Journal of Engineering Science and Technology
9th EURECA 2017 Special Issue September (2018) 27 - 39

DESIGN OF A FULLY AUTOMATED VENTILATION SYSTEM FOR VEHICLES SUBJECTED TO DIRECT SUNLIGHT

AHMED M. ABDELWAHAB[1], AHMED A. OMER[1],
OBAI YOUNIS[2,1,*], DANIA AHMED[1]

[1]Faculty of Engineering, University of Khartoum, Khartoum, Sudan
[2]College of Engineering at Wadi Addwaser, Prince Sattam Bin Abulaziz University, KSA
*Corresponding Author: o.elamin@psua.edu.sa

Abstract

A study has been conducted to investigate the heat trap phenomenon inside a vehicle's compartment produced by the temperature rise due to sun radiation, the inside of the cabin will exhibit higher temperature compared to the outside surroundings. The amount of heat trapped in the enclosed compartment will bring various health risks, suffocation and uncomfortable feeling to both the driver and the passengers. More conditioning power is necessary to cope with this phenomenon which can severely shorten the lifetime of the compressor unit. This research is intended to design a ventilation system to lower the inside temperature of the compartment and eliminate the dangers of exposure to this high temperature. The investigation utilizes CFD analysis using ANSYS FLUENT commercial package to determine the best position for an axial flow fan to optimize air distribution, three different positions were considered and the numerical study was assessed with a scaled down prototype and a temperature drop of 26 ^{o}C was achieved.

Keywords: Axial flow fan, CFD, Heat trap phenomenon, Sunlight radiation, Ventilation.

1. Introduction

Generally, in countries like Sudan where the temperature rises up to 45°C in summer, re-entering a car that parked directly under the sun can cause uncomfortable feeling and suffocation. There are also major health risks attributed to this temperature rise as a result of breathing contaminated air inside the vehicle produced by the plastic accessories inside. Heat stroke in an adult can occur when body temperature exceeds 5°C, children can reach dangerous body temperatures at much faster rate. Studies indicate that each year children die from Hyperthermia (an acute condition that occurs when the body absorbs more heat that it can dissipate) as a result of being left in a parked car [1].

This extreme rise in the temperature levels inside the parked car cabin is caused due to the heat transferred by a combination of conduction, convection and radiation. Conduction occurred in the stagnant volume of air inside the car cabin, while convection takes place as the air inside the cabin contacts the cabin material itself. Moreover, heat is being radiated form the glass and body of the car, however, in this case radiated heat is the most influencing factor. The temperature variation inside the car depends on the thermal radiation exchanged between the environment and body of the car and also the radiation absorbed and emitted by the interiors of the cabin. In order to address this problem various approaches have been adopted. For instance, an improved ventilator system that comprises of a high volumetric capacity fan and motor powered by solar panel was proposed by Innovage Company [2]. Another study adopted an automatic ceiling fan using PIR sensor by Md Radzi [3].

This study is considering an axial cooling fan which will be located between the back speakers' area behind the rear seat controlled by an Arduino board based circuit that functions according to analogue signals received from LM35 sensors and based on these readings it operates, utilizing a dedicated battery. The microcontroller based circuit senses the temperature difference between inside and outside of the car, with the help of LM35 sensors, and depending on the temperature difference switches the fan on or off. To determine the specifications of the fan and its placement, temperature variation of a parked car directly under the sun was observed and flow simulation and analysis were performed in ANSYS FLUENT workbench where analysis of HVAC systems based on numerical calculations with sufficient accuracy and acceptable results is now possible for HVAC researchers by using improved computer technology and CFD techniques [4]. A model of a 2012 Hyundai Accent was tested in different conditions. After the simulation was completed an experiment was performed on a prototype, fabricated based on a scaled down model of the vehicle, to observe the performance, and assess the installation and configuration of the smart ventilation system.

2. Theory of Heat Analysis in Vehicles

The Heat Balance Method (HBM) was used for the determination of the cooling load inside the compartment [5] and developed according to [6]. The variation of temperature inside a car was taken from a previous study conducted by Elbadawi et al. [7], in which they found that the average temperature inside the compartment can reach up to 78 °C while the outside temperature in summer can be as high as 46°C. While another study made in Iraq proved that the air temperature inside the cabin reaches 70°C and dashboard temperature approach 100°C [8]. These values were used in the cooling load formulas to determine the ambient load, Direct Radiation load, Diffuse Radiation load and Reflected Radiation load. All these values were added together to obtain the total cooling load of the vehicle which was 2.31 kW. The total cooling load is significantly larger than the natural ventilation, which is measured in a previous study and found approximately 1 kW [9].The heat flux was calculated for each glass and 25% of each value was assumed to be absorbed by the front and rear seats, these values is shown in Table 1.

As assumption was made that heat is only transmitted to the vehicle through the glass and all other surfaces are insulated. These values were verified using a solar power meter which is an instrument used to measure the heat flux from solar radiation for a unit surface area.

Table 1. Heat flux values for each glass.

Boundary Surfaces	Heat flux (W/m^2)
Front Glass	739.30
Front passengers Glass/glass	555.90
Front Sear	616.98
Rear Passengers Glass/glass	494.80
Rear Glass	736.39
Rear Seats	616.56

2.1. Setup and boundary conditions

After calculating the cooling load requirements, the flow rate was calculated and found to be a minimum of 137 CFM (cubic feet per minute). Catalogues from PELONIS Technologies Co. Ltd. [10] were used for the selection of the fan, the closet standard was found to be and axial fan 12 V fan with 150 CFM which discharges air with a velocity of 6.288 m/s. This value was considered as a boundary condition later on in ANSYS.

The geometry used is shown in Fig. 1. Dimensions were extracted from a Hyundai accent 2012. The turbulence model used to replicate the phenomenon is SST-Kω which was found suitable for this situation due to the combination of internal and external flow inside the compartment. The discretization scheme used was QUICK. The inlet fan was specified as a velocity inlet with 6.288m/s and the outlet was decided to be the natural outlet located under the seats. This outlet was chosen after a short survey in local workshops specialized in the maintenance of air conditioning systems. The two holes under the front seats can be located in various types of vehicles and automobiles. Then it was considered as Outflow in the boundary conditions, due to its natural flow outside the vehicle.

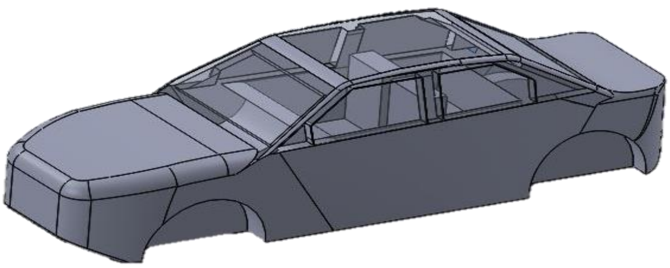


Fig. 1. CAD model of a 2012 Hyundai Accent.

2.2. Mesh generation

Due to the sensitivity of the case and complexity of the geometry it was decided to use ICEM software to generate the mesh. The advantage of ICEM is the ability to control the mesh quality aspects manually resulting in a mesh with quality over 90%, as shown in Fig. 2.

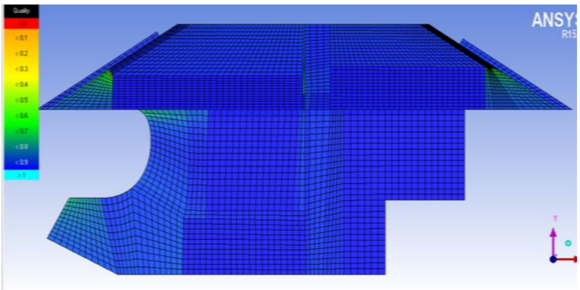


Fig. 2. Mesh quality specified by colour (Blue colour indicates high quality).

Four mesh sizes were chosen for the mesh independency study and are shown in table (2).

Table 2. Mesh sizes.

Mesh No.	No. of nodes	Min volume (m)	Max volume (m)
Mesh 1	144842	6.272E-8	6.72E-5
Mesh 2	381616	1.500E-8	1.90E-5
Mesh 3	205651	3.100E-8	4.60E-5
Mesh 4	256553	2.170E-8	4.34E-5

Since the study is of a transient nature, three-time step sizes were tested, and an optimum time step size was chosen in order to minimize the computational power. The time stesps were 0.1 sec, 0.5 sec and 1.0 sec.

Then two cases were studied and their respective results were observed. The first study used a constant heat flux calculated from the cooling load formulas, the second study used a constant surface temperature which was taken from a previous study by Khan [10] to be 81 °C. The purpose of this study was to analyze the effects of transient heat flux and difference between constant and transient heat flux on the glass. And indeed, the constant heat flux case yielded unrealistic results. In terms of temperature. It was obvious that the heat flux will decay to a point near zero as shown in Fig. 3.

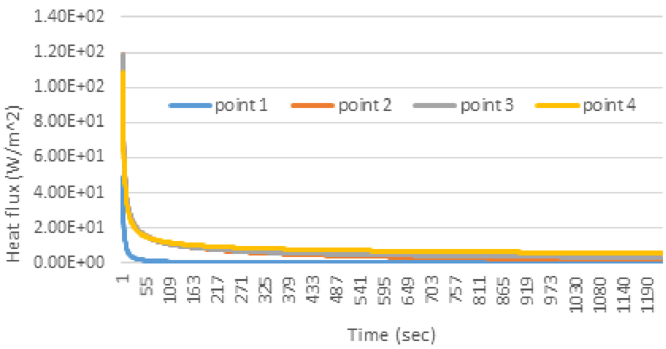


Fig. 3. Decay rate of the heat flux applied to the glass.

2.3. Air induction methods

The method, how air is being induced in the car cabin is essential for air distribution and consequently the rate of cooling inside the car cabin. Thus, it was decided to further modify it and run several cases to check for the best distribution of air and ventilation performance for the fan previously selected. Three cases were considered where the inlet port was changed in terms of orientation and number of fans, the three cases are (1) Single fan, (2) 30° tilted fan, and (3) Two fans with half the flow rate for each fan.

The first case was the first design proposal using the fan selected previously and was not subject to any modifications.

3. Results of Numerical Study

After conducting the mesh independency study for both constant heat flux and constant surface temperature, the optimum mesh was found to be the same which is mesh 4 with 256,553. The comparison was made on a line that contains 50 sample points and starts from the front glass and passes through the axis of the fan in the middle. The physical property used for comparison was temperature as it is the most important parameter in this study. Using error bars with $\pm$5% percentage error in the smallest mesh (mesh 2). The time Independency study was also made with the same technique above but using different location, which were 4 points located inside the cabin, as shown in Fig. 4. These 4 points were used later on as the position of the four sensors in the prototype:

1. Above front passenger head and near the front glass.
2. Near the driver head and next to the front passenger glass.
3. Near the front passenger seat
4. In the back seat

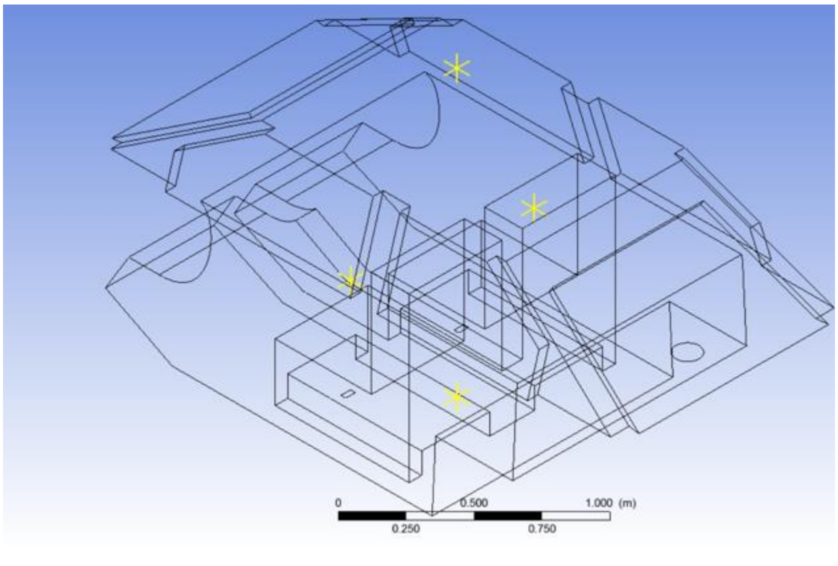


Fig. 4. Assigned positions of sensors.

These four spots where considered because they represent the targeted areas in which passengers will experience the heat rise when entering the vehicle. The optimum time step size among the three for both, the constant heat flux and constant surface temperature, was found to be 0.5 second.

After specifying the optimum time step and mesh size the 3 inlet cases were simulated to check for the best air distribution and ventilation efficiency and results of these 3 cases were taken at 5 seconds, 10 seconds and 150 seconds for two different cases which are.

Heating the cabin for 10 minutes then operating the fan, which can be considered the worst-case scenario the fan would have to face.

Operating the fan after 3 °C temperature difference between the inside and outside is reached, which is the way the prototype would be programmed to do in the experiment.

The difference between the above mentioned cases was not considerable and whether heating the cabin, or operating the fan after temperature difference reaches 3 °C between inside and outside, the temperature inside the compartment would not differ very much for the 3 inlet cases mentioned above as shown in Figs. 5 for each inlet case in the cooling period only.

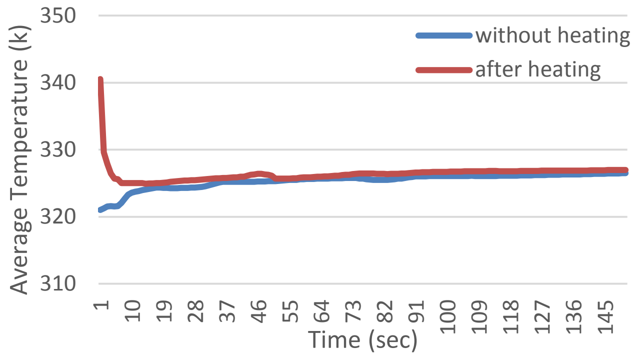
As observed, no differences are noticeable if the cabin is to be heated for 10 minutes then the fan operates in the three inlet cases or if the system is to operate when difference in temperature between inside and outside is 3°C. Thus, the system was designed based on the latter case to pertain the fan life time, and reduce the operation time it would take to cool down the vehicle.

The contours of temperature for the three inlet cases in the second approach of constant surface temperature are taken at different time intervals to monitor the distribution of air. The results are displayed in a vertical plane passing through the axis of the single fan as shown in Fig. 6.

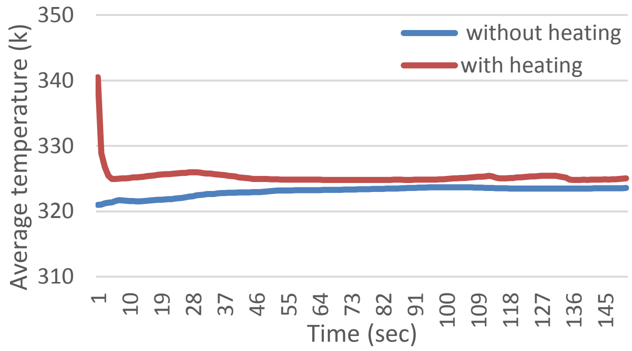
The system-regardless of the case used- in general achieved a very convenient temperature drop as shown in Figs. 7-10 for 150 seconds operation time.

From these figures, the following observations can be noticed:

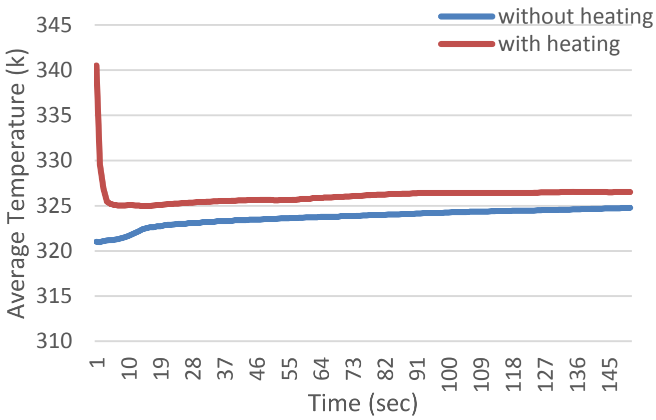
- The temperature distribution of the two fans is better and the temperature drop is higher than the other two cases proving that ventilation is best served using two fans.
- The effect of the rear glass on air streamlines in the case of a single fan caused the air to move to the middle of the cabin which can be considered a targeted area, however, it lacks the efficiency of ventilation and heat removal due to the major losses in kinetic energy of air caused by the stagnation of the streamlines in the rear glass.
- In the 30° tilted fan the air moves parallel to the roof of the cabin and when the streamlines reach the front glass area it pushes the high temperature streamlines to the front targeted area (driver seat). Also, the rear passenger seats heated up even more due to the poor air distribution in rear cabin part.



(a)



(b)



(c)

Fig. 5. Ventilation performance for the three cases (a) Single fan (b) Dual fan (c) 30° tilted fan.

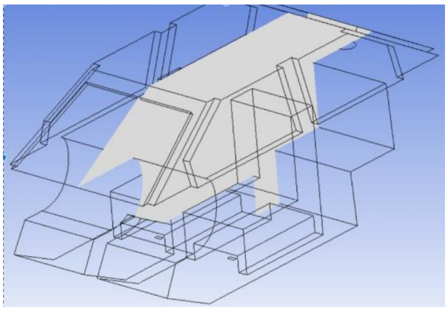
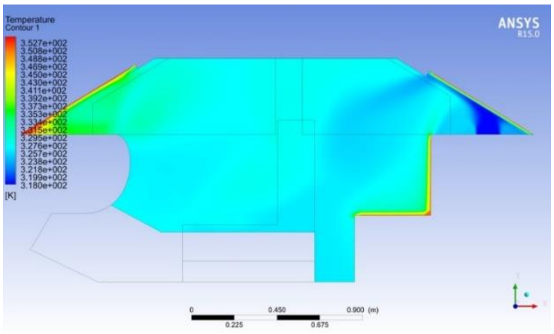
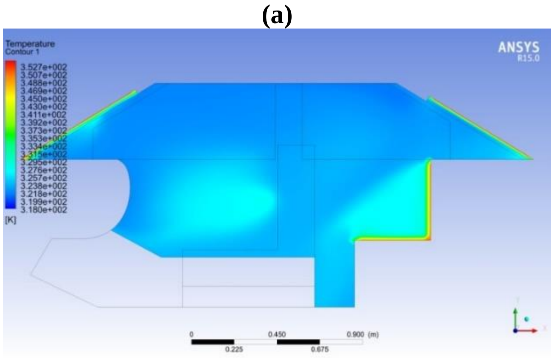


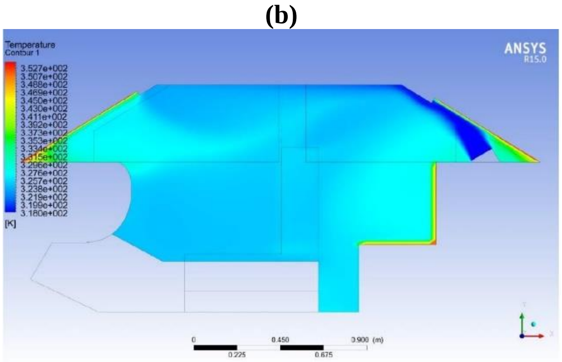
Fig. 6. Vertical Plane used to display the temperature distribution



(a)



(b)



(c)

Fig. 7. Temperature distribution for the three cases after 150 seconds operation time, (a) Single fan, (b) Dual fan, (c) 30° tilted fan.

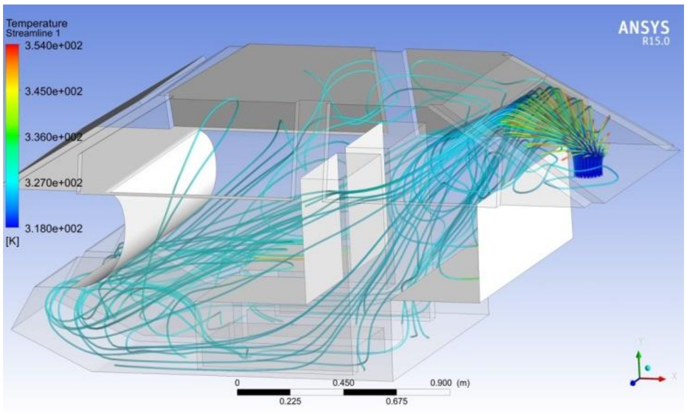


Fig. 8. 3D streamlines of air distribution in single fan case.

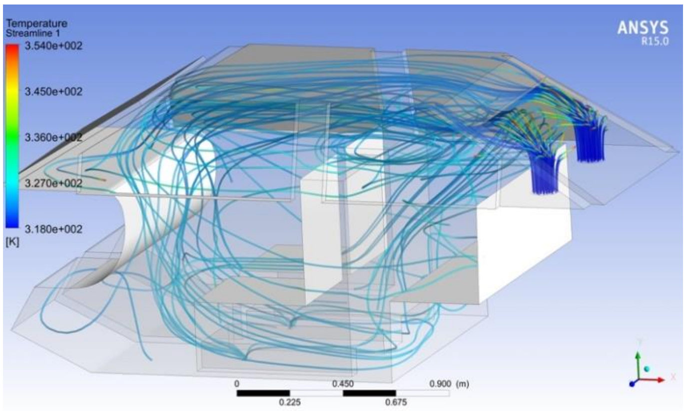


Fig. 9. 3D streamlines of air distribution in dual fan case.

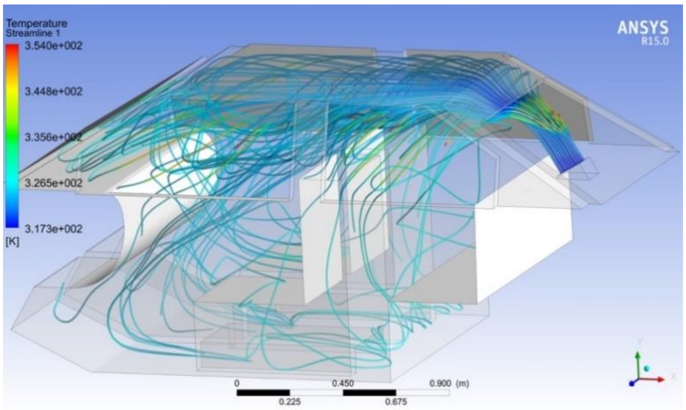


Fig. 10. 3D streamlines of air distribution in dual fan case.

The average temperature was calculated from the same 4 sensors points, mentioned before. Figure 11 shows the difference of temperature distribution between the three cases.

As observed, the average temperature was initially increasing; due to the heat distribution that occurs initially inside the compartment when the fans are operating. Though, the two fans achieved the highest efficiency of ventilation and heat removal. The average temperature of the compartment after 150 seconds in three cases; (1) Plain fan at 53 °C, (2) Two plain fans at 51 °C, and (3) 30° tilted fan at 52 °C.

It is noted that the difference between the three cases did not exceed 2 °C. Which-to the driver and passenger- would not be noticeable. Thus, for economic concerns, the single fan configuration was used to fabricate the prototype.

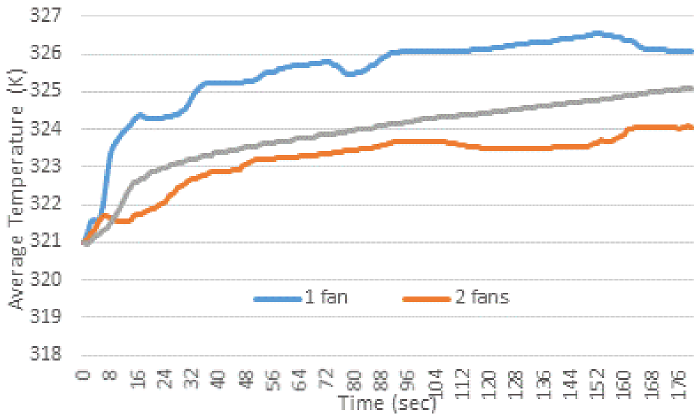


Fig. 11. Average temperature for the three cases as a function of time.

4. Experimental Setup and Results

The first step to fabricate and manufacture the prototype was to scale down the actual model for the purpose of minimizing the total cost of manufacturing. A scale of 1:3 was used and 2D drawings were prepared with the necessary dimensions for the manufacturing procedure. It may be noted that all the circular sections of the actual model were removed and replaced with sharp squared edges in order to simplify the procedure.The interior of the cabin model consists of a dashboard and rear seats. The other interior components were removed because it has been reported that the heat exchange between the car body and the car interior was very limited according to Tseng et al. [12]. Then the cooling load was calculated for the prototype to determine the required fan flow rate. Which was found to be 14.77 CFM.

The heat flux calculations were not necessary at this stage because the prototype was manufactured for the purpose of testing the system reliability and performance.

The process used L-angle steel to make a skeleton of the prototype then this skeleton was covered using mild steel sheets, as shown in Fig. 12, due to its availability and low price in market. The sheet metal was attached by means of arc welding. After the process if finished. The two outlet holes were drilled in the positions mentioned earlier.

Fig. 12. Manufacturing process of a scaled down prototype.

After finishing the manufacturing process, the prototype was covered with black paint to enhance the heat transfer in order for the sensors to heat up in a much faster rate. Then the prototype shown in Fig. 12, was covered internally with Aluminum foil-EPE foam insulator mats with thickness of 5 mm. the insulator was installed in order to achieve the assumption of restricting the heat flow only from glass.

Additionally a 5 mm SGG ANTELIO clear glass was installed produced by Saint-Gobain Co. Ltd. [10]. With a solar factor of 0.81 which expresses the transmissivity of the glass.

The electronic system used in the prototype consist of the following, Arduino Uno Board, LCD shield, LM35 temperature sensors, 12V battery, Type ULN2003 transistor array driver IC, 5×7 cm PCB board, 50×50×20 mm axial cooling fan.

The above components represent the complete electronic control system, where the Arduino board is programmed using (C/C++) to read the temperature from 4 sensors located in the four points specified above and take their average then subtract that value from the outside temperature and based on that difference the Arduino board will send a 5V signal to the ULN2003 driver which will convert it into a 12V signal to feed the cooling fan. The function of the LCD shield is only to display the average temperature and outside temperature of the prototype.

The 12V battery is directly connected to the ULN driver which feeds the fan. The axial fan used in the experiment had a flow rate of 20 CFM, 0.4A supply current and equipped with PWM ball bearings. The system was then assembled using jumper wires as shown in Fig. 13.

The results showed significant changes when compared to the numerical results because the readings were taken manually by the use of a stop watch. The fan operates based on a code written to discharge air when the temperature exceeds 3°C.

When the fan is observed during operation it fluctuates as the signal amplitude changes as shown by the green line in Fig. 14, in order to maintain the difference between the inside and outside below 3°C. The whole system is a simplified module of microcontroller module in [13].

The other reasons for the difference between the experimental and numerical can be concluded in points. There was no heat sink (front and rear seat) that convert into heat sources and emit heat to the compartment. The insulation used in the prototype did not express the real case scenario. It was only installed to restrict heat from glass only as assumed in the cooling load calculation.

Other than that, the material used to manufacture the prototype has different thermal conductivity and absorptivity than assumed.

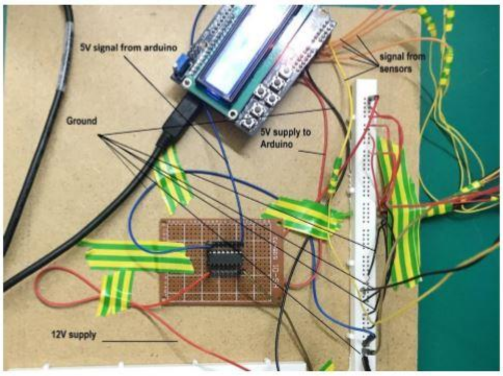


Fig. 13. Assembly of the control circuit.

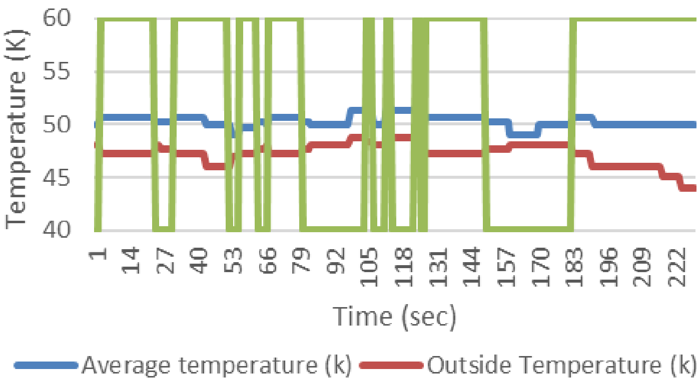


Fig. 14. Temperature and signal diagram.

5. Conclusions

The reasons why the ventilation system could not reach the ambient temperature can be justified due to transient Heat flux: when the temperature inside the compartment increase, the heat flux decreases and approaches zero, this can also be true in the case of cooling when the temperature decreases the heat flux will increase to make up for the heat loss due to ventilation. Thus, the ventilation system will stop in a point where it cannot drop below it.

The Air Capacity: the inlet of the vehicle is relatively small resulting in a stagnant air stream in some areas, when the air consumes all of its heat capacity, it will not be able to draw any more heat out of the vehicle. Thus, to maintain an internal temperature similar to the ambient there are three possible choices to decide from. Increasing the outlet size, cooling air to a temperature below the ambient before it enters the compartment or reducing the distance between the fans and targeted areas to minimize the losses in kinetic energy.

Acknowledgments

We acknowledge the department of mechanical engineering at the university of Khartoum for funding this projects, we are also thankful for Eng. Hussam Hassan for assisting us with the electronic matters of this project.

References

1. Vaccination Information Network (2007). Heat trapped inside cars a greater health risk than infectious diseases. Retrieved December 5, 2015, from https://www.vaccinationinformationnetwork.com/heat-trapped-inside-cars-a-greater-health-risk-than-infectious-diseases/.
2. Innovage (2016). Innovage company solar ventilator. Retrieved January 10, 2016, from http://www.innovage.com/SolarVentilator.
3. Md Radzi, M.R. (2010). *Automatic ceiling fan using PIR Sensor*. Universiti Teknologi Malaysia.
4. Sevilgen, G.; and Kilic, M. (2012). Three dimensional numerical analysis of temperature distribution in an automobile cabin. *Thermal Science*, 16(1), 321-326.
5. Fayazbakhsh M., and Brahami, M. (2013). Comprehensive modelling of vehicle air conditioning loads using heat balance method. *SAE International*, Technical Paper, 2013-01-1507.
6. Kamar, H.M.; Senawi, M.Y.; and Kamsah, N. (2012). Computerized simulation of automotive air conditioning system: development of mathematical model and its validation. International Journal of Computer Science Issues, 9(2), 23-34.
7. Elbadawi, N.; Elhadi Z.; and Noureldeen, Z. (2014*). Car solar ventilation system to avoid glass overheating*. Final year project report, University of Khartoum, Sudan.
8. Aljubury, I.M.A.; Farhan, A.A.; and Mussa, M.A. (2015). Experimental study of interior temperature distribution inside parked automobile cabin. *Journal of Engineering*, 21(3), 1-10.
9. Mansor, M.S.F.; Abd Rahman, U.Z.; Zainal Abidin, M.S.; Md Zain, M.Z..; and Md Yusof. M.R. (2014). Variation of car cabin temperature influenced by ventilation under direct sun exposure. *Journal of Mechanical Engineering and Sciences* (*JMES*), 1014-1023.
10. Khan, M.U. (1999). Development of ventilation technique to limit the car cabin temperature under a blazing sun. University of Toyama.
11. Saint Gobain (2015). SGG Antelio clear glass. Retrieved December 5, 2015, from http://uk.saint-gobain-glass.com/trade-customers/glass-and-solar-radiation
12. Tseng, C.-Y.; Yan, Y.-A.; and Leong, J.C. (2014). Thermal accumulation in a general car cabin model. *Journal of Fluid Flow, Heat and Mass Transfer*, 1, 48-56.
13. Raić, B.; and Radovan, A. (2014). Microcontroller managed module for automatic ventilation of vehicle interior. *2014 37th International Convention on Information and Communication Technology, Electronics and Microelectronics* (*MIPRO*), Opatija, Croatia.