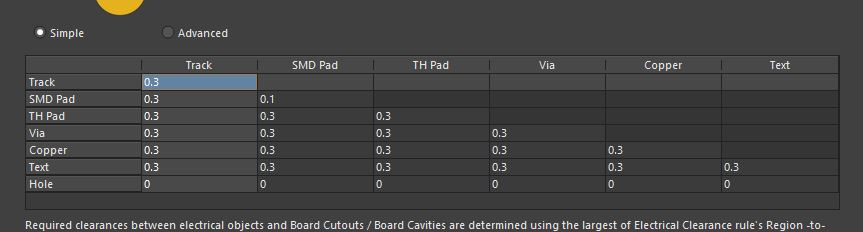
**PCB Design Report**

1. **Rules and Validations**

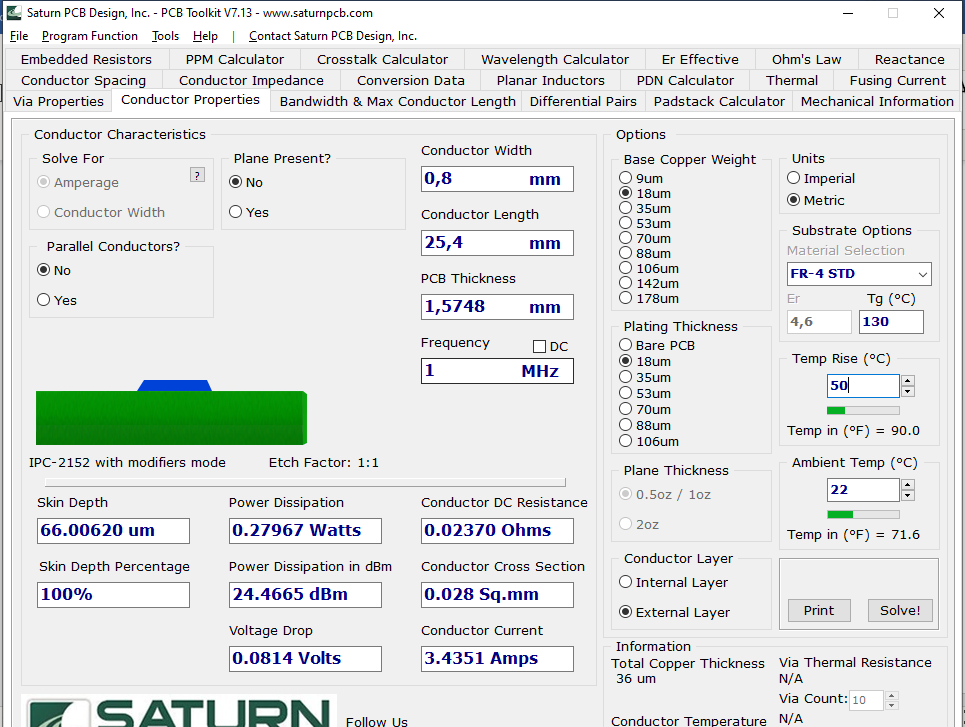
After concluding our simulation report and feedback session, we have made some corrections on component selection and schematic design, which is discussed above, then we moved into PCB design part. As we stated before, our main idea is to have a compact, reliable, adjustable and high power dense electronic card, so we have counted these features during design process.

In this project, we have used Altium Designer 20, and our design has two layers, and both layers include components. Before starting our design, we have determined the design rules to have a reliable and safe card, which can be seen in figure 1. However, for medium voltages (150-300V), according to IPC2221A, clearance between lines should be minimum 1.25mm, for our medium voltage tracks (rectifier), we have used that minimum value. For 15V and 30V range, this spacing should be minimum 0.25mm, where our clearance rule satisfies that value.

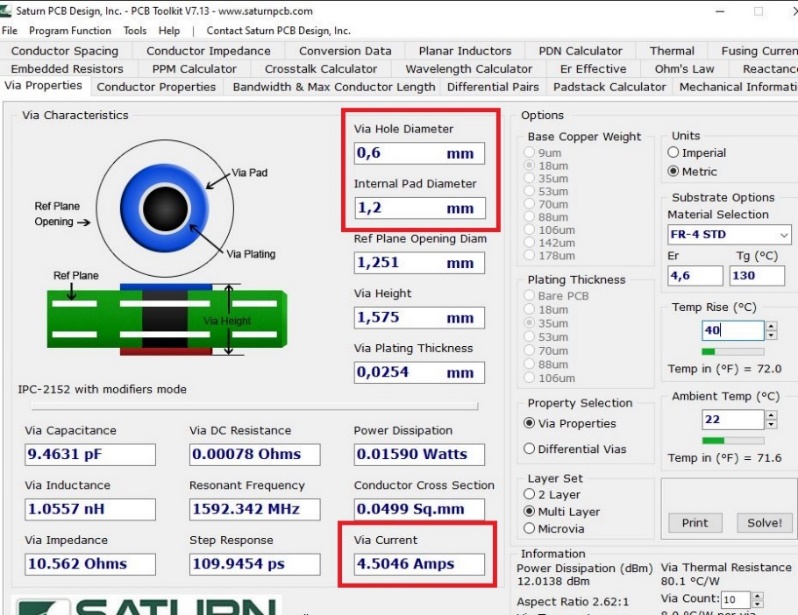


**Figure 1:** PCB Design Rule (Clearance)

From simulations, we have seen that, in rectifier and buck converter, we have a maximum 2.5A current, and our lines and vias should satisfy this value. For rectifier and buck converter lines, we used fixed 0.8mm line width, and minimum 0.6mm via hole, 1,2mm via pad to stay in safe zone. For other low current lines, we have used various line width between 0.3mm and 0.5mm, and we used fixed via dimensions which are 0.4mm hole and 0.8mm pad. We have validated rectifier line and via ratings from Saturn PCB Toolkit, which can be seen in figure 2 and 3 respectively.

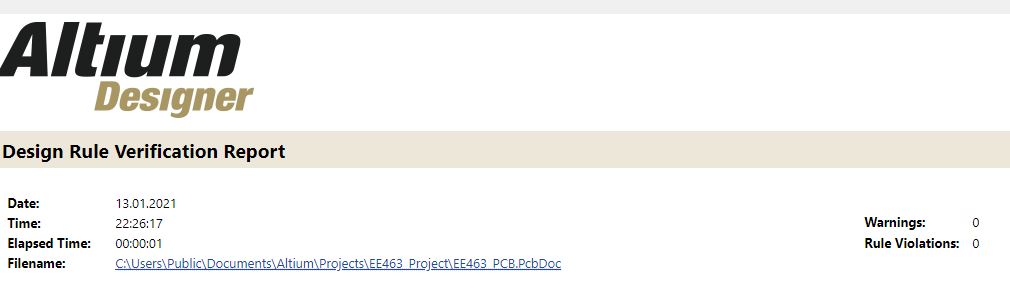


**Figure 2:** Line Current Validation



**Figure 3:** Via Current Validation

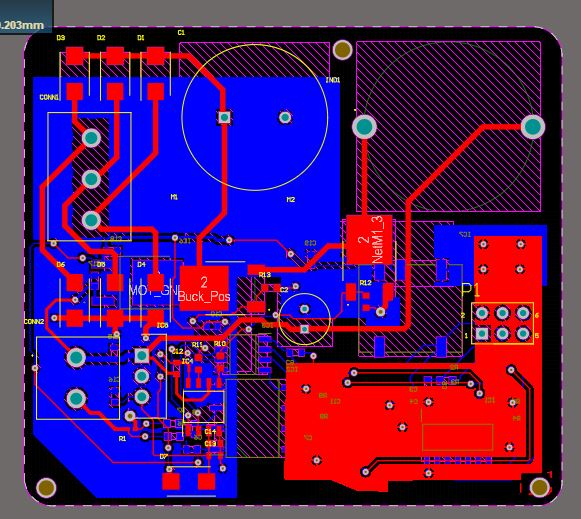
After finalizing our rules and getting validation for our pre-design about lines and vias, we have constructed our PCB design, and when we finalized that design, we used the Design Rule Check of Altium Designer, to see violations and connection problems. And finally, we have achieved a card without any error, which can be seen in figure-4.



**Figure 4:** Design Rule Check

1. **2D Design**

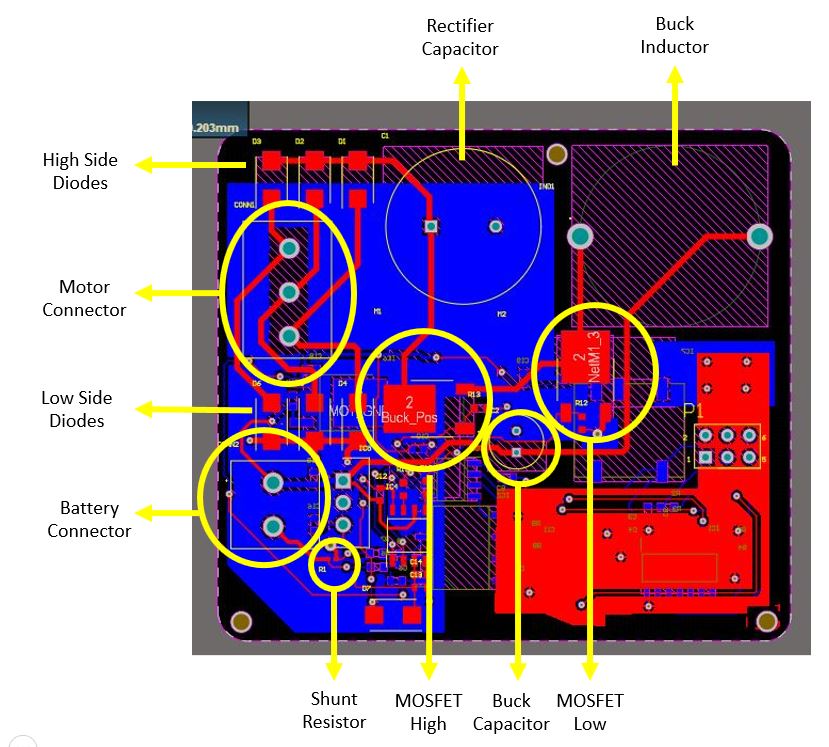
In order to have a small, compact and a useful design, we have placed our components to both layers. Generally, we have placed the isolated part into bottom layer (digital components), because we have used a lot of vias in this section of the board, and if we did this in rectifier or buck converter part, it would be problematic due to high currents, for vias. Overall view of our PCB design can be seen in figure 5.



**Figure 5:** Overall PCB Design

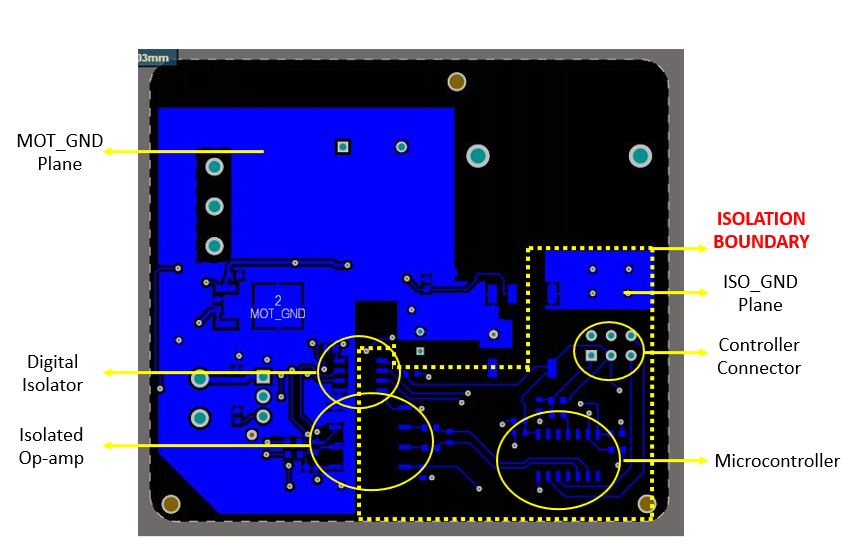
As we mentioned before, we have arranged medium voltage (300V), clearance as 1.25mm minimum, and for other lines we have used clearance of Design Rule, which is 0.3mm. Moreover, according to IPC rules, difference between PCB pad and line/device pad should be minimum 0.4mm, we have applied that rule, too.

If we look to placements of our components on board, we will firstly investigate rectifier and buck converter components. In this part, we have biggest components on the board. Moreover, heatsinks of MOSFET’s have taken a lot of space on board, however our heatsink places on the DPAK MOSFET, and wings of the heatsink placed with offset from board, so we have placed some of the small components under these wings, which helped us to decrease the dimension of the board. Rectifier and buck converter component placement can be seen in figure 6.



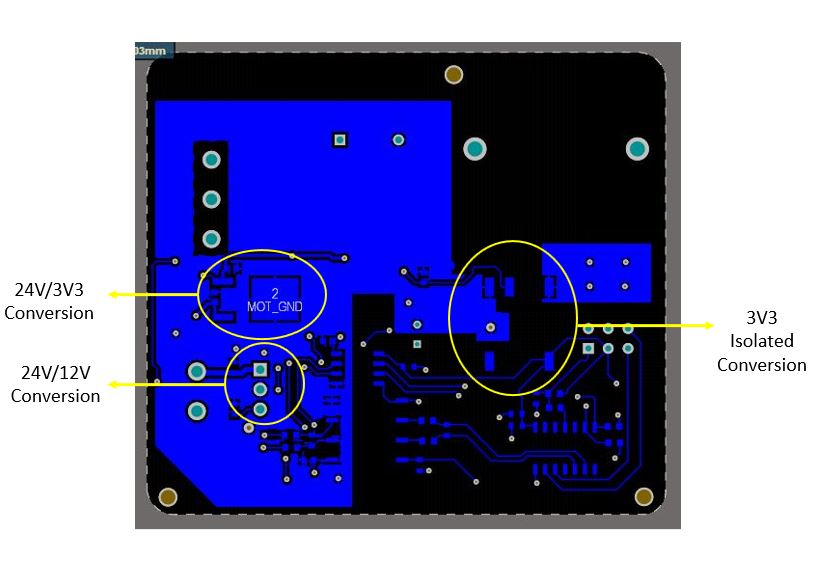
**Figure 6:**  Rectifier and Buck Converter Placement on PCB

In component placement, we will secondly investigate isolation boundary and digital part of our design. According to the datasheet, we should not take any line from the bottom of the isolator, however, placing a thin ground plane would not be problematic, as we did. We have left enough space between MOT\_GND and ISO\_GND, which are our grounds for motor side, and digital side, respectively. Placement of digital components, isolation boundary and planes of board can be seen in figure 7.



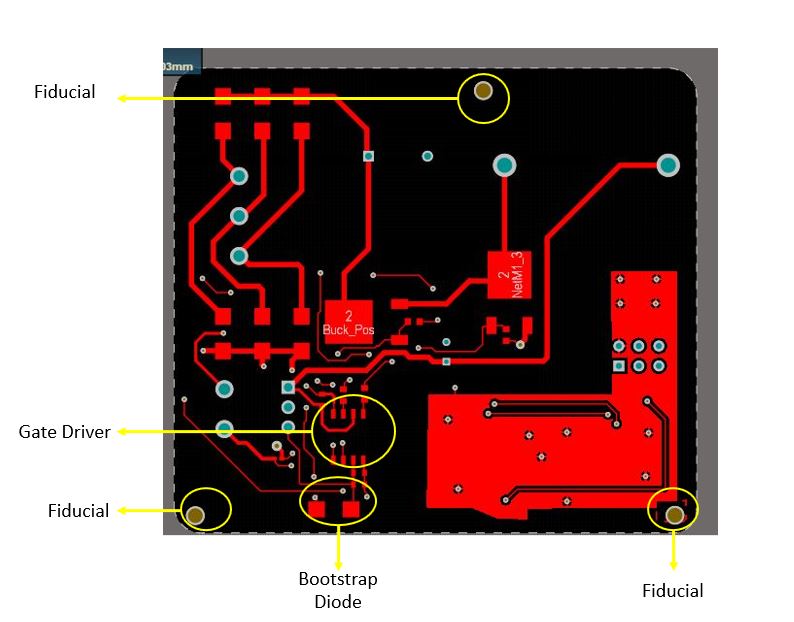
**Figure 7:** Isolation and Digital Component Placement on PCB

Thirdly, we will investigate power conversion units of our design. 24V/12V and 24V/3V3 components are placed close to Battery Connector, and 3V3 Isolator placed between isolation boundary, and ground pins are connected to corresponding ground, which has given to us a reference boundary distance. Placement of power conversion units can be seen in figure 8.



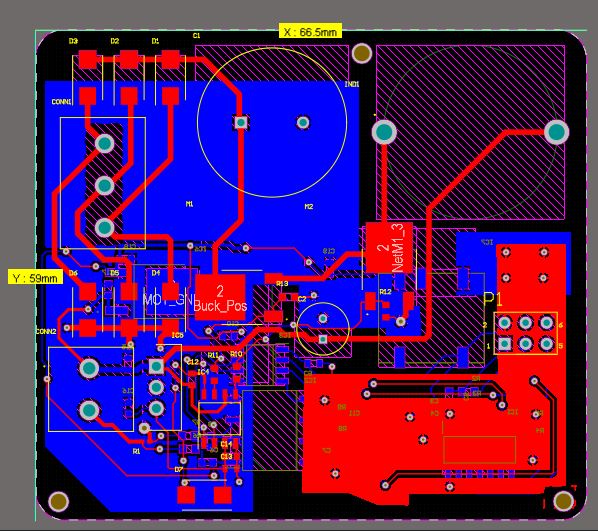
**Figure 8:** Power Conversion Component Placement on PCB

Lastly, we will investigate gate driver and mechanical connection placement. We have placed our gate driver near to 24V/12V converter, because we use 12V only in our gate driver. We have placed metric-2 fiducials, in order to immobilize our design in a box. Gate driver and fiducial placement can be seen in figure 9.



**Figure 9:** Gate Driver and Fiducial Placement on PCB

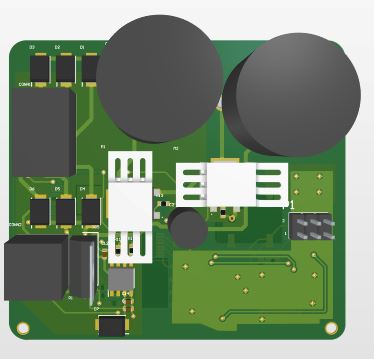
We have investigated component placement (except discrete capacitor and resistors) on our PCB for 2D design. In 2D, we will lastly investigate dimensions of our board. As we mentioned before, we have placed our components both of the layers, and we have placed some of the small components under heatsink trace, to have a small and dense design. Moreover, we have constructed our design by obeying clearance rules in order to have a safe and reliable product. At last, we have a card with 66.5mm X 59mm dimensions. In figure 10, card dimensions can be seen.



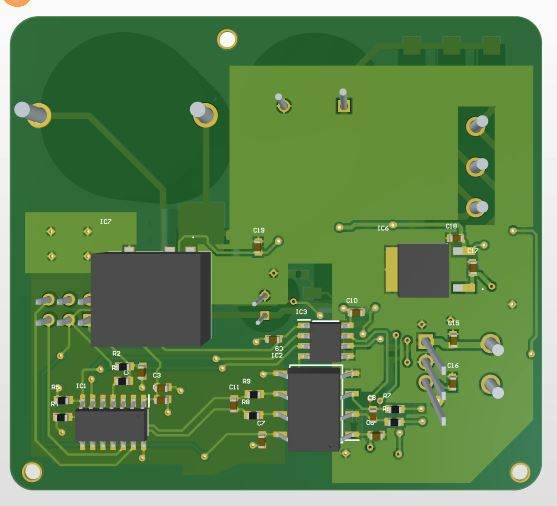
**Figure 10:** Board Dimensions

1. **3D Design**

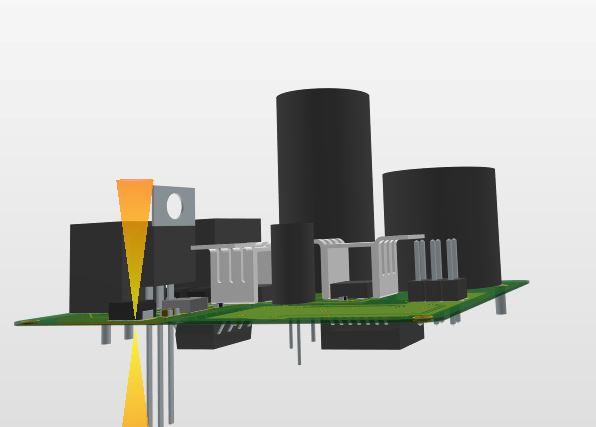
To have a reliable and professional design, we have placed 3D designs of our components to our component PCB library. Moreover, by constructing 3D design, now we are ready to select/design box for our board, which shows us that our design is an industrial product. Top view, bottom view and side view of our PCB design can be seen in figure 11, 12 and 13, respectively.



**Figure 11:** Top View of 3D PCB Design

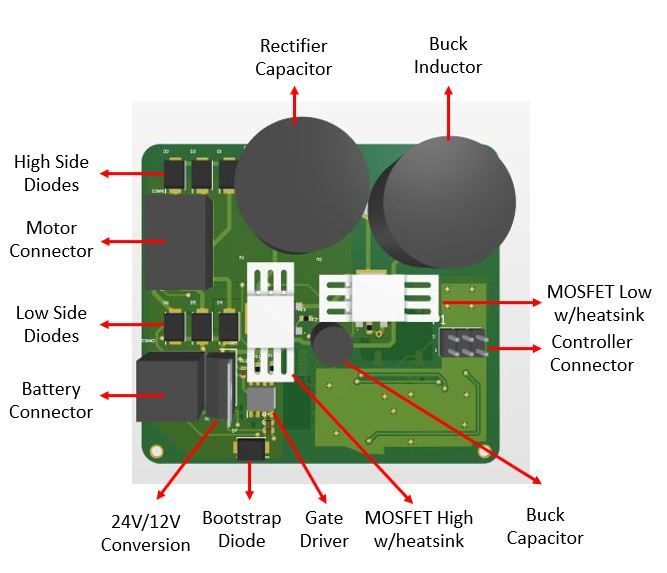
****

**Figure 12:** Bottom View of 3D PCB Design

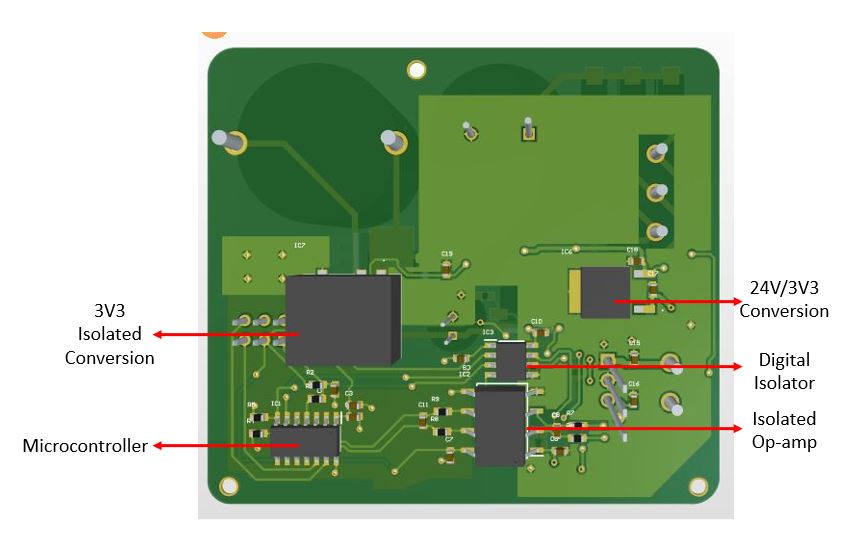


**Figure 13:** Side View of 3D PCB Design

We gave shown our component placement in 2D design part, however realizing these components in 3D and comparing with 2D explanations could give a better understanding of our design. Component placement on 3D model for top and bottom layer can be seen in figure 14 and 15, respectively.



**Figure 14:** Top Layer Component Placement for 3D

****

**Figure 15:** Bottom Layer Component Placement for 3D

At the beginning of the PCB design report, we have discussed the current ratings of our circuit, and used 18um Base Copper Weight and 18um Plating Thickness, which sum up to 1 oz Cu, in our 2-layer circuit. To have a reliable card, we will use 1.6mm card thickness. We will use the card material FR-4 because, compared to Aluminum one FR-4 is more efficient because conduction loss of Copper is lower than conduction loss of Aluminum. Moreover, due to material of FR-4 it is more resistive to water exposure. To have a low price, we will select transition temperature (Tg) as 130-140.

At the top layer, the rectifier capacitor has the maximum length. When we look datasheet of ESH series, we see a list for size codes and our capacitor’s code is N2, which means our longest component on top layer is 38mm, including tolerance. When we look bottom layer, the 3V3 isolator has the maximum length, which is 6mm, including tolerance. As we discussed, our board thickness is 1.6mm and if we put 10mm tolerance for box for soldering or screwing, our total box vertical length will be 55.6mm. In 2D Design section, we have stated our board’s horizontal dimensions as 59mm X 66.5mm and if we put 10mm margin for both X and Y axes to place our board into box correctly, **total dimensions of our box will be 69mm X 76.5mm X 55.6mm.** Which means, **total volume of our industrial product is 293484.6 .**

1. **Conclusion of PCB Design**

In this part of the report, we firstly discussed rules and validations to prove reliability of our design. Then, we have discussed and explained our 2D design, by referring component placement, isolation boundary and fiducials. Lastly, we have shown our 3D design and explained component locations. Moreover, we have discussed board parameters and lastly, we have designed our box dimensions and calculated volume of our design.

In PCB design, we believe that we are tried to work as a professional by considering all of the possible problems (line spacing, line width, via hole dimension, isolation boundary, copper thickness etc.). And we believe that, we have created a usable (programmable for different current ratings), reliable and compact design.