Development of Ventilation Technique to Limit the Car Cabin Temperature Under a Blazing Sun

Head of the Research Project: Prof. OKUI, K.

Report submitted by KHAN, M.U., Venture Business Laboratory, University of Toyama

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. Higher air condition power is necessary to cope with this situation.

At this stage, this research intended to invent a ventilation system which will be financially and technically suitable. Two step objectives were set to achieve this goal. First step, we are going to measure required ventilation. For this purpose, temperature of the car cabin, necessary air flow rate and suitable ventilation location were investigated. Numerical simulation was also performed for better understanding of the problem.

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.

Two types of ventilation methods were used. One was suction type and another one was discharge type ventilation. Figure 2a shows schematic diagram of the experimental set-up for suction type ventilation. Atmospheric air was sucked by a blower to pass it into the car cabin. Before entering into the car cabin air flow rate was measured by the Manometer/Orifice combination. Two air inlets were used, one was setup on the top of the front panel and another one was set-up below the front panel. Air outlet was set-up on the passenger side rear panel for both cases.

Figure 2b shows schematic diagram of the experimental set-up for discharge type ventilation. Atmospheric air was enters into the car air inlet and discharge it by blower. Before discharging from the blower air flow rate was measured by orifice manometer combination. Air inlet was set-up on the top of the front panel and air outlet was set-up on

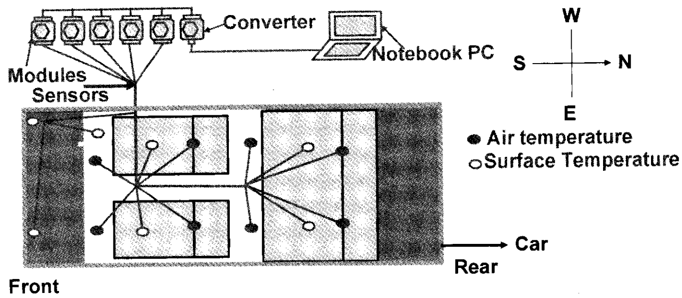
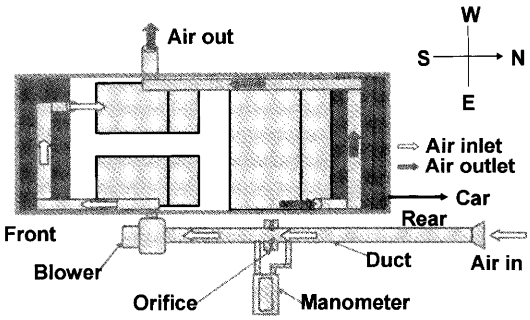
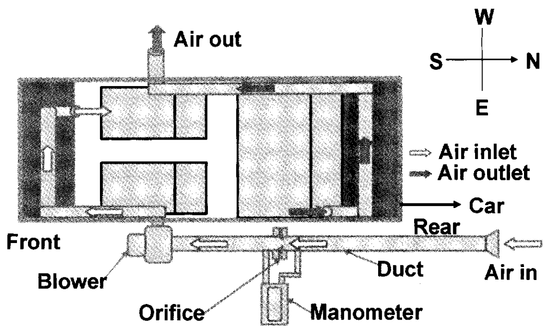


Fig. 1 Schematic diagram of experimental setup for temperature measurement



(a) Suction type ventilation



(b) Discharge type ventilation

Fig. 2 Schematic diagram of experimental setup for flow rate measurement

the passenger side rear panel. 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 Temperature variation of car cabin temperature without ventilation

Figure 3 shows temperature variation of the front panel surface and air space temperature near drivers head without ventilation. On this day average atmospheric temperature was 32℃ during 10:00am to 03:00pm. Maximum temperature of the car cabin was found on the front panel surface and it was 83℃. Air space temperature near driver head was 67℃.

3.2 Effect of different air flow rate on temperature mitigation

Influence of the airflow rate was shown in figure 4. Temperature variation near the drivers head was shown in this figure. This figure shows, airflow rate 50 and 75m^3/h can suppress temperature around 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

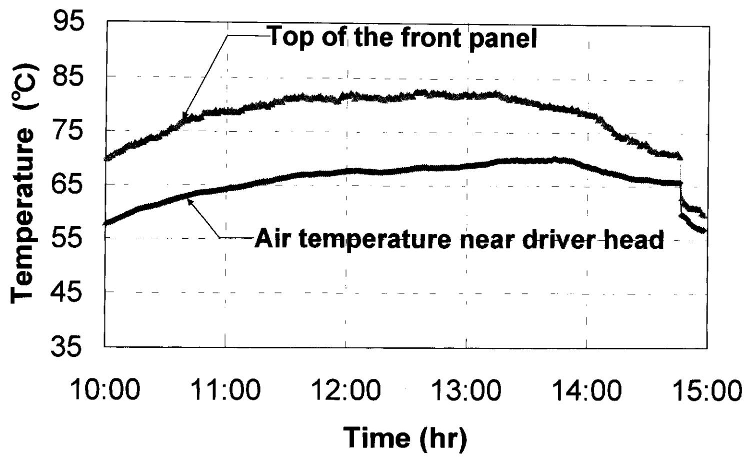


Fig. 3 Temperature variation with time at different locations (without cooling)

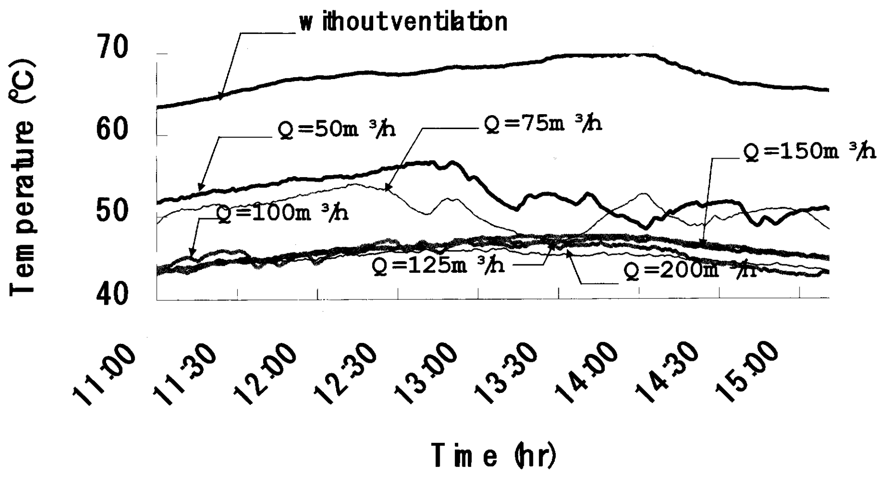


Fig. 4 Temperature variation with time at different flow rates (m^3/h)

temperature

suppression. From this figure, it may suggested that air flow rates $100m^3/h$ will be sufficient to mitigate temperature within comfortable range.

3.3 Effect of different location of air inlet and ventilation methods on ventilation

Figure 5 shows temperature mitigation at different air inlet locations and at different ventilation methods of air space temperature near drivers head. 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

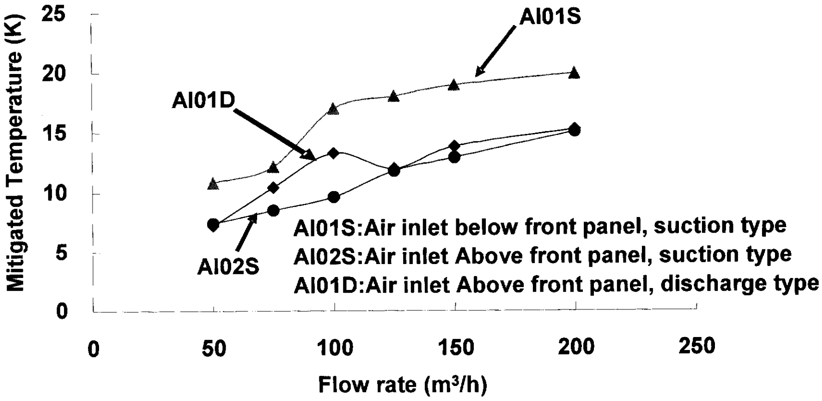


Fig.5 Effect of different air inlet locations and different ventilation methods on temperature mitigation

calculated for both air inlet locations. When air inlet was below front panel and ventilation method was suction type, mitigated temperature was 12K for air flow rate 50 and 75 m^3/hr. However for 100, 125, 150 and 200m^3/h mitigated temperature was near about 20K. When air inlet was above front panel and ventilation method was suction type, mitigated temperature was below 10K for air flow rate 50 and 75 m3/hr. For 100, 125, 150 and 200m^3/h mitigated temperature was above 10K. When air inlet was above front panel and ventilation method was discharge type, mitigated temperature were 7K and 10K for air flow rate 50 and 75 m3/hr respectively. For 100, 125, 150 and 200m^3/h mitigated temperature was near about 15K. From these analysis air inlet at below front panel shows better temperature mitigation than air inlet at above front panel. Suction type ventilation method shows better temperature mitigation than discharge type ventilation.

4. Numerical simulation

4.1 Simulation techniques

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 6. Mesh numbers were 37x25x31. Heat was radiated from the roof, windows and seats. Location of the inlet and 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 Simulation results and discussions

Figure 7 shows temperature distributions inside the car cabin without ventilation. Maximum temperature of the car cabin found around the top of the front panel surface and it was 81^{0}C and air space temperature near drivers head was 66^{0}C. This results show almost similar pattern of experimental result.

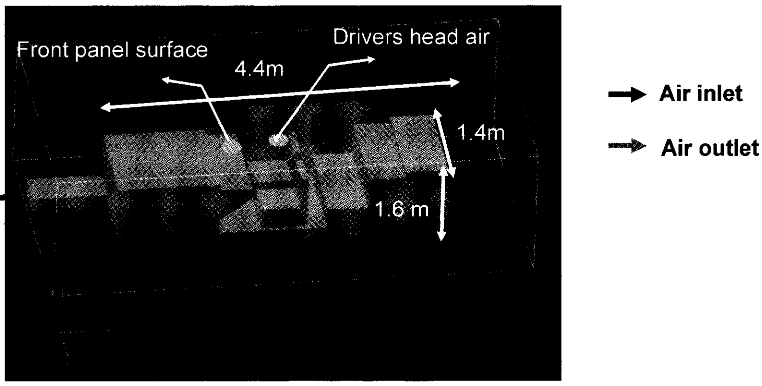


Fig. 6 Part of the simulation model

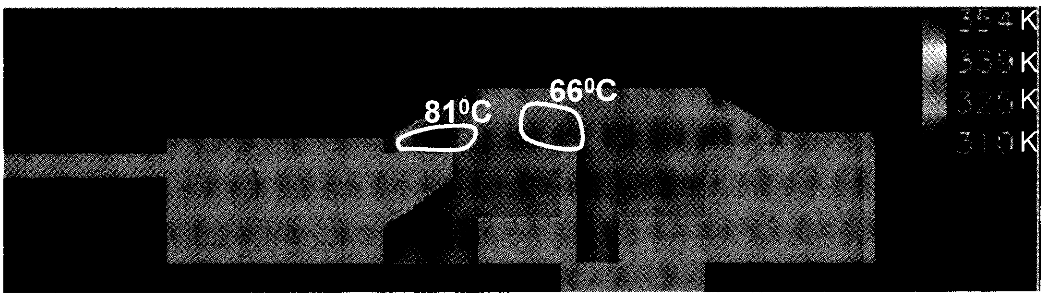


Fig. 7 Temperature distribution in the car cabin without ventilation (Section X-Y)

5. Concluding remarks

Car cabin temperature was measured experimentally and model steam turbine was designed and tested. Air space temperature near driver's head was found 67^{0}C. Air flow rate should be around 100m^3/h for desired temperature mitigation. Air inlet at below front panel shows better temperature mitigation than air inlet at above front panel.

Reference

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

(2) KHAN, M.U., KAWAGUCHI, K., OKUI, OHBA, H.,"Temperature Mitigation of the Parked Automobile under a Blazing Sun", 41st Heat Transfer Symposium.