Ventilation of the Parked Automobile under Blazing Sun

Head of the Research Project: Prof. OKUI, K.
Report submitted by KHAN, M.U., Venture Business Laboratory, Toyama University

1. Introduction

When passengers enter a vehicle parked in the sun for an extended period requires thermal comfort. At the same time sensible car components also required limited thermal level to perform properly. To limit the car cabin temperature several researchers proposes different techniques. Their research are divided into two classification one is to protect the car from the solar radiation by different methods and another one is giving proper ventilation into the car cabin. Advanced window glazing and thermal insulations are the methods to restrict solar radiation into the car cabin which in turns helps to limit the car cabin temperature. But these two methods are bit expensive and it also diminishes exposure of the car body. Second type of solution includes ambient cooling and venting system and fan system driven by solar battery for ventilation. In the long run these two are not suitable with respect to financial and technical point of view.

At this stage, this research intended to invent a ventilation system which will be financially and technically suitable. To achieve this goal temperature of the car cabin, suitable air flow rate was measured and location of the air inlet was searched experimentally and numerically. After getting suitable air flow rate and location of the air inlet, it will make the path to design the ventilations system. This paper will describe temperature measurement and optimization of air flow rate to limit the car cabin temperature within comfortable range.

2. Experiment

Outline of the experimental set-up for temperature measurement was shown in figure 1. This figure shows the top view of the experimental set-up. Temperatures were measured in the different locations of the car cabin and shown by the black and white circle. T-type thermocouple was used for this purpose. Five analog modules were used for data acquisition. Sampling rate of the data acquisition was 1 samples/min. Accuracy of the modules was $\pm 0.1\%$. This analog data was converted into digital data by analog/digital converter. Digitized data was analyzed by the computer.

Atmospheric air was used for ventilation. Two different locations for air inlets were used one was on the top of the front panel and other one was used at the bottom of the front

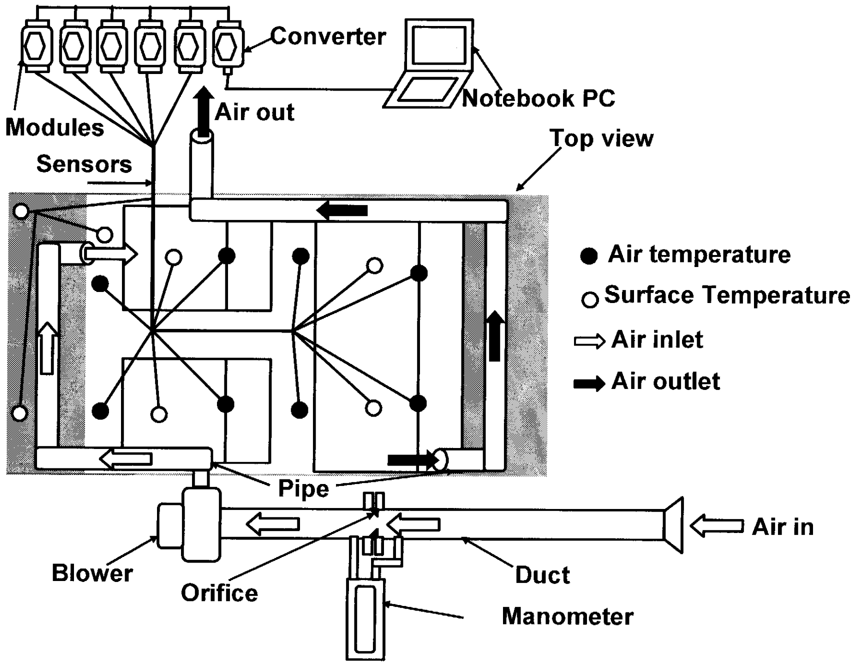

Fig. 1 Outline of the experimental set-up

panel which was inclined at 35° with car floor. At the same time, the location of the vent for air outlet was shown in figure 1. Before entering into the car cabin, air flow rate was measured by the combination of orifice and manometer. Different rate of air flow were passed into the car cabin e.g. 50, 75, 100, 125, 150, 200m^3/h at starting of the experiment. For this purpose, orifice was calibrated before experiment.

3. Experimental results

3.1 Effect of Solar radiation without ventilation

Figure 2 shows temperature of the top of the front panel and the air temperature near the drivers head without ventilation. Maximum temperature of the car cabin was found on the front panel and it was 83℃. Air space temperature near driver's head was 67℃.

3.2 Comparison of effect of different air flow rate on temperature mitigation

Influence of the airflow rate was shown in figure 3. Temperature variation near the drivers head was shown in this figure. These implied that airflow rate for 50 and 75m^3/h can suppress the temperature below 55℃. Airflow rates 100, 125, 150 and 200m^3/h can suppress the temperature below 50℃ and these flow rates show almost same behavior of temperature suppression.

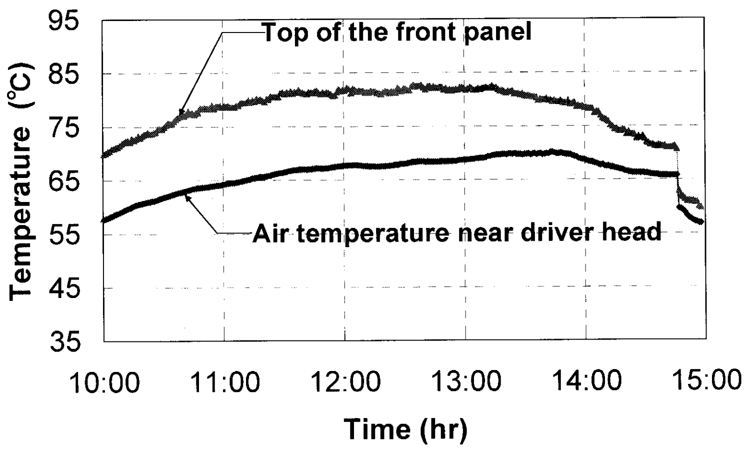

Fig. 2 Temperature variation with time at different locations (without cooling)

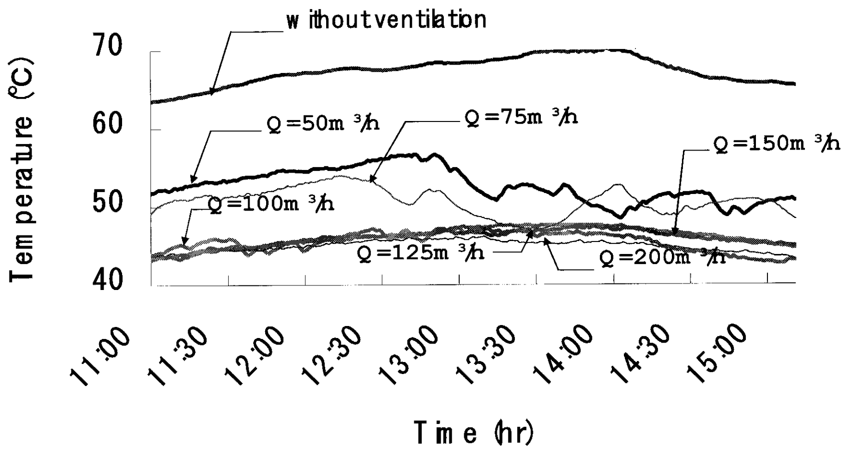

Fig. 3 Temperature variation with time at different flow rates (m^3/h)

3.3 Comparison of different location of air inlet on ventilation

Figure 4 shows the comparison between the effects of air inlet locations on the temperature suppression. From figure 3 it is evident that from 11:30am to 12:30pm temperature fluctuation was not severe as it was on the later stage. For this reason, to calculate the temperature mitigation, at first average temperature of this time was taken from the data of without ventilation. Then it was subtracted from the average temperature at different flow rate. This temperature difference or amount of the temperature mitigation was also calculated for both air inlet locations. Air at below front panel shows more temperature mitigation then air inlet at above front panel.

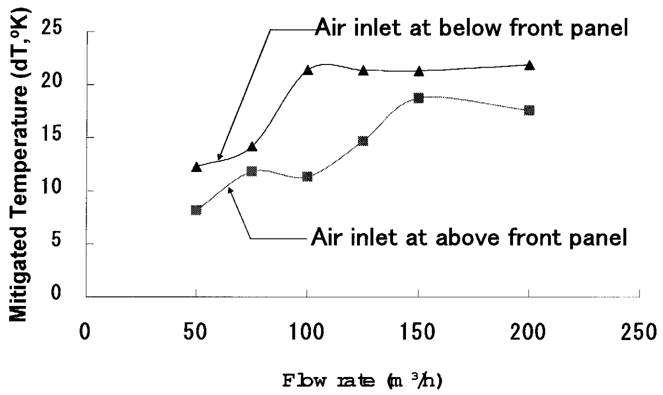

Fig. 4 Temperature mitigation with different flow rates at different locations of air inlets

4. Numerical simulation

4.1 Simulation technique

Three dimensional Naviers-Stokes equations were used for Numerical simulation. Finite volume method was used as simulation technique. Multi-disciplinary CFD and heat transfer software CFD2000 was used for numerical simulation. Boundary fitted co-ordinate system was applied to build the car model which dimensions were same as the car used in the experiment. Outline of the simulation model was shown in figure 5. Mesh numbers were 37x25x31. Heat was radiated from the roof, windows and seats. Location of the inlet and

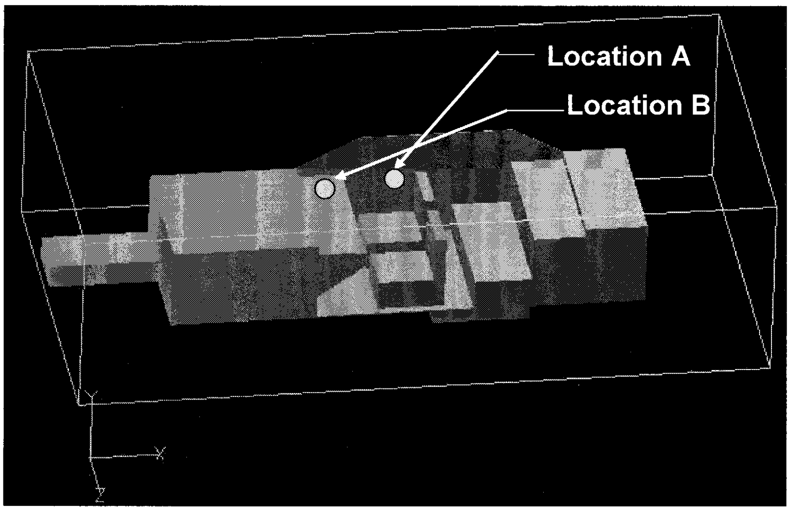

Fig. 5 Outline of the simulation model and locations of observation points

outlet of the air flow were same as the experiment but air inlet at the bottom of the front panel was parallel with the floor instead of inclined air inlet. Constant velocity and temperature air was passed through outside of the car. Simulation was carried out for 50, 100, 150 and 200m^3/h.

4.2 Numerical results and discussions

Temperature distribution at different locations of the car cabin was shown by the Figure 6. Temperature on the front panel was found 81℃ and it was maximum temperature in the car cabin for without ventilation. Air temperature near the drivers head was 65℃. Figure 7 shows the influence of the different air flow rates on the temperature mitigation. Ambient air was passed inside the car cabin at 50, 100, 150 and 200m^3/h flow rates.

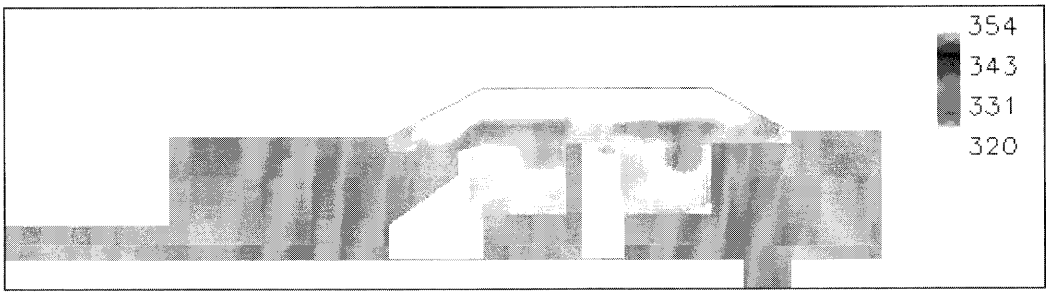

Fig. 6 Temperature distribution in the car cabin without cooling
(Section X-Y of figure 5)

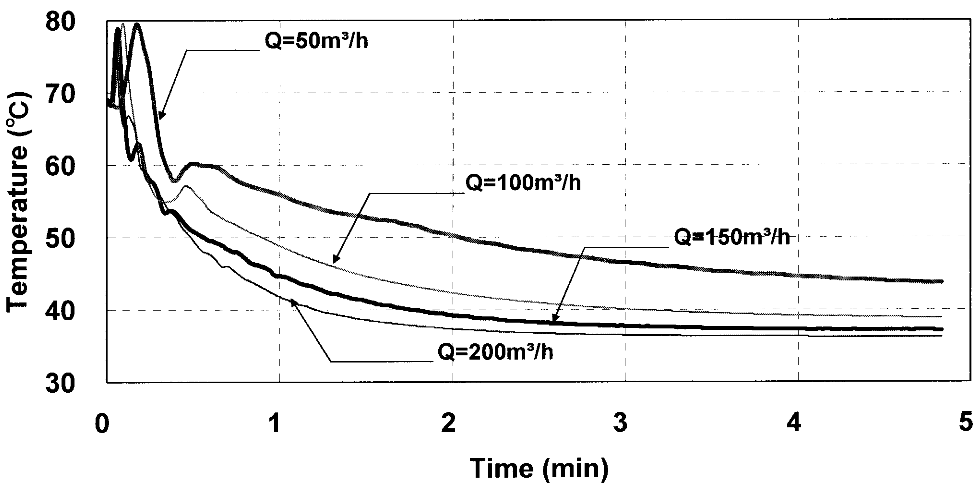

Fig. 7 Temperature distribution with time near drivers head

Temperature decreases below 40°C for 100, 150 and 200 m^3/h, which is 5°K lower than the experiment. When the flow rate was 50 m^3/h differences between experiment and numerical result was 10°K. In practical situation car cabin is always leaking some amount of ventilated air. However for numerical simulation there was no such loss in ventilation. That's why numerical simulation shows better temperature mitigation then experiment.

5. Concluding remarks

Car cabin temperature was measured experimentally and numerically. Maximum temperature was found on the front panel and it was 83℃. Suitable location for the air inlet was found at the bottom of the front panel which was inclined with floor of the car at 35°. For air flow rates 100, 125, 150, 200m m^3/h showed same amount of temperature mitigation when air inlet was at bottom of the front panel and these shows superior temperature mitigation then the air inlet at the top of the front panel. Numerical results also shows almost similar pattern of the experimental result. From these experiment and numerical result it was concluded that inclined air inlet at the bottom of the front panel and air flow rate 100m^3/h is suitable for optimum ventilation system for comfortable car cabin.

6. Future plan

Temperature measurement of the car cabin, suitable air flow rate and location of the ventilation system were searched. At the same time, solar powered fan system development was carrying out. After developing this system, performance test will be held out in this year. Ventilation system will be evaluated by installing it in the car which was used for temperature measurement.

Reference

(1) Frank K., Principles of Heat Transfer, Third ed. 1976