Journal of Engineering Science and Technology
Vol. 7, No. 5 (2012) 589 - 600

THE EFFECT OF SOLAR RADIATION ON AUTOMOBILE ENVIRONMENT THROUGH NATURAL CONVECTION AND MIXED CONVECTION

MD. FAISAL KADER[1], MOHAMMAD ALI JINNAH[1,]*, KUM-BAE LEE[2]

[1]MCE Department, Islamic University of Technology, Gazipur, Bangladesh
[2]Division of Mechanical & Automotive Engg, Kongju National University, Korea
*Corresponding Author: jinnah@iut-dhaka.edu

Abstract

In the present paper, the effect of solar radiation on automobiles has been studied by both experimentally and numerically. The numerical solution is done by an operation friendly and fast CFD code – SC/Tetra with a full scale model of a SM3 car and turbulence is modeled by the standard k-ε equation. Numerical analysis of the three-dimensional model predicts a detailed description of fluid flow and temperature distribution in the passenger compartment during both the natural convection due to the incoming solar radiation and mixed convection due to the flow from defrost nozzle and radiation. It can be seen that solar radiation is an important parameter to raise the compartment temperature above the ambient temperature during summer. During natural convection, the rate of heat transfer is fast at the initial period. In the mixed convection analyses, it is found that the temperature drops down to a comfortable range almost linearly at the initial stage. Experimental investigations are performed to determine the temperature contour on the windshield and the local temperature at a particular point for further validation of the numerical results.

Keywords: CFD, HVAC, Insolation, Defroster nozzle, Automobile environment.

1. Introduction

The substantial advancement in the field of Computational Fluid Dynamics (CFD) encouraged a number of researchers to investigate the various studies in the field of automobile Heating, Ventilating and Air-Conditioning (HVAC) systems. Due to the raising mobility, people spend longer time in vehicles and thermal comfort becomes an important selection criterion during the acquisition

Nomenclatures

c_p	Specific heat at constant pressure, J/kgK
F_{ij}	View factor
g	Acceleration due to gravity, m/s^2
I_i	Incident radiant energy of the surface i
I_s	Incident insolation energy received by surface i
i	i-Surface
J_i	Radiosity, W/m^2
j	j-Surface
K	Thermal conductivity, W/mK
k	Turbulent energy, m^2/s^2
N_i	Number of particles generated at surface i
N_{ij}	Number of particles reached surface j among N_i
p	Pressure, Pa
$\dot{q}$	Heat source, J/m^{3}s
t	Time, s
T	Temperature, K
T_{exp}	Experimental temperature, K
T_{num}	Numerical temperature, K
u	Velocity, m/s
x	Coordinates
Greek Symbols	
α	Insolation absorptance of surface i
α_s	Insolation absorptance of surface i
β	Volumetric expansion coefficient, 1/K
σ	Stress tensor
ε	Turbulent dissipation rate, m^2/s^3
μ	Dynamic viscosity, kg/ms
μ_l	Laminar coefficient of viscosity, kg/ms
μ_t	Turbulent coefficient of viscosity, kg/ms
ρ	Density, kg/m^3
ρ_s	Insolation reflectance of surface i

of a car. The thermal comfort inside an automobile compartment depends on the physical variables that characterize the surroundings such as temperature, air velocity, acceleration and light intensity. Extensive researches on vehicle systems and components could be able to meet up the specific customer requirements. Nevertheless, a design engineer could predict a system performance and optimize its objectives cost-effectively by utilizing CFD tools with fewer experiments**.**

Earlier researchers stressed to improve the HVAC design for better automobile environment. Dugand and Vitali [1] carried out an experimental investigation using Thermo-graphic technique to detect thermal fields on emitting surfaces. The authors proposed a specific combination of hardware/software for the processing of the images obtained and recommended various ways of improving windshield/defrosting systems. Chakroun and Fahed [2] addressed the behaviour of the air temperature inside a car parked in the sun with different covering arrangements by using indoor climate analyzer and such a technique was found to be very effective in reducing the inside air temperature of a car parked in the sun. The effects of the covering arrangements of the car and the solar fan on the initial cooling rate of an air-conditioning system during normal driving conditions were also analyzed. Abdul Nour [3] conducted a study using the STAR-CD code. He examined the windshield flow fields and the vehicle defroster system under various operating conditions. The comparison between the hot-wire velocity measurements and the numerical predictions showed good agreement for various defroster and windshield flows.

Numerical simulations of a two-dimensional and a three-dimensional airflow in a passenger compartment were performed by Hara et al. [4] and these studies mainly concentrated on validations of HVAC systems. Also, the effect of four HVAC design parameters on passenger thermal comfort was analyzed in a simplified passenger compartment using CFD by Lin et al. [5]. They found that the location of the vents and the air flow rate were the most important parameters which influenced the thermal comfort of the passengers. The position of the outlet in the rear of the car was equally important for the thermal comfort of rear passenger. De Silva et al. [6] critiqued differing aerodynamic sidepod designs and their effect upon radiator heat management. Various features from inlet size, sidepod shape and size, presence of an undertray, suspension cover, gills and chimneys are analysed for their effects. Computational Fluid Dynamics (CFD) analyses are performed in the FLUENT environment, with the aid of GAMBIT meshing software and SolidWorks modelling.

Since the airflow inside the passenger compartment is complex and three-dimensional in nature, flow measurements have generally been made using scaled models of compartments. A study by Ishihara et al. [7] examined the airflow inside a one-fourth scale three-dimensional model. They used the Particle Tracking Velocimetry (PTV) technique to qualify and quantify the flow inside the model. The study concentrated on the effects of altering the design of the dashboard or instrument panel (IP) vents. The experimental and calculated results showed good qualitative agreement with respect to the flow patterns and vortex distributions in the rear compartment. A notable study by Aroussi and Aghil [8] used a one-fifth scaled Perspex model of a passenger compartment for experiment by Particle Image Velocimetry (PIV) technique that validated the CFD results. A further study by Aroussi et al. [9] concentrated to simulate the turbulent fluid flow over and heat transfer through a model of a vehicle windshield defrosting and demisting system. The numerical results of the study were very encouraging and compare favourably with measurements obtained from the actual vehicle by Thermograph and Hot Bulb probe techniques. The findings highlighted some of the drawbacks of the existing design of the windshield systems and showed that the maximum flow rates occurred in the vicinity of the lower part of the windshield, progressing from the defroster nozzle in the dashboard.

Kader et al. [10] focused on temperature distribution characteristics of an automobile interior both numerically and experimentally when operating the HVAC system in the summer and the defrosting performance was analyzed by changing the injection angle and the distance from the windshield. Defrosting performance was excellent when the injection angle of the defrost nozzle was between 15 to 25 degrees with a standard distance ratio of one. During the cooling period, it was found that the temperature and humidity decreased to a comfortable range almost linearly at the initial stage. The numerical predictions are in good agreement with the experimental results. From the above review, it can be stated that some achievement has been accomplished in understanding airflow characteristics inside an automobile compartment, but still there is a need for further scrutiny. In summer, the comfort in an automobile strongly depends on the exchanges of thermal radiation between the automobile compartment and the environment because radiation is undoubtedly the most instantaneous phenomena of energy transfer.

In the present study, the flow field and temperature distribution within a 3D full scale model of an automobile compartment and on the windshield were investigated using CFD to determine the capability of the method. The influences of discharge from the defroster nozzle on the automobile internal aerodynamics and radiation have been assessed.

2. Experimental Setup

The present experiment was performed on a 2006 SM3 model vehicle of Samsung Automobile Company with a 1,500 cc diesel engine, presented in Fig. 1. The automobile was instrumented with sensor at the driver's feet and face in the compartment to measure the air temperature with respect to time. The equipment used for measuring temperature is shown in Fig. 2 whilst the measurement locations are presented in Fig. 2. The measurement was performed under hot sunny summer conditions (ambient temperature was 35°C dry bulb). The fully opened defroster nozzles were used to supply the flow at a velocity of 8 m/s (normal to the surface). The measuring time was 30 minutes at noon in the month of September.

Fig. 1. The Automobile Used in Experiment.

Fig. 2. Experimental Apparatus for Measuring Temperature and Velocity.

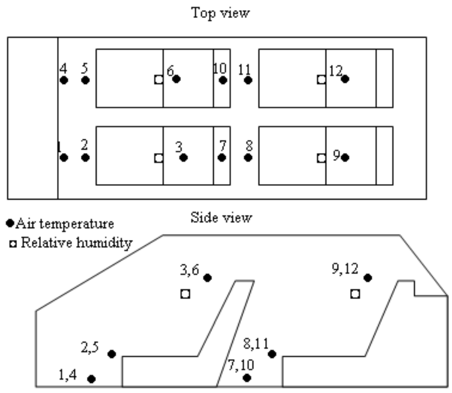


Fig. 3. Automobile Interior Showing Measurement Locations and Vertical (V1) & Horizontal (H1 and H2) Planes.

3. Numerical Investigation

The simplified three dimensional geometry of an automobile compartment is generated in Computer-graphics Aided Three-dimensional Interactive Application (CATIA). The simplified geometry represents the actual dimensions of a standard car compartment with accurate locations of the nozzles and vents including the seating arrangements. The numerical code used in this study is the finite volume CFD code Scryu Tetra (SC/T) version 7 [11]. The software has three main parts namely pre-processor, solver and post-processor and abbreviated as SC/Tpre, SC/Tsolver and SC/Tpost respectively. The model is imported to SC/Tpre and unstructured computational meshes of fluid cells (653251 tetrahedra) are generated. The numerical model and the computational mesh are shown in Fig. 4. The de-froster nozzles, windshield and side windows are meshed closely to increase the accuracy of the numerical result. The flow inside the compartment is unsteady and incompressible turbulent flow. The turbulence is modeled by using the standard *k-ε* turbulence equations. The numerical solution is done in SC/Tsolver.

3.1. Governing equations

The fluid inside vehicle compartment is assumed as an incompressible fluid. The governing equations solving the numerical simulation are [11]

Mass equation: $$\frac{\partial \rho u_i}{\partial x_i} = 0 \qquad (1)$$

Momentum equation:

$$\frac{\partial \rho u_i}{\partial t} + \frac{\partial u_j \rho u_i}{\partial x_j} = -\frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j}\mu\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i}\right) - \rho g_i \beta (T - T_0) \qquad (2)$$

where, $\mu = \mu_l + \mu_t$ $\qquad \mu_t = C_t \rho \frac{k^2}{\varepsilon}$

Energy equation: $$\frac{\partial \rho c_p T}{\partial t} + \frac{\partial u_i \rho c_p T}{\partial x_i} = \frac{\partial}{\partial x_i} K \frac{\partial T}{\partial x_i} + \dot{q} \qquad (3)$$

k-ε equation: $$\frac{\partial \rho k}{\partial t} + \frac{\partial u_i \rho k}{\partial x_i} = \frac{\partial}{\partial x_i}\left(\frac{\mu_t}{\sigma_k}\frac{\partial k}{\partial x_i}\right) + G_S + G_T - \rho\varepsilon \qquad (4)$$

$$\frac{\partial \rho \varepsilon}{\partial t} + \frac{\partial u_i \rho \varepsilon}{\partial x_i} = \frac{\partial}{\partial x_i}\left(\frac{\mu_t}{\sigma_\varepsilon}\frac{\partial \varepsilon}{\partial x_i}\right) + C_1 \frac{\varepsilon}{k}(G_S + G_T)(1 + C_3 R_f) - C_2 \frac{\rho \varepsilon^2}{k} \qquad (5)$$

where, $G_S = \mu_t\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i}\right)\frac{\partial u_i}{\partial x_j}$, $G_T = g_i \beta \frac{\mu_t}{\sigma_t}\frac{\partial T}{\partial x_i}$ and $R_f = -\frac{G_T}{G_s + G_T}$.

These equations include many empirical constants which are given below [11]:

$\sigma_k = 1, \sigma_\varepsilon = 1.3, \sigma_t = 0.9,\ C_t = 0.09,\ C_1 = 1.44,\ C_2 = 1.92,\ C_3 = 0.0$.

The finite volume method is used in SC/Tsolver to solve the equations listed above on a computer. A finite volume method solves the governing equations by converting them into an integral conservation form that is expressed on each fractional unit of decomposed elements, or control volume. The view factor F_{ij}

from surface *i* to surface *j*, thermal flux q_r and radiosity J_i of surface *i*, are given by the following equations:

$$F_{ij} = N_{ij} / N_i \quad (6)$$

$$q_r = \alpha_s (I_i + I_s) \quad (7)$$

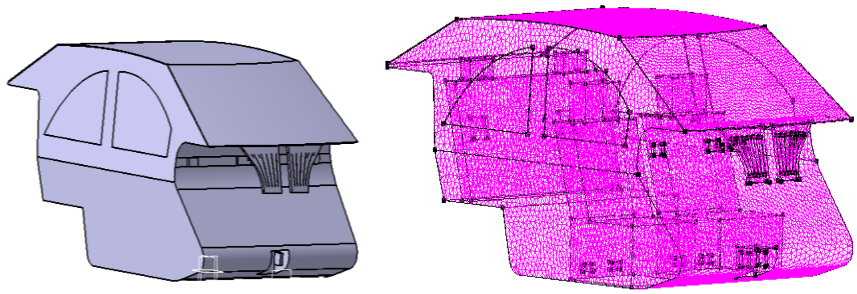
$$J_i = \rho_s (I_i + I_s) \quad (8)$$

3.2. Material properties

The material properties used for the numerical analyses have been shown in Table 1. The thermal expansion co-efficient is considered as 0.00341 for the air inside the compartment only. Car glass properties are applied to windshield, banks windshield and side glasses.

Table 1. Material Properties.

Material	ρ (kg/m^3)	μ ×10^{-5} (Pa·s)	c_p (J/kg.K)	K (W/ m·K)
Air	1.206	1.83	1007	0.0256
Car glass	2190	-	740	1.38
Car roof (iron)	7871.4	-	439.2	81.16



(a) (b)

Fig. 4. (a) The Numerical Model of an Automobile Compartment, (b) Overall Computational Mesh of the Model.

3.3. Boundary/initial conditions

The initial temperature of the compartment is 20°C for natural convection and 48°C for mixed convection analysis. For natural convection analysis no inflow and outflow conditions is selected. For mixed convection, the defroster nozzles are employed to inject the air flow at a velocity of 8 m/s. The unsteady inlet temperature vary from 41°C to 15°C as the first and final temperatures respectively over a period of 30 min. Fixed static pressure is selected for outflow boundary condition. The wall of the compartment is considered as adiabatic while radiation heat transfer is considered through the windshield and other glasses. The ambient temperature is considered as 35°C and isothermal.

4. Results and Discussion

The study concentrates on the natural convection inside an automobile compartment driven by the incoming solar radiation (insolation) and the mixed convection due to this flow from the defrost nozzle and the insolation effect.

The velocity profile is analyzed in the vertical plane (V1) with respect to initial and final period. The velocity vector crossing through the middle of the compartment is shown in Fig. 5. First of all, the movement of fluid without an external source introduces natural convection. Initially the temperature difference between the car compartment and the environment is higher, which result in the higher velocity of fluid 0.0781 m/s at the initial period and that drops to 0.0354 m/s over a period of 30 min. To illustrate the lateral fluid motion within the compartment, the horizontal planes H1 and H2, crossing through the occupant's neck and feet are presented in Figs. 6 and 7 respectively. It is evident that windshield is responsible for major radiation. The flow originates from the windshield and front side glasses and then moves forward. The obstructions cause by the front seats and the rear seats generated several lateral re-circulation zones in the front and rear passenger compartments. These eddies are driven by the longitudinal flow between the seats and in the gaps adjacent to the compartment doors.

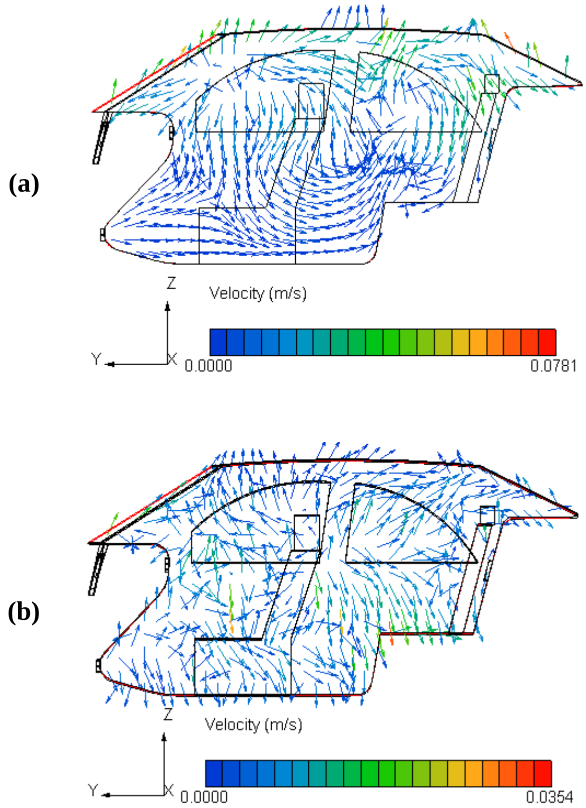


Fig. 5. Velocity Vectors Across the Plane V1 (a) Initial and (b) Final in the Vertical Plane Crossing Through the Middle of the Compartment.

(a)

(b)

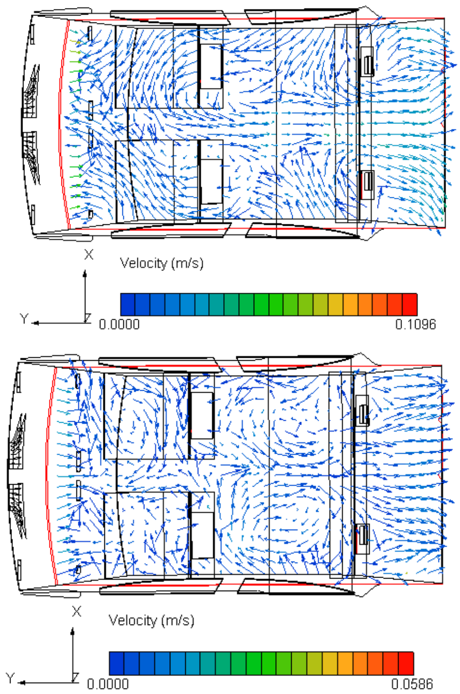


Fig. 6. Velocity Vectors Across the Plane H1 (a) Initial and (b) Final.

(a)

(b)

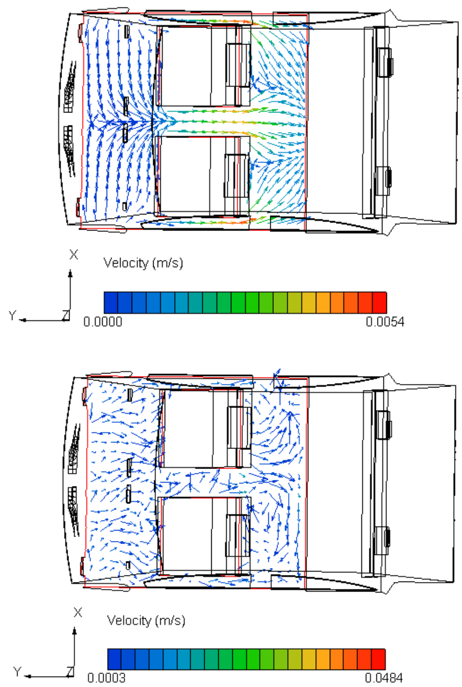


Fig. 7. Velocity Vectors Across the Plane H2 (a) Initial and (b) Final.

The magnitude of velocity at the aforesaid planes is shown in Fig. 8, it can be seen that initially the velocity of fluid with higher magnitude exists in the upper part of the compartment, which is dropped down upto 500 s and then stabilized. On the other hand, initially a very low magnitude of velocity has been observed in the lower part of the compartment, which is increased up to 700 s and then stabilized. That is due to the higher rate of heat transfer by insolation to the upper part of the compartment through windows at the initial period. Then the heat transfer takes place from upper part to the lower part of the compartment through natural convection.

The temperature at eight different positions inside the compartment has been presented in Fig. 9. The upper part of the compartment experiences quick temperature rise from 20°C to 32°C within 500 s, whereas lower part of the compartment remains almost same upto 400 s because the heat transfer rate increases initially by the insolation to the upper part of the compartment through windows and the roof of the vehicle achieves higher temperature and the radiation through the windshield and other glasses. After that lower part temperature increases to 26°C over a period of 30 min because the heat transfer takes place from upper part to the lower part of the compartment through natural convection.

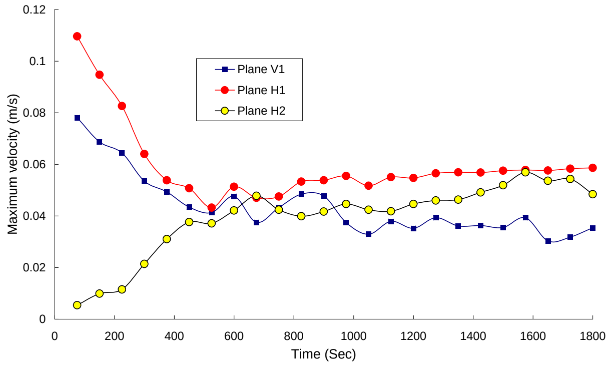


Fig. 8. Numerical Predictions of Velocities (Maximum) in the Planes V1, H1 and H2.

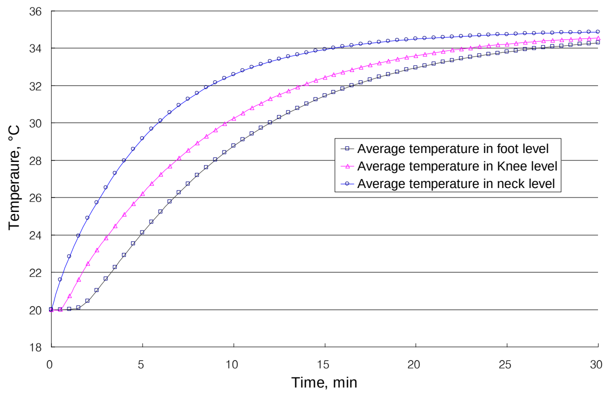


Fig. 9. Numerical Temperature Distributions at Different Positions Shown in Fig. 2.

The numerical and experimental temperature distributions at positions 1 and 5 have been presented in Figs. 10 and 11 respectively, while Fig. 12 represents the deviation analyses for positions 1 and 5. The experimental measurements show that the inlet temperature varies from 41°C to 15°C as the first and final temperature respectively over a period of 30 min and the same inlet temperature is applied to the numerical analyses and the random selection of these temperatures depends on the local temperature at any place. According to the experimental results, it is found that the temperature lowers down to the standard almost linearly from 48°C to 25°C within 12 min. After that, slight decrease has been noticed up to 30 min. A very good numerical result is found with radiation heat transfer through the windshield and other glass surfaces because the convective heat transfer highly depends on the windshield and other glass surfaces. One important observation is that, an almost constant temperature difference of 2°C is observed between the numerical results in the upper part of the compartment, whereas this temperature difference is less than 0.5°C, which is the effect of strong radiation heat transfer through the windshield and other side glasses those belongs to the upper part of the compartment. In addition, it must be mentioned that measurements are taken in a parked automobile in the absence of passengers. The automobile in motion or an automobile with passengers may affect the measurements. A good agreement has been found between the numerical predictions and experimental results. The deviation $[(T_{exp}-T_{num})/T_{exp}]$ lies between 3% for point 1 and 10% for point 5, the top of the compartment achieves the equilibrium state faster than any other places due to proper heat convection phenomena. At point 5, recirculation zones are formed due to the obstruction caused by the seat. It takes longer time to attain stable conditions due to some disturbance of the heat transfer.

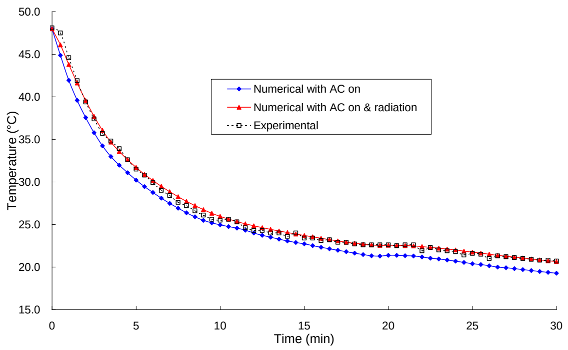


Fig. 10. Numerical and Experimental Temperature Distributions at Position 1.

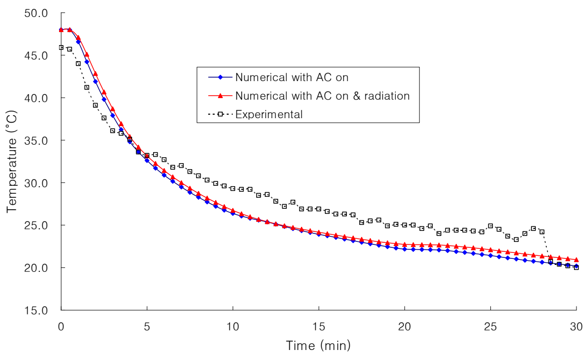


Fig. 11. Numerical and Experimental Temperature Distributions at Position 5.

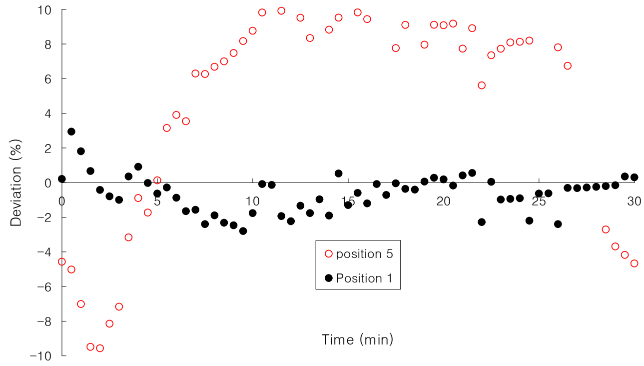


Fig. 12. Deviation Analyses between the Numerical and Experimental Temperature Distributions.

5. Conclusions

The present study has shown that CFD code can be successfully used to investigate the fluid flow pattern and temperature distribution inside the vehicle compartment. The behaviour of fluid flow and heat transfer characteristics within a vehicle compartment is very important for controlling the effect of major design parameters. Comparison with the experimental results obtained by temperature measurements shows that the CFD code can be a very useful design tool for an automobile HVAC system and the interior design with different types of glass properties. The standard *k-ε* turbulence model and a properly refined grid are found acceptable for the analysis of this particular automobile HVAC system. The present experimental results can be used to validate numerical predictions and to improve the performance of automotive HVAC modules in the initial design stage. The results of the current study highlight a number of drawbacks in the existing design for the vision of the driver and the thermal comfort inside the car cabin.

References

1. Dugand, M.M.; and Vitali, D.F. (1990). Vehicle internal thermo-fluid dynamics. Experimental and Numerical Evaluation. *SAE paper No.* 9052335.
2. Chakroun, W.; and Al-Fahed, S. (1997). Thermal comfort analysis inside a car. *International Journal Energy Research*, 21(4), 327-340.
3. Abdul Nour B.S. (1998). Numerical simulation of vehicle defroster flow field. *SAE paper No.* 980285.
4. Hara, J.; Fujitani, K.; and Kuwahara, K. (1988). Computer simulation of passenger compartment airflow. *SAE Paper No.* 881749.
5. Lin, C.H.; Han, T.; and Koromilas, C.A. (1992). Effects of HVAC design parameters on passenger thermal comfort. *SAE paper No.* 920264.

6. De Silva, C.M.; Azmi, M.N.; Christie, T.; Abou-Saba, E.; and Ooi, A. (2011). Computational flow modelling of formula-SAE sidepods for optimum radiator heat management. *Journal of Engineering Science and Technology* (*JESTEC*), 6(1), 94-108.
7. Ishihara, Y.; Hara. J.; Sakamoto, H.; Kamemoto, K.; and Okamoto, H.; (1991). Determination of flow velocity distribution in a vehicle interior using a visualization and computation techniques. *SAE paper No.* 910310.
8. Aroussi, A.; and Aghil, S. (2000). Characterisation of the flow field in a passenger car model. *Optical Diagnostics in Engineering*, 4(1), 1-15.
9. Aroussi, A.; Hassan, A.; and Morsi, YS. (2003). Numerical simulation of the airflow over and heat transfer through a vehicle windshield defrosting and demisting system. *Heat and Mass Transfer*, 39, 401-405.
10. Kader, M.F.; Youn, Y.M.; Jun, Y.D.; and Lee, K.B. (2009). Characterization of the HVAC performance with defroster grillers and instrument panel registers. *International Journal of Automotive Technology*, 10(3), 305-312.
11. SC/ Tetra User's Guide, Version 7, Software Cradle Co., Ltd. September, 2007.