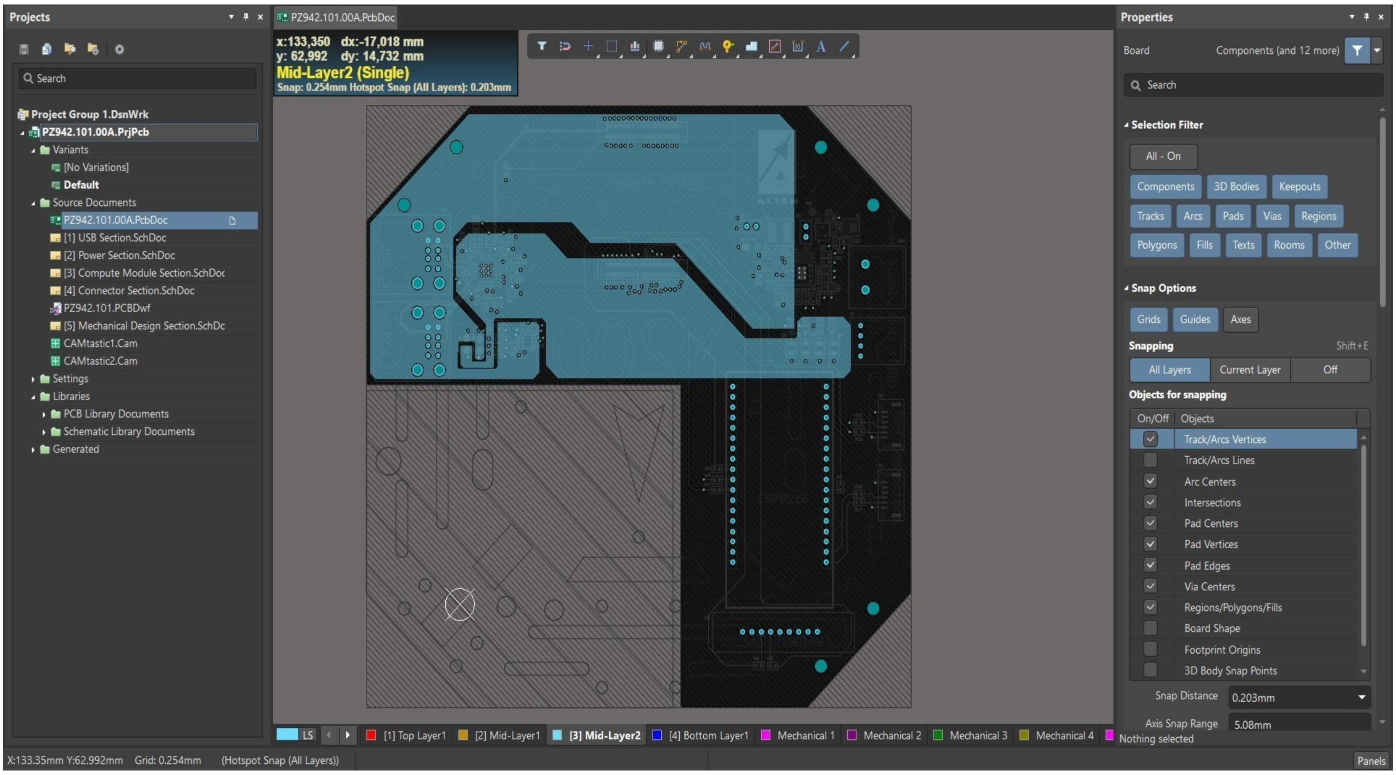

Figure 28: Mid-layer 2 Polygon Structure for the +3.3V and VBUS

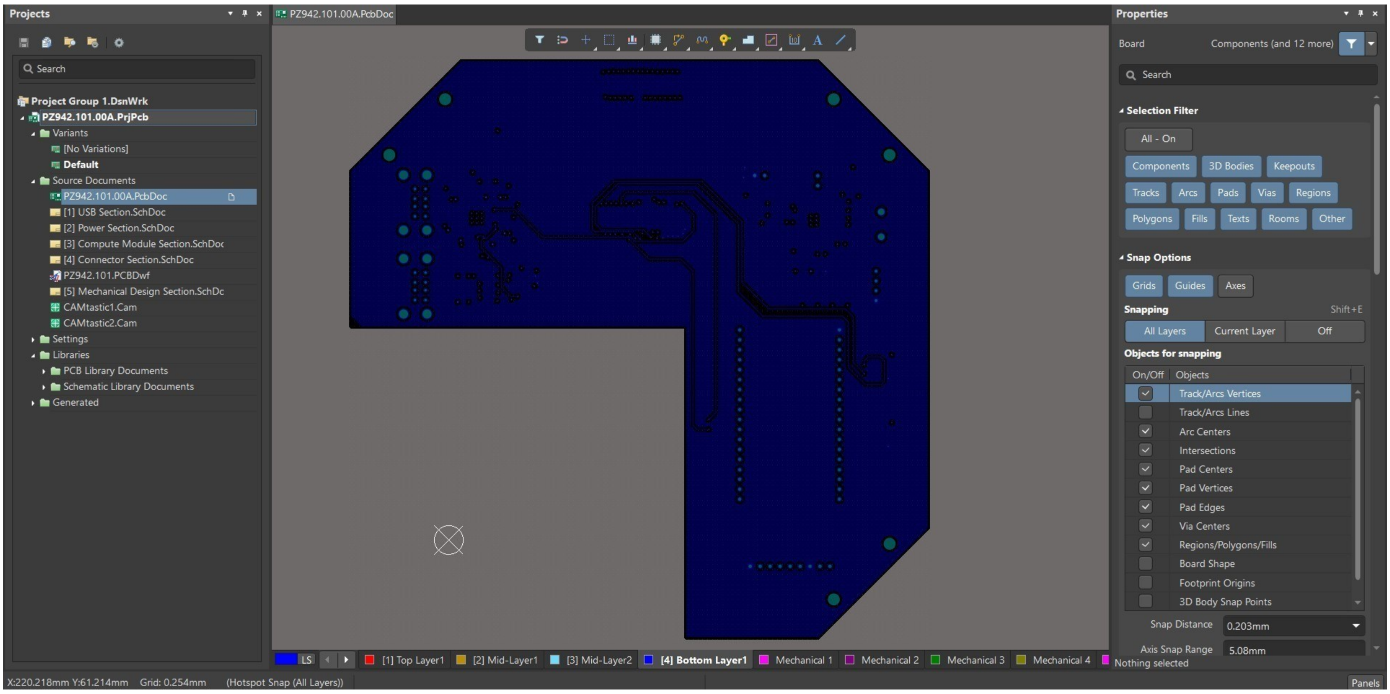

Figure 29: Bottom-Layer Polygon Structure for the +5V

When the placement, routing and polygons creation is completed, not to have any problems in the upcoming sections, design-rule check must be done and see if there is any error to fix. When there becomes no error, the next step in PCB design is obtaining the assembly drawings and output files.

Assembly drawing indicates the location of each component and the any required instructions that need to be completed for the proper assembly. Since the silkscreen designator cleans up, the text for the assembly needs to be cleaned up too. In assembly drawing, each part of the sections is supposed to be identified in the drawing print to support special processes. The detailed profiles and areas should be included too. These detailed profiles and area can be card ejectors, stiffeners for the board and rework of instructions. According to the customer name, the title block of the assembly is supposed to be filled in the assembly drawing. Here is the assembly drawing of the project printed out for this thesis.

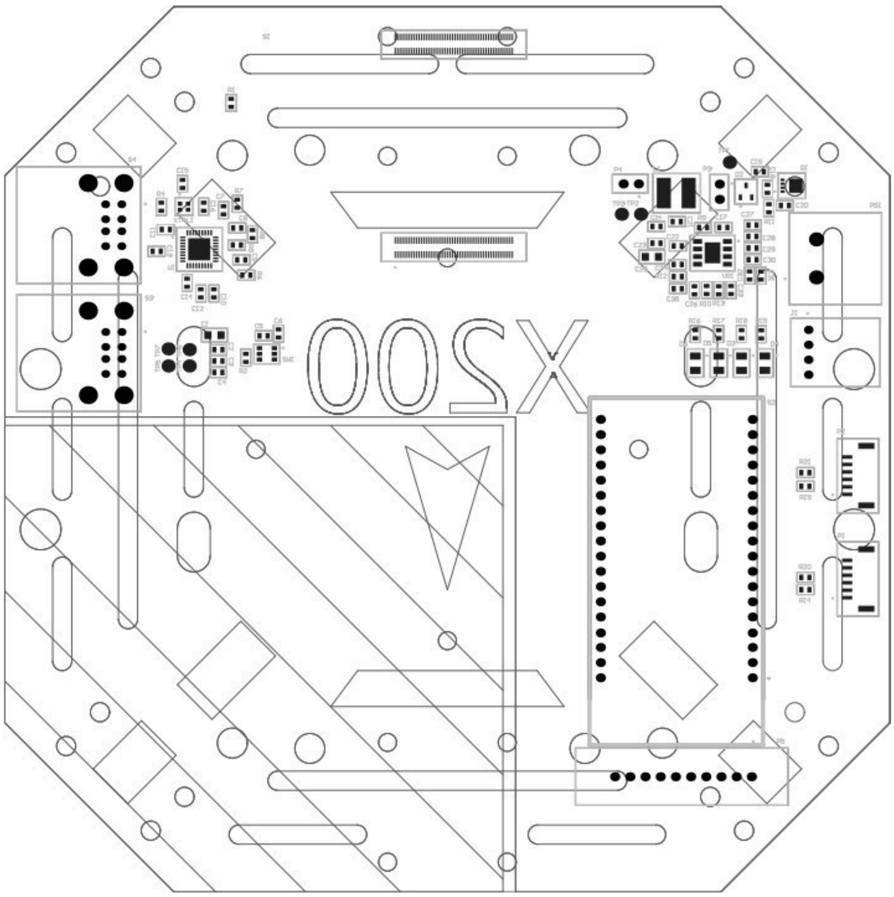

Figure 30: Assembly Drawing of the Designed PCB

There are four types of output files that gives the main information about the design for the manufacturing process as discussed in [15]. The files are:

- Bill of Materials
- Pick and Place
- Gerber Files
- NC Drill

4.6.2. Bill of Materials (BOM)

Bill of material can be defined as simply a list. In the industries as printed-circuit board design and manufacture, bill of material is used to be able to list all of the chosen, needed parts in the manufacturing to create a unique, desired PCB according to the project. There are many types of information in bill of materials files. The most common and preferred elements are comment, description, designator and the footprint respectively.

i. Comment: Every part on the PCB should have the unique part number, or an identifier in the bills and materials section. This information is listed as comments. Although this is not a must, as a comment, company designated part-numbers are listed. In addition to this, rather than using such part-numbers, part numbers for the vendor component or the other designators may be used too. For instance, for this thesis, the given part number by the company is PZ942.101

ii. Description: In this part, the basic description of the designer or the official description of the products are listed to have a common idea about the part.

iii. Designator: All of these used components in the design should have their own reference designators. The designators are generally represented as components initial with the number. For instance, in this project, the 10uF capacitor has a designator as C21. 21 means that the used capacitor is 21st capacitor in the entire design whereas C is the capacitor's global representation.

iv. Footprint: Footprint is the name of a physical CAD which are used in the schematic library part and created in the PCB library section.

In addition to comment, description, designator, and the footprint parts in the bill of materials, depending on the company requirements, other elements are considered too. These elements can be tolerances, product structure, values, and raw material notes. The only negative effect having all these elements in the bill of materials is that the report can be quite long based on the element numbers. Therefore, generally the companies prefer the first four main elements in the bill of material reports.

4.6.3. Pick and Place

When the chosen components are delivered in the bulk, the pick and place system gets rid of the components from their packaging. It places the components to the board by checking their orientation and puts them in the correct order. Most of the components for the pick and place are above the tape reels of the different sizes. That is why during the change of reels, much attention should be paid. Since most of the components are going to have the similar part numbers or the same appearances, errors may be obtained.

4.6.4. Gerber Files

Gerber drawings or gerber files provide quite critical information during the fabrication process of the printed-circuit boards. These files are actually approved industry standard formats to submit the PCB's layout data. Gerber drawings include the requirements of the design by showing the all layers of the PCB which is extremely critical for both the PCB assembly and for the PCB fabrication. During this thesis project, as a format, Gerber X2 format is used. Once the designer creates a layout design for the circuits by using a software, designer send the design for the fabrication. However, the manufacturer does not speak the same language as the designer all the time. Therefore, the PCB designer find a solution to forwards the data in the most standard way. In such case, Gerber drawing format is the solution to be able to prevent these obstacles between the designer and the manufacturer as also discussed in study [15]. The gerber file of the thesis project is represented below:

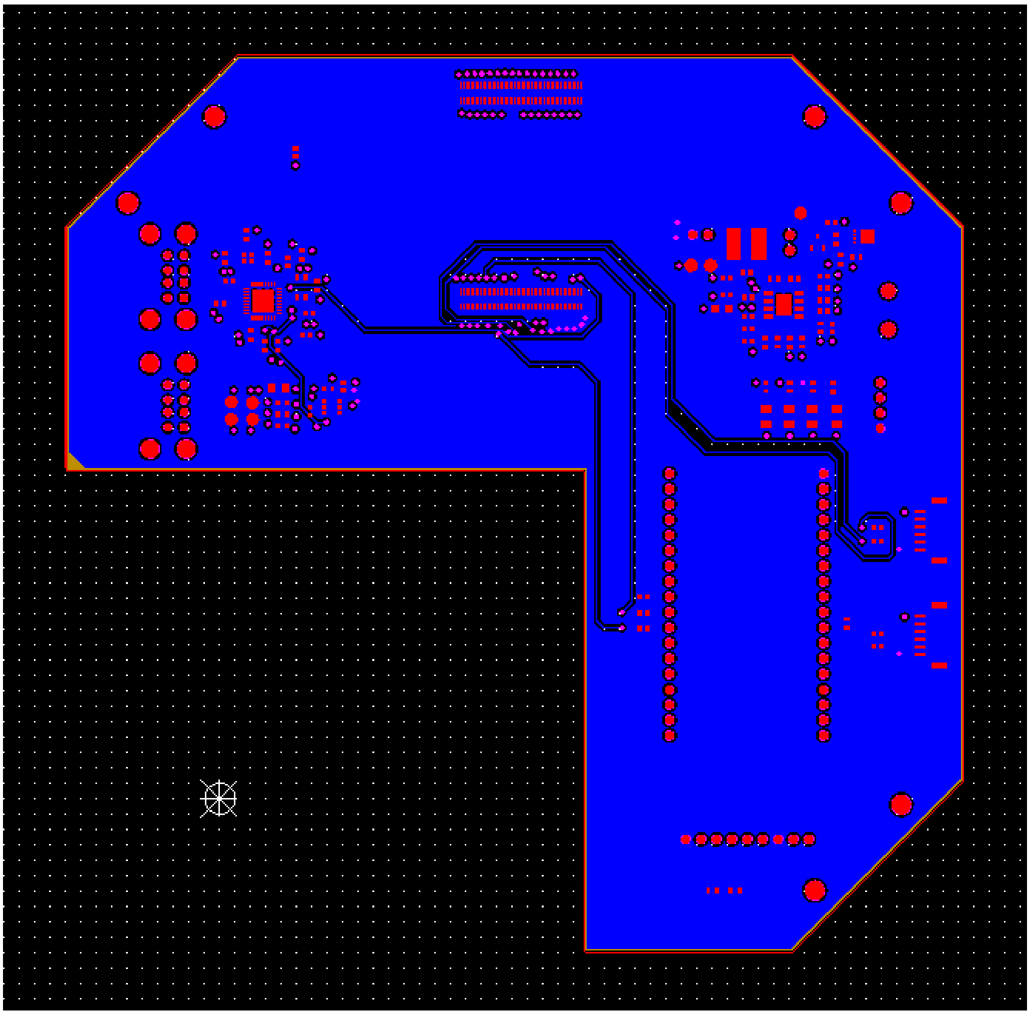

Figure 31: Gerber File of the Designed PCB

4.6.5. NC Drill

In the printed-circuit board, drill holes are put with drill bit sizes which are standard when the stackup becomes fabricated. This operation is done before the PCB is plated. In the drill table, the drill bit sizes and the location of drills are listed. The location of drills are also indicated in the Gerber drawings. The standard drill sizes in bits are presented in metric or inch as units. No matter how tiny the drill hole is, the pad and the drill hole are supposed to be designed simultaneously to obtain reliability, assembly and high yield as discussed in the study [16]. The NC drill of the project done for the thesis is represented below:

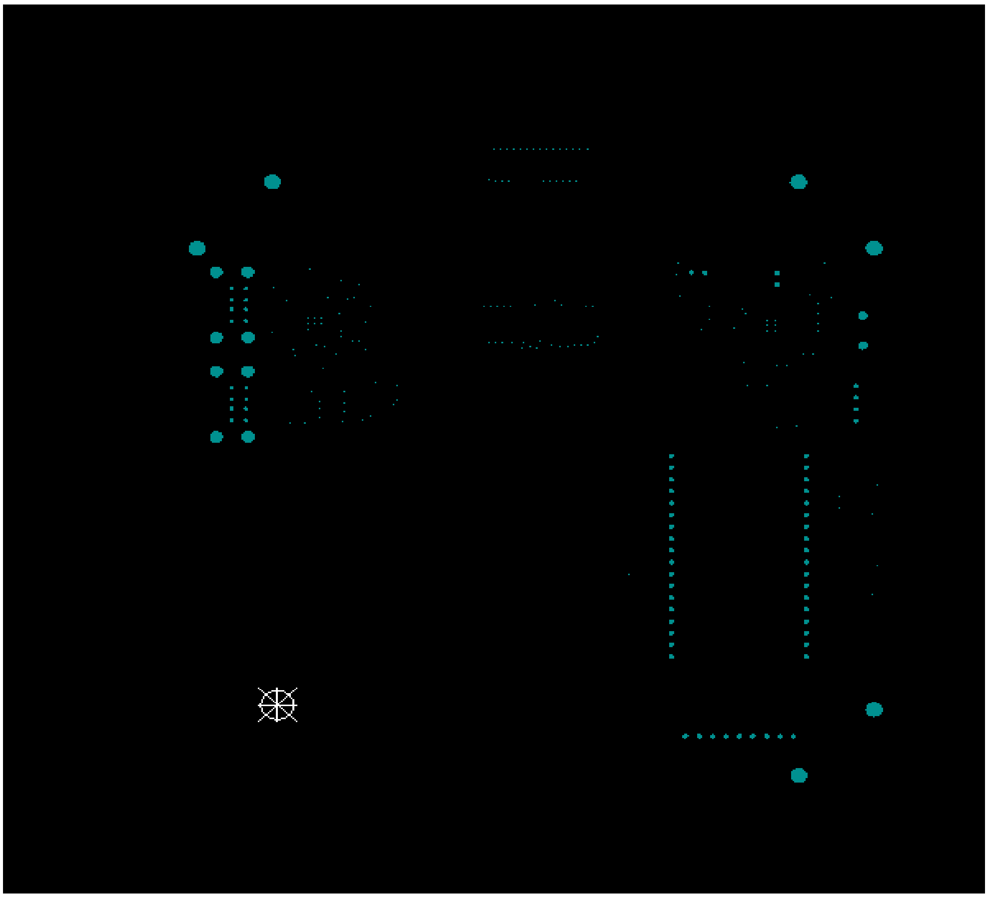

Figure 32: NC Drill File of the Designed PCB

The common point of these four files is that each of them introduces every single detail about the designed printed-circuit board with all the used layers for the board review, board fabrication and to get assembled. Here is the final version of the designed printed-circuit board in 2D, in 3D, and with all types of output files for the manufacturing part of the project.

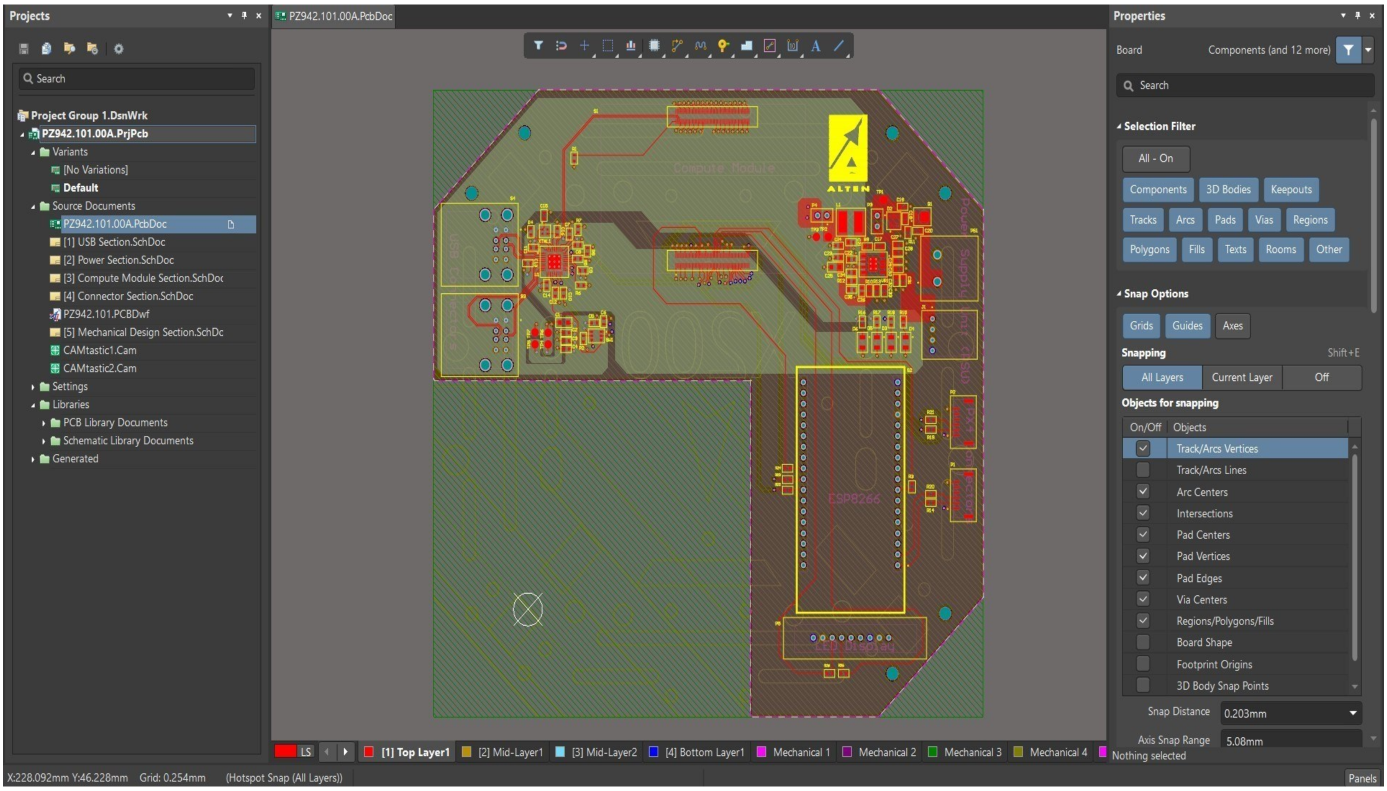

Figure 33: Last Version of the Designed PCB in 2D

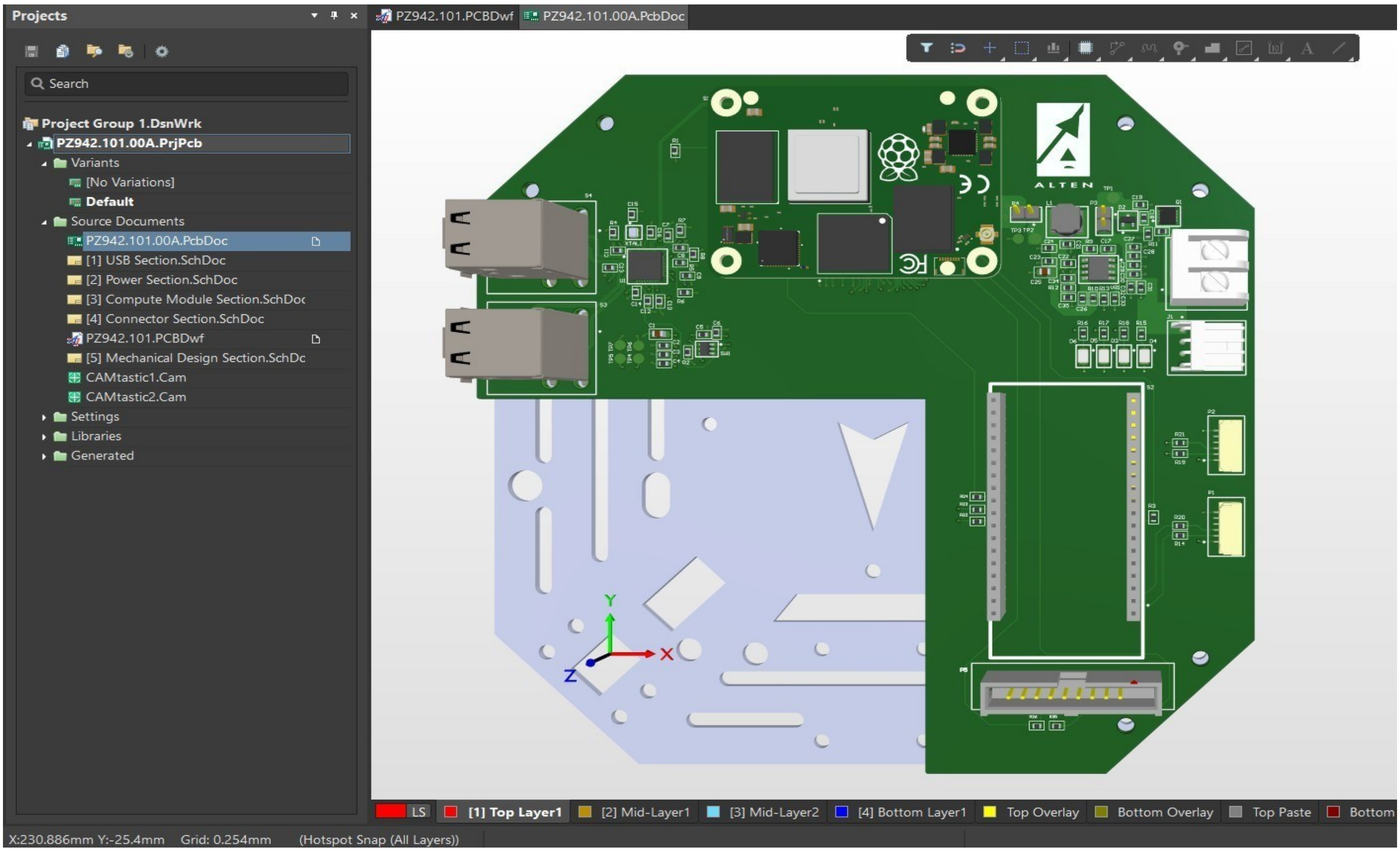

Figure 34: Last Version of the Designed PCB in 3D

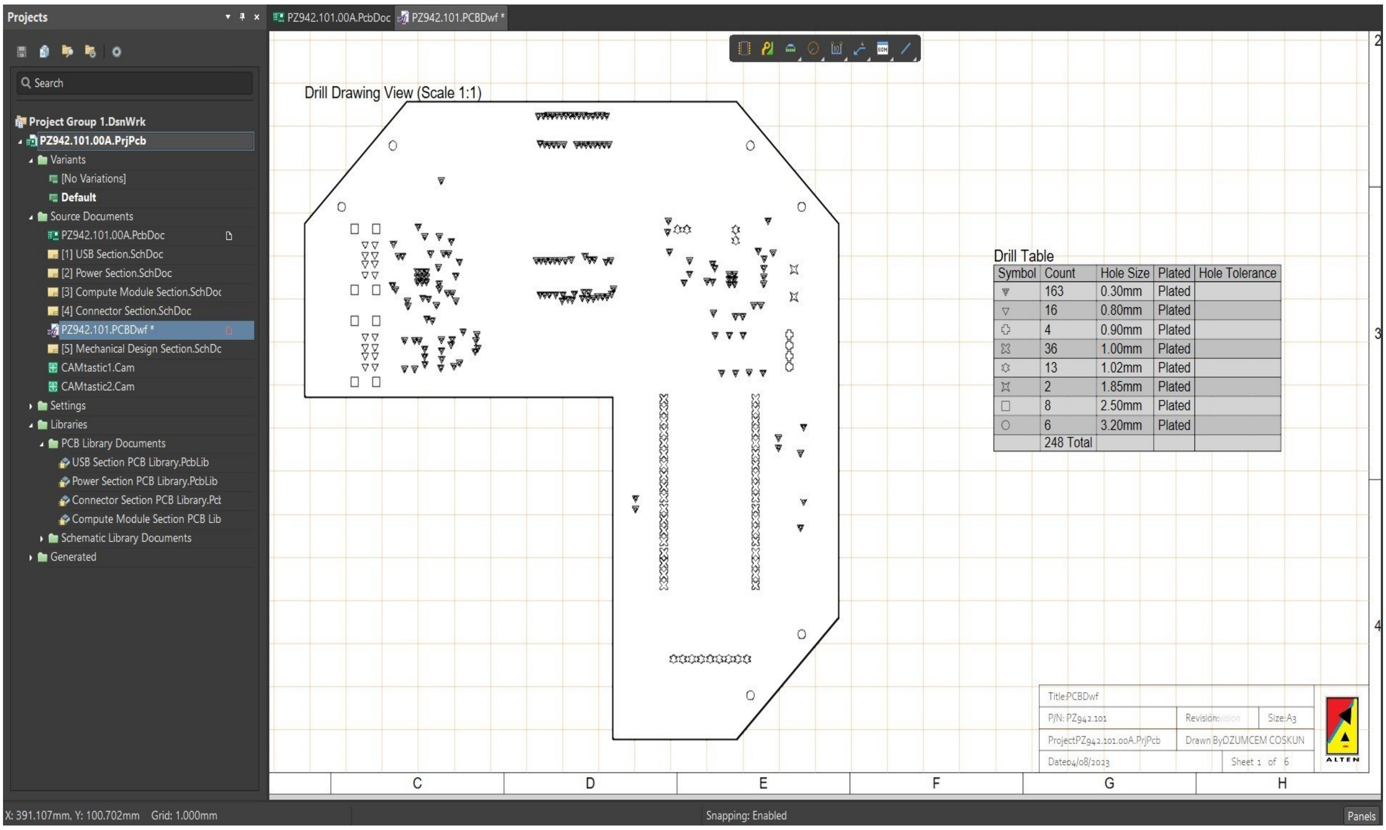

Figure 35: Drill Drawing View and Drill Table of the PCB

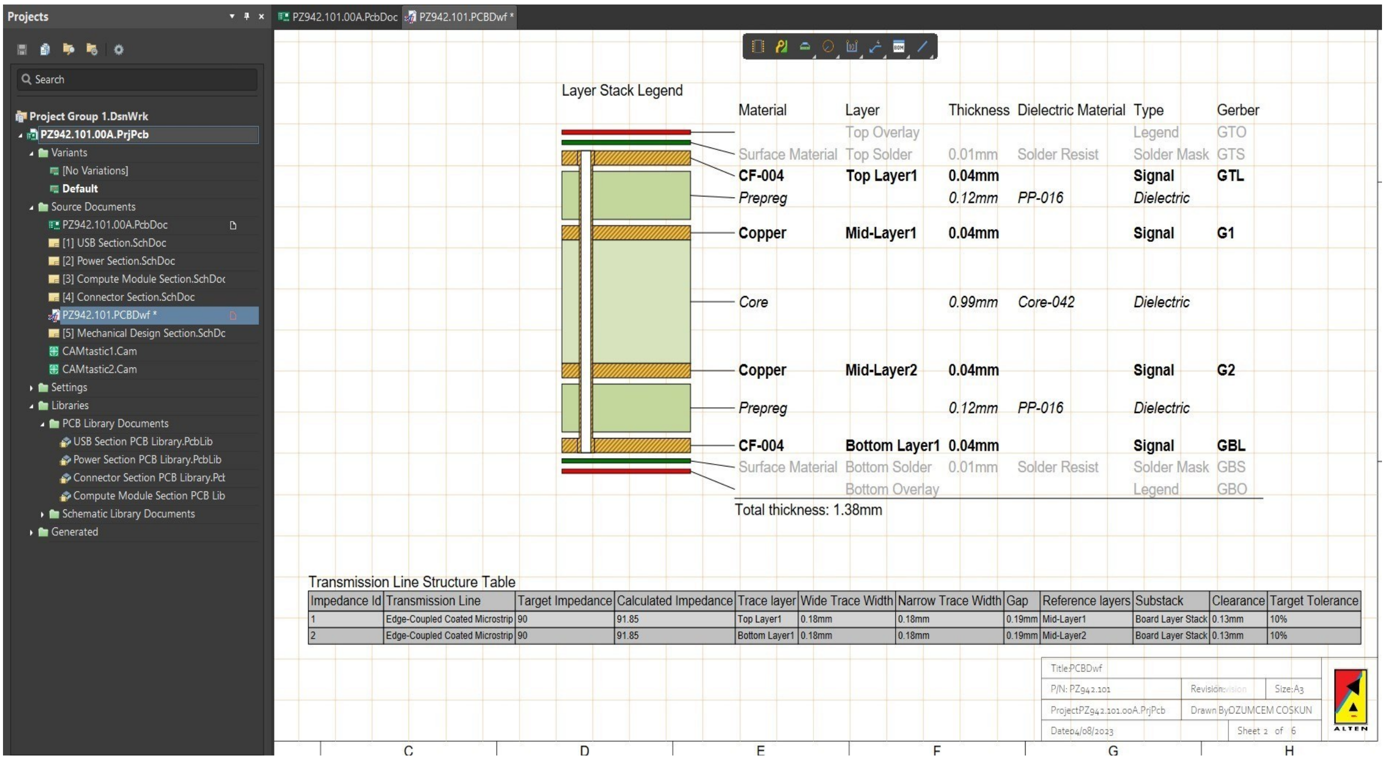

Figure 36: Layer Detail Information of the PCB

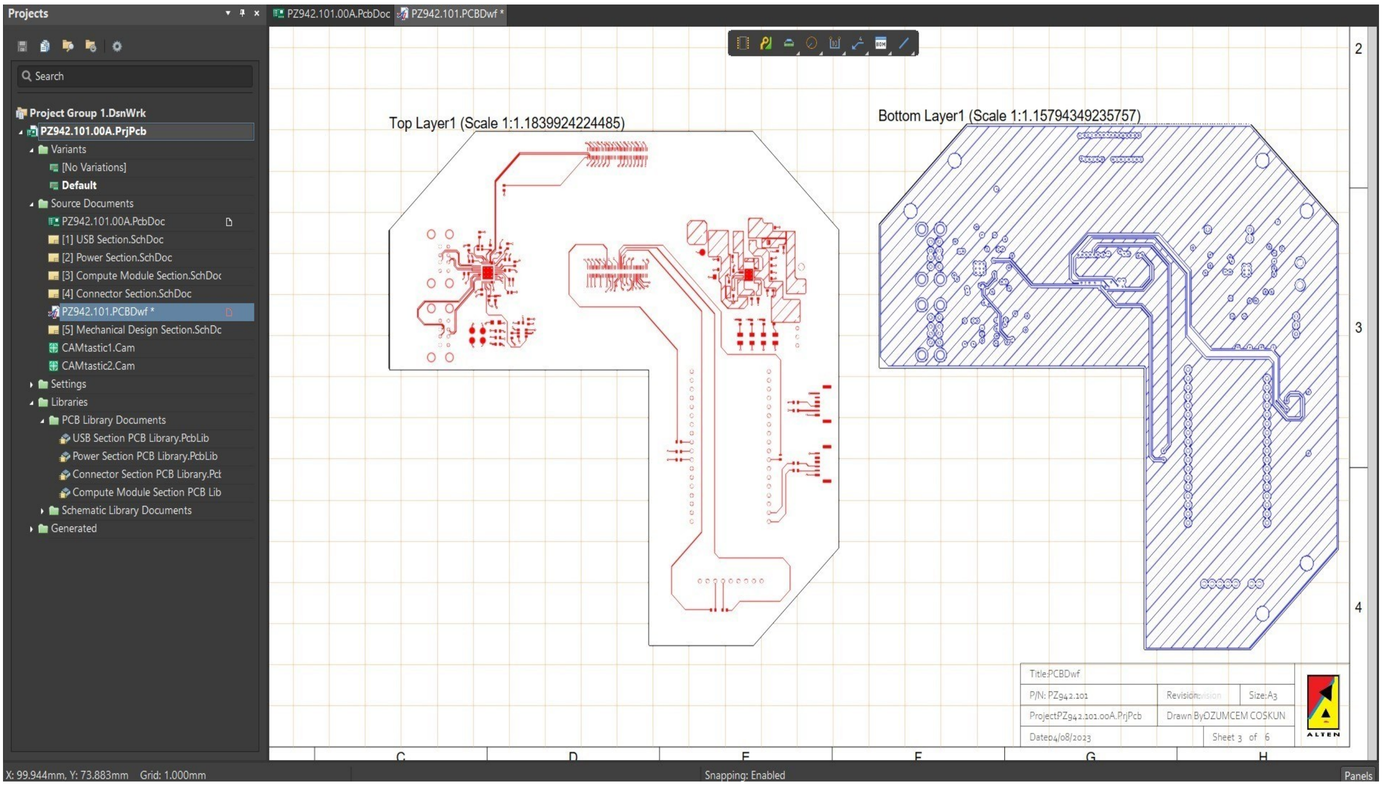

Figure 37: Top-Layer and Bottom-Layer Scales of PCB

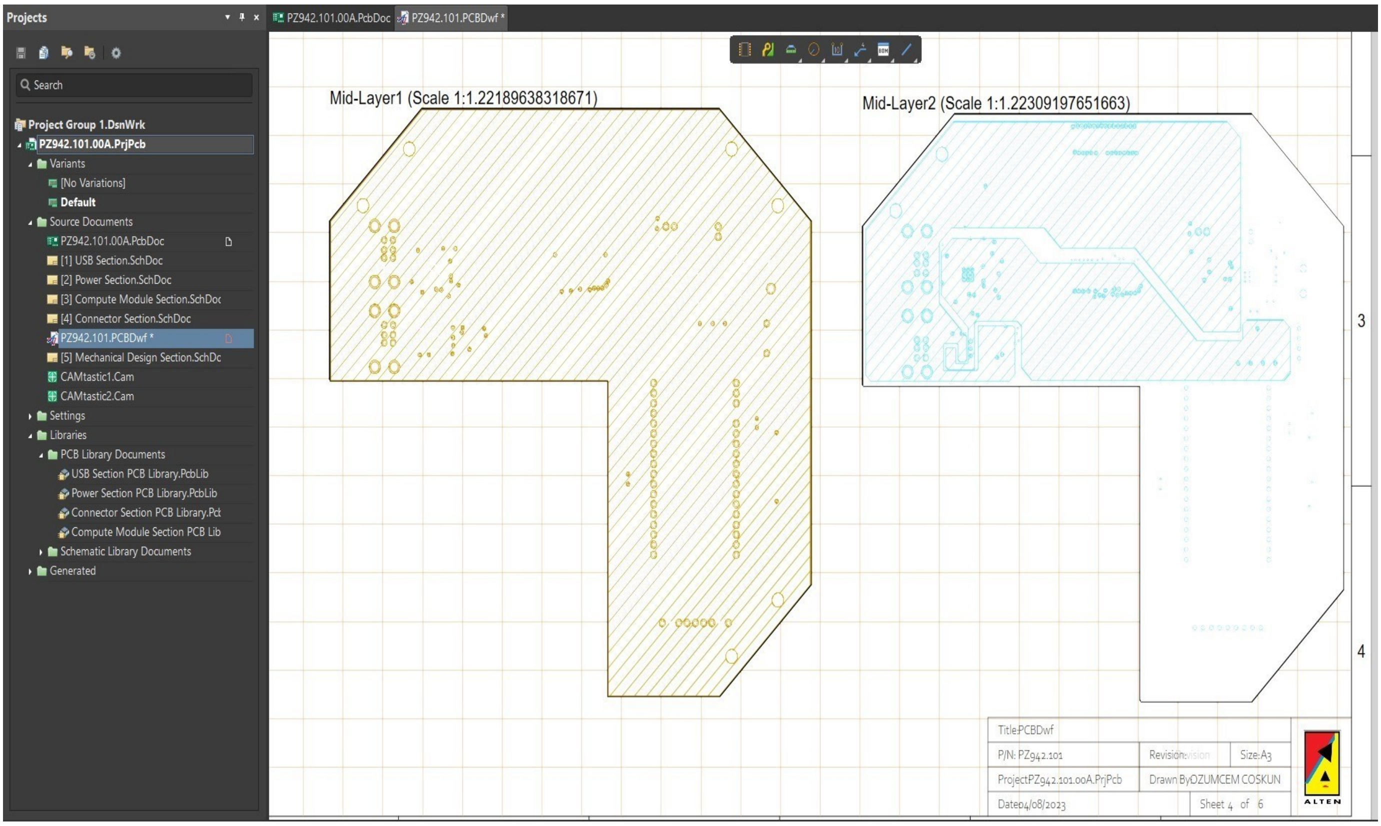

Figure 38: Mid-Layer 1 and Mid-Layer 2 Scales of PCB

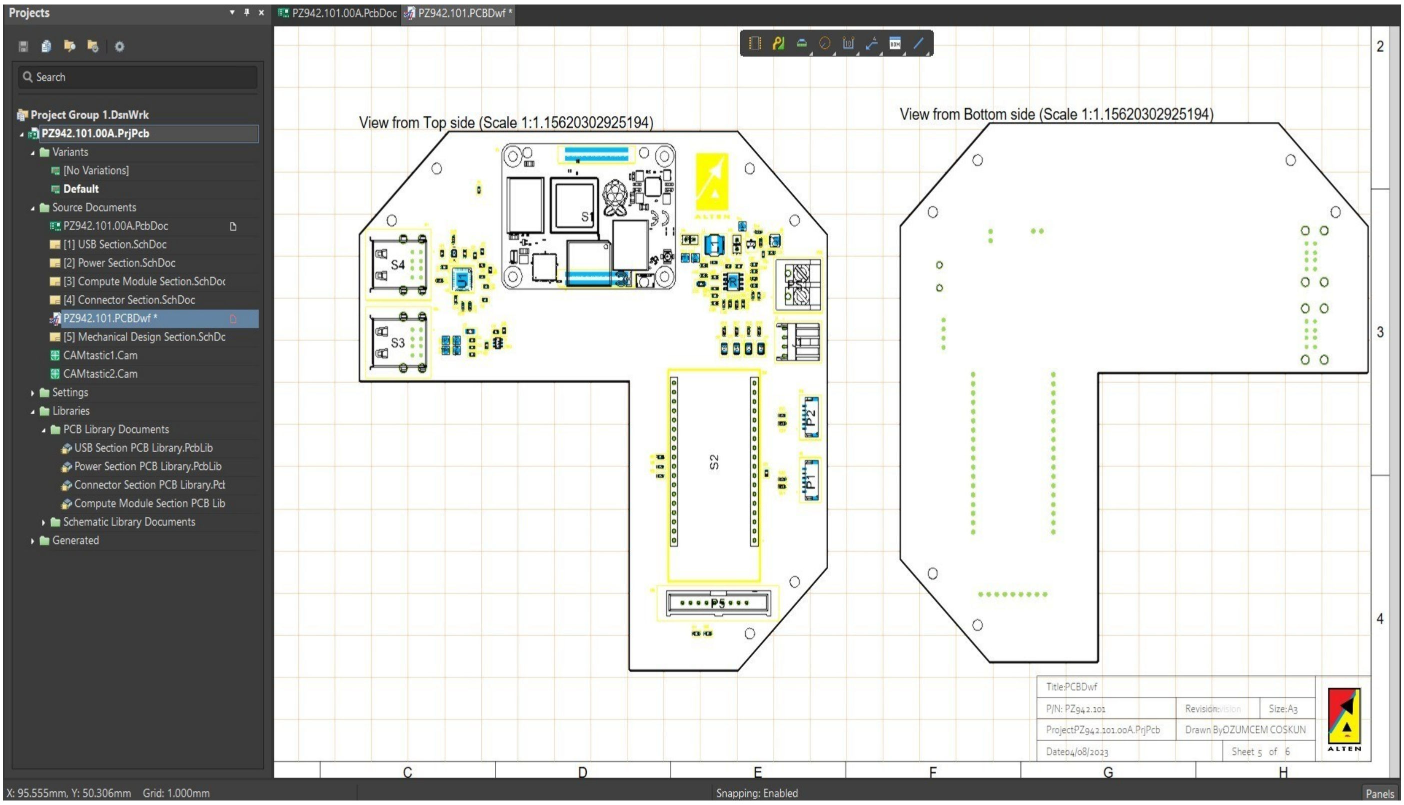

Figure 39: Top Side Scale and Bottom Side Scale of the PCB

V. CONCLUSION

Printed-circuit board has always been one of the main interests in the engineering fields. Especially in electrical and electronics engineering, this subject always got much attention to be learned and developed. As indicated in this thesis, there has been research about the printed-circuit boards with different perspectives. With the light of these works, in today's word, these boards are used in every part of our lives. However, there are still some uncertainties that makes researchers more and more curious about the board itself. Concepts like printed-circuit boards will never have an end and it will always be open to be explored more.

In this thesis, design of a printed-circuit board is researched both theoretically and practically. While doing that, one of the main sections of the printed-circuit board was explored which is the communication style of such boards. In addition to this, not only one but three different communication ways are used and achieved for the specific device, quadcopter. All the knowledge and research are mounted on printed-circuit board project. To be able to make the board more useful for the quadcopter project, single-layer printed-circuit board is rejected but multi-layer printed-circuit board type with four layers is preferred. From component selection, the circuit creation to end of the design, this thesis work provided an opportunity to experience the entire design steps of a real PCB. With the help of this thesis activity, the company ALTEN is now able to use the designed multi-layer board that can use three different communication protocols for the planned quadcopter project. Since PCB is a never-ending subject, there will always be more questions in minds to search and learn more in the upcoming period. Therefore, as a PCB designer, I strongly believe that the printed-circuit boards will be used more in our lives in more efficient ways and the challenges that the industry has with the structure of PCBs will be reduced with the help of these research activities.

VI. REFERENCES

[1] D. L. Jones et al., “PCB Design Tutorial,” June, 2004, [Online]. Available: https://alternatezone.com/electronics/files/PCBDesignTutorialRevA.pdf

[2] R. S. Khandpur et al., “PRINTED CIRCUIT BOARDS Design, Fabrication, Assembly and Testing,” McGraw-Hill, 2006, doi:10.1036/0071464204

[3] Anand, V. Singh and, V. K. Ladwal, “Study on PCB Designing Problems and their Solutions,” 2019 International Conference on Power Electronics, Control and Automation (ICPECA), 2019, doi:10.1109/ICPECA47973.2019.8975402

[4] M. S. Sharawi et al., “Practical Issues in high speed PCB design,” IEEE Potentials, 2004, doi: 10.1109/MP.2004.1289994

[5] M. Sharma, N. Agarwal and, S. R. N. Reddy, “Design and development of daughter board for USB-UART communication between Raspberry Pi and PC,” International Conference on Computing, Communication & Automation, 2015, doi:10.1109/CCAA.2015.7148532

[6] R.R.Pahlevi, A. G. Putrada and, M. Abdurohman, “Fast UART and SPI Protocol for Scalable IoT Platform,” 2018 6th International Conference on Information and Communication Technology (ICoICT), 2018, doi:10.1109/ICoICT.2018.8528745

[7] Z. W. Hu et al., “I2C Protocol Design for Reusability,” 2010 Third International Symposium on Information Processing, 2010, doi:10.1109/ISIP.2010.51

[8] J. Chen and S. Huang, “Analysis and Comparison of UART, SPI and I2C,” 2023 IEEE 2nd International Conference on Electrical Engineering, Big Data and Algorithms (EEBDA), 2023, doi:10.1109/EEBDA56825.2023.10090677

[9] Y. D. Save, R. Rakhi, N. D. Shambhulingayya, A. Srivastava, M. R. Das, S. Choudhary and, K. M . Moudgalya, “Oscad:An open source EDA tool for circuit design, simulation, analysis and PCB design,” 2013 IEEE 20th International Conference on Electronics, Circuits, and Systems (ICECS), 2013, doi:10.1109/ICECS.2013.6815548

[10] S.Muralikrishna and S. Sathyamurthy, “An overview of digital circuit design and PCB

design guidelines- An EMC perspective," 2008 10th International Conference on Electromagnetic Inference&Compatibility, 2008.

[11] B. R. Archambeault, and J. L. Drewniak, "PCB DESIGN FOR REAL-WORLD EMI CONTROL," Springer Science+Business Media, 2004, doi: 10.1007/978-1-4757-3640-3

[12] M.Chakraborty, J. Kettle, and R. Dahiya, "Electronic Waste Reduction Through Devices and Printed Circuit Boards Designed for Circularity," IEEE Journal on Flexible Electronics, 2022, vol. 1, pp. 4-23, doi: 10.1109/JFLEX.2022.3159258

[13] C. Marques, J. M. Cabrera, and C. F. Malfatti, "Printed circuit boards: A review on the perspective of sustainability," Journal of Environmental Management, 2013, vol. 131, pp. 298-306

[14] D. Chen, J. Hsiao, Y. Ling, C. T. Chen and, X. K. Cai, "Unified and efficient power integrity impedance target flow for flexible PCB design," 2017 12th International Microsystems, Packaging, Assembly and Circuits Technology Conference (IMPACT), 2017, doi:10.1109/IMPACT.2017.8255936

[15] G.M. Leon, E. G. Dominguez and, C. Sifuentes, "Processing files for manufacturing printed circuit boards," Procedia Engineering 35.240-244, doi: 10.1016/j.proeng.2012.04.186

[16] Z. Peterson et al., "Customize Your PCB Drill Sizes With Altium Designer," June, 2021, [Online]. Available: https://resources.altium.com/p/pcb-drill-sizes