Journal of Visualization, Vol. 12, No. 4 (2009) 375-382

Regular Paper

Investigation on the Flow Characteristics inside an Automotive HVAC System with Varying Ventilation Mode

Kang, J. H. *[1] and Lee, S. J. *[2]

*1 Technical Research Laboratories, POSCO, Pohang 790-300, Korea.
*2 Department of Mechanical Engineering, Pohang University of Science and Technology, Pohang 790-784, Korea.
E-mail: sjlee@postech.ac.kr

Received 27 December 2008
Revised 23 March 2009

Abstract : The ventilation flow in a heating, ventilation and air conditioning (HVAC) module of a passenger car was investigated experimentally. Three different ventilation modes with varying temperature mode were tested to study the effect of ventilation mode on the velocity field inside the HVAC module. For each mode, more than 450 instantaneous velocity fields were measured using a particle image velocimetry (PIV) velocity field measurement technique. The instantaneous velocity fields were ensemble averaged to obtain the spatial distribution of mean velocity and spanwise vorticity. The present work highlights the usefulness of the PIV technique for the analysis of the flow inside an HVAC module. The experimental results can be used not only to understand and improve the ventilation flow of an HVAC module but also to validate numerical predictions.

Keywords : HVAC, PIV, ventilation mode, temperature mode, flow visualization.

1. Introduction

With the advancements in the field of automotive engineering, there have been increasing demands for optimizing the thermal comfort of occupants in a passenger compartment aside from enhancing safety and driving ability. The thermal comfort felt by passengers can be improved by satisfying the heat balance equation for the human body, maintaining proper skin temperature and perspiration rate, removing fumes and contaminants, and supplying fresh air. Therefore, accurate information on the thermal environment inside a passenger compartment is essential in estimating the thermal comfort level. To do so, detailed information on the thermal environment is a basic prerequisite. Such environment can be roughly represented by the spatial distributions of velocity, temperature, and humidity in the vehicle interior. However, it is almost impossible to obtain such information with satisfactory spatial and temporal resolution.

Most passenger vehicles have an HVAC device to control the thermal environment of the passenger compartment. In general, the HVAC system causes a complicated three-dimensional unsteady non-isothermal turbulent flow in the vehicle interior. The ventilation flow is affected by several parameters such as the location and shape of the air-vents for ventilation, the flow rate, the disposition of the passenger, and so on. To improve the performance of the HVAC system as well as thermal comfort, a detailed understanding of the air flow inside the HVAC system is required. Recent advancements in computational fluid dynamics and experimental techniques have made it easy to analyze the ventilation flow in a complicated compartment.

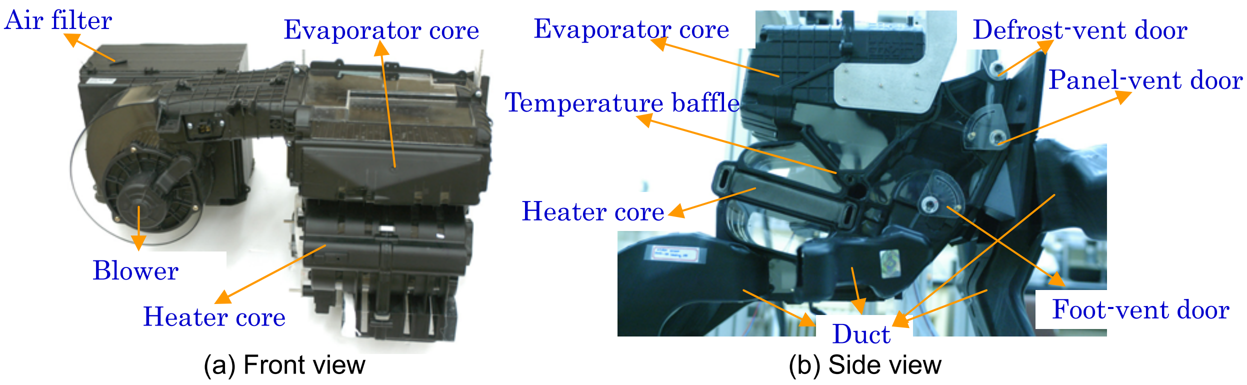
The high-speed air coming from a sirocco fan does not easily pass through the HVAC module due to the presence of temperature control baffles which curve the flow pathway and a heat exchanger. Since the air flow inside the HVAC module has three-dimensional (3D) unsteady flow characteristics, it is not easy to measure it experimentally. Hence, most previous studies on isothermal ventilation flows inside an automotive HVAC module have been performed through simple flow visualization or numerical simulations (Aroussi et al., 2001; Bennett et al., 2002; Kitada et al., 2001; Shojaee et al., 2004). Several experimental studies accessed the air flow inside the HVAC module using point-wise measurement techniques such as a hot-wire anemometer, a five hole probe, and laser doppler velocimetry (LDV). For instance, Ishihara et al. (1992) analyzed the isothermal flow in a vehicle interior of a full-scale car and 1/4-scale model employing a laser-light-sheet method. They found some differences between the local flow field of the prototype and that of the scaled-down model, but the general flow structures were in good agreement. On the other hand, Yoon and Lee (2003) investigated experimentally the ventilation flow inside a 1/10-scale vehicle model. Hirota et al. (2006) investigated turbulent air mixing in the T-junction of the HVAC system, while James et al. (2004) developed a cylindrical HVAC case.

In addition, Lin (1994) investigated the flow in a simplified HVAC duct due to the technological limitations of conventional measurement techniques. Wan and Kooi (1991) investigated numerically the effect of supply and exhaust opening vents on thermal comfort in a passenger vehicle. Currie (1997) numerically simulated the flow field and temperature distribution in a passenger compartment and evaluated the thermal comfort of occupants. To analyze some important designs, Cho and Kim (1997) numerically investigated the flow characteristics and heat transfer in an automotive HVAC module.

As far as we have surveyed, there is very limited quantitative flow information on the air ventilation inside an HVAC module. In the present study, a real automotive HVAC unit of which some parts were replaced with transparent windows was used to investigate the effects both ventilation mode and temperature baffle on the flow structure inside the HVAC module. The velocity fields of the ventilation flow inside the HVAC module were measured using a high-resolution PIV velocity field measurement technique. The principal objectives of this study are to accumulate experimental data for each ventilation mode for validating CFD predictions and to understand the flow structure in better detail. The present results would work as a basic design data to improve the HVAC system for occupants' thermal comfort.

2. Experimental Methods

The experiments were carried out in a real-size HVAC module which has been used in H motor company. Figure 1 shows the HVAC module tested in this study. As shown in Fig. 1(b), the HVAC module was made of a polycarbonate plate of high transparency for illuminating a thin laser light sheet to obtain clear flow images. The temperature and ventilation modes were controlled by using a



(a) Front view (b) Side view

Fig. 1 Photographs of the HVAC module tested in this study temperature baffle and ventilation doors, respectively. To simulate precisely the real operation conditions, the air duct system connected to the passenger compartment was attached to the end of the HVAC module. The flow rate was carefully controlled using a DC power supply (12V, 21A) and a pressure loading device. The pressure load acting on the HVAC system was adjusted by sliding a variable-area prism toward the upstream of the connecting diffuser in order to make the operation condition almost the same as the real one.

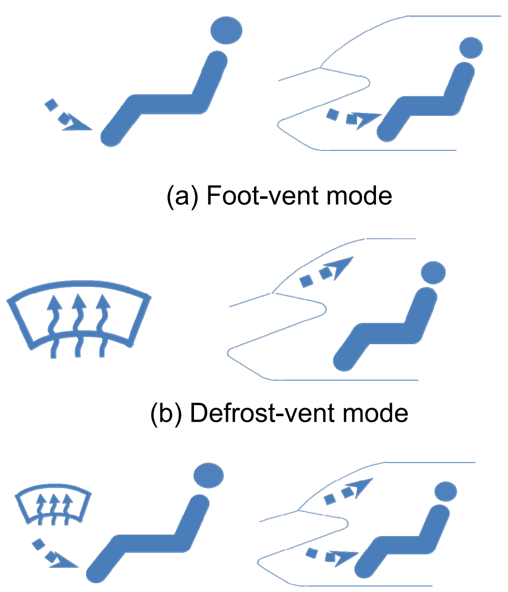
Three different ventilation modes in two temperature modes were tested under the same flow rate: the defrost-vent mode, the foot-vent mode, and the hybrid-vent mode. For the defrost-vent mode, the defrost-vent door was opened and the other doors were tightly closed. In the foot-vent mode, the foot-vent door was only opened. In the hybrid-vent mode, on the other hand, both the defrost-vent door and the foot-vent door were opened, but the panel-vent door was closed. Table 1 shows three different ventilation modes with respect to conditions of vent doors. In addition, Figures 2 and 3 show the schematic diagrams of the three different ventilation modes and pathways for two temperature modes, respectively. As shown in Fig. 3, two different temperature modes were established by varying the position of the temperature baffle.

Table 1 Ventilation mode configurations according to open/close of vent doors

	Defrost-vent door	Foot-vent door	Panel-vent door
Defrost-vent mode	Open	Close	Close
Hybrid-vent mode	Open	Open	Close
Foot-vent mode	Close	Open	Close

It is nearly impossible to measure the whole flow field inside an HVAC module with a point-wise measurement technique. In addition, conventional flow visualization methods provide only qualitative flow information. Due to the rapid advancements in computers, optics, and digital image processing techniques, the instantaneous velocity fields of a turbulent flow can be obtained using a PIV technique (Adrian, 1991).

The complicated internal structure induces a complex 3D flow inside the HVAC module. Ideally, this kind of complex flow would be analyzed using a 3D three-component velocity field measurement technique such as the holographic PIV technique. However, such 3D PTV techniques remain extraordinarily complex and expensive to apply. The two-dimensional (2D) PIV method has difficulty

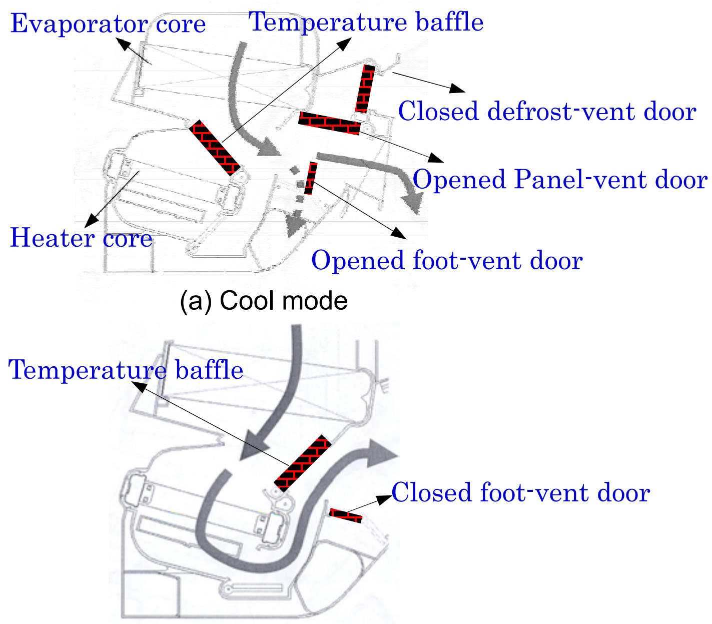


(a) Foot-vent mode

(b) Defrost-vent mode

(c) Hybrid-vent mode

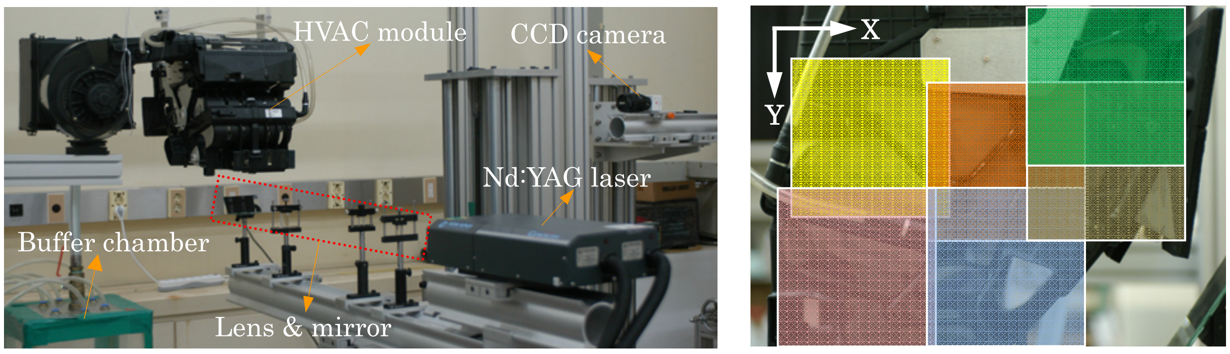
Fig. 2 Schematic diagrams of the three different ventilation modes



(a) Cool mode

(b) Warm mode

Fig. 3 Pathway for the two temperature modes tested in this study



(a) Experimental set-up (b) Measurement sections

Fig. 4 Photographs of the experimental set-up and six measurement sections for PIV

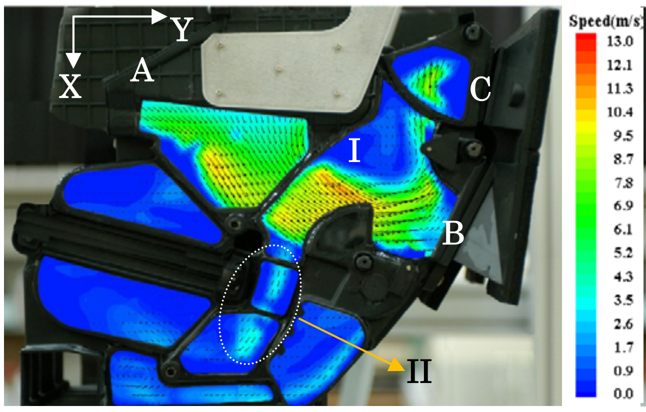
in measuring the out-of-plane velocity component. However, fortunately the lateral flow motion is relatively weak in the HVAC module as compared to in-plane velocity components. Therefore, the 2D PIV velocity field measurement system was employed in the present study to measure the in-plane velocity field data for analyzing the ventilation flow inside an HVAC module.

The PIV system consists of an Nd:YAG pulse laser, a 2K x 2K CCD camera, a delay generator, mirrors and optical lenses for illuminating light sheet, an image frame grabber, and a personal computer, as shown in Fig. 4(a). Figure 4(b) shows six measurement sections in the x-y plane for PIV. The Nd:YAG laser has two heads especially designed for PIV measurement. The maximum energy output of the laser is about 125mJ per pulse, and the pulse repetition rate is 15Hz. The short pulse width of each laser pulse (~ 7ns) allows the highly turbulent flow motions to be frozen in a clear particle image. The laser light sheet of 1mm in thickness was formed by passing the laser beam through cylindrical lenses and mirrors. It illuminated the inner measurement section of the HVAC module. The high-resolution CCD camera was positioned perpendicular to the illuminated measurement plane to capture the sequential particle images. A delay generator was used to synchronize the Nd:YAG laser and CCD camera. The time interval between two pulses, Δt, was controlled by adjusting the delay generator.

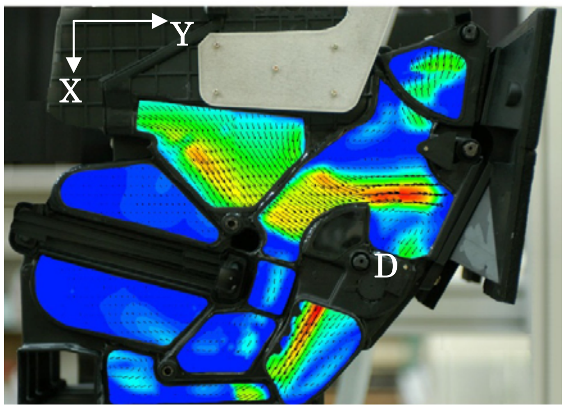
The FFT-based cross-correlation PIV technique was employed to calculate the velocity from the consecutive particle images (Jang and Lee, 2007). The size of the interrogation window was 32 x 32 pixels^2, and each interrogation window was overlapped 50%. The time interval between two consecutive images ranged from 50 to 100 μsec depending on the maximum flow speed for each ventilation mode. Small olive oil droplets of 1~3 μm in mean diameter were used as tracer particles. Compressed air was supplied to a chamber containing olive oil to generate tracer particles through a Laskin nozzle having several fine holes. More than 450 instantaneous velocity fields were obtained for each experimental condition, and they were ensemble averaged to obtain the spatial distributions of the mean velocity and vorticity.

3. Results and Discussion

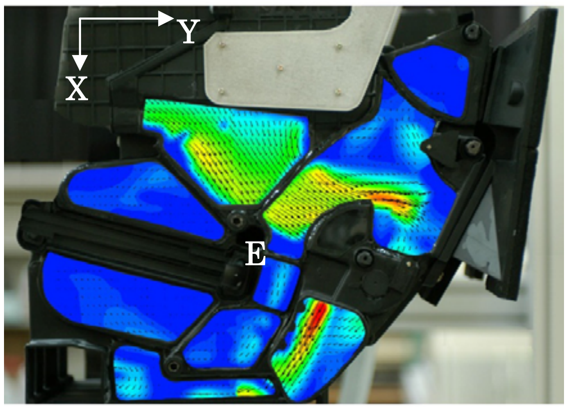
Figures 5 and 6 show the mean velocity fields and vorticity contours according to the ventilation modes at the cool mode condition. For the defrost-vent mode (Fig. 5(a)), the air flow passing through the evaporator core (A) moves directly toward the panel vent (B), then the air abruptly turn to the defrost vent (C) after bumping against the panel-vent door. Since this flow moves along the wall duct in the counterclockwise direction, stagnation flow is formed around region I. This flow phenomenon can also be seen clearly in the spanwise vorticity contours of Fig.6 (a). The strength of the counterclockwise vorticity (red color contour) is increased as the flow moves toward the panel vent. The clockwise vorticity appearing along the temperature baffle is relatively weak. In addition, some incoming air bumping against the panel-vent door goes downward in the pathway (II) connected to



(a) Defrost-vent mode

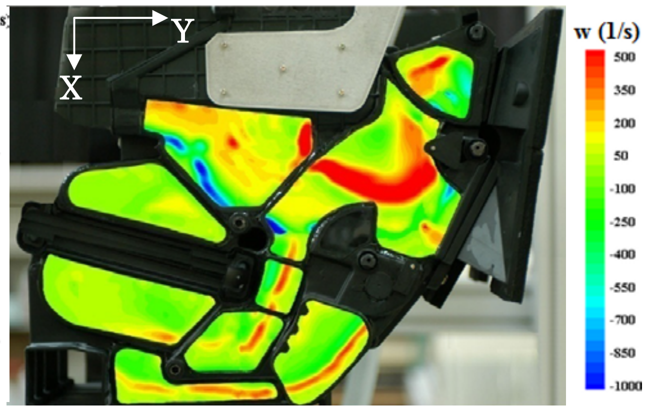


(b) Hybrid-vent mode

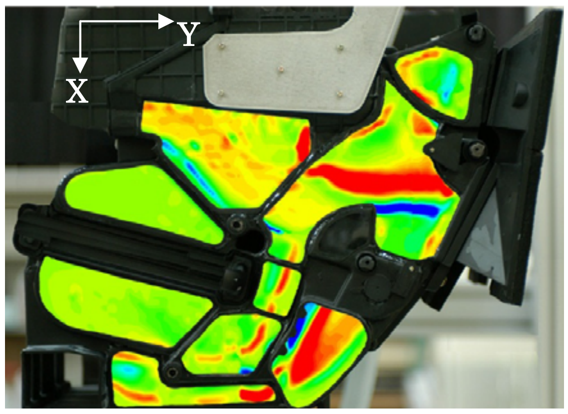


(c) Foot-vent mode

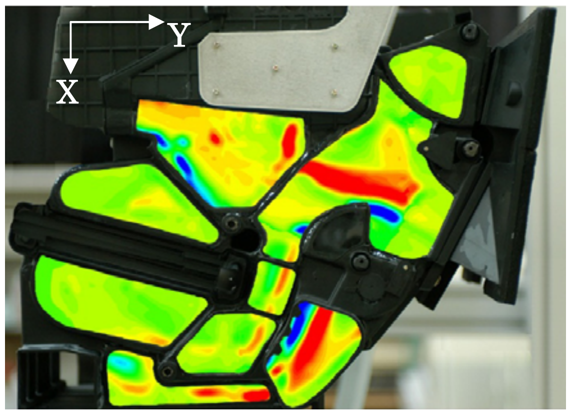
Fig. 5 Mean velocity fields for three different ventilation modes at cool mode



(a) Defrost-vent mode



(b) Hybrid-vent mode

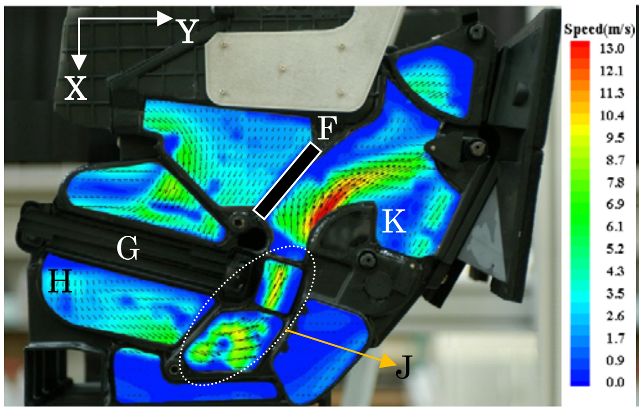


(c) Foot-vent mode

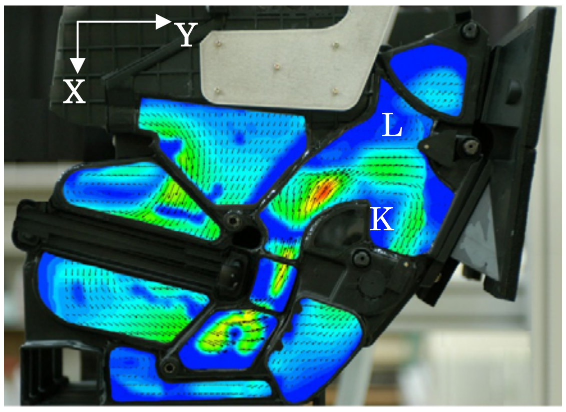
Fig. 6 Vorticity contours for three different ventilation modes at cool mode

the heater core.

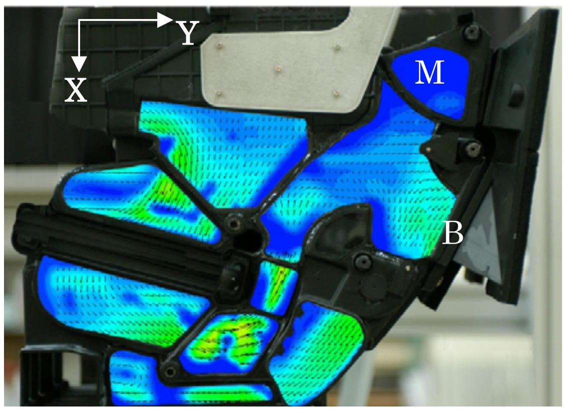
For the hybrid-vent mode (Fig. 5(b)), contrary to the defrost-vent mode, air coming toward the panel-vent door splits into two conduits connected to the opened foot vent (D) and defrost vent. The flow resistance seems to decrease a bit due to the opening of the two vent doors as compared with the defrost-vent mode. Therefore, we can see the increase in mean velocity of the incoming flow inside the HVAC module. The stagnation flow that appeared at the vicinity of the defrost vent is also decreased slightly. However, the reversal flow moving backward toward the heater core is not much improved due to the structural blockage of the curved interior ducts of the HVAC module. As shown



(a) Defrost-vent mode



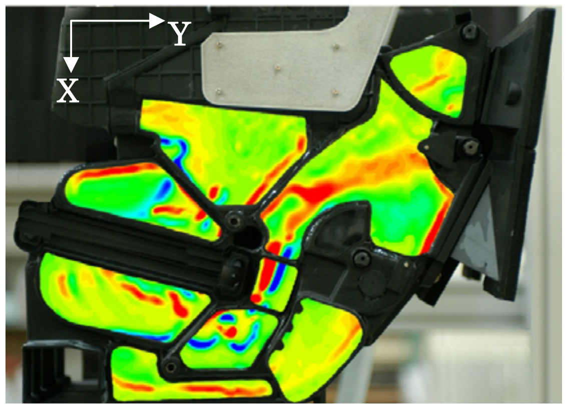
(b) Hybrid-vent mode



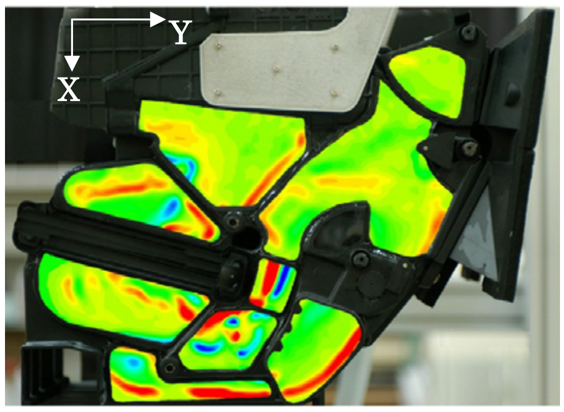
(c) Foot-vent mode

Fig. 7 Mean velocity fields for three different ventilation modes at warm mode

(a) Defrost-vent mode



(b) Hybrid-vent mode



(c) Foot-vent mode

Fig. 8 Vorticity contours for three different ventilation modes at warm mode

in Fig. 6(b), a pair of spanwise vorticity which is rotating in the clockwise and counterclockwise directions appears in front of the panel vent door. The strength of the counterclockwise vorticity moving through the defrost vent is smaller than that of the defrost-vent mode. The air flow going out of the foot vent seems to form strong vorticity.

For the case of the foot-vent mode (Fig. 5(c)), the incoming flow goes toward the foot vent after turning in the clockwise direction, and a stagnation flow region occur at the vicinity of the defrost vent. The speed of the incoming flow is decreased in the region nearby the panel vent due to increased flow resistance as compared to the hybrid-vent mode. The flow moving backward toward the heater core also occurs in this mode. From these results, we can see that the efficiency of an automotive HVAC can be improved at the cool mode by installing a supplementary door at the narrow duct (E), which protects the momentum loss of incoming air. Vorticity contours are similar to those of the hybrid-vent mode, excluding the vanished vorticity in the region near the defrost vent (Fig. 6(c)).

Figures 7 and 8 show the mean velocity fields and vorticity contours according to the ventilation mode at the warm mode. In the case of the warm mode, the temperature baffle (F) blocks the horizontal pathway connected to the panel vent. Therefore, the air flow coming from the evaporator core passes through the heater core (G) and rotates along the duct in the counterclockwise direction. The speed of air passing through the heater core is reduced largely due to the enlarged duct (H). Then the flow speed is increased rapidly while passing through the narrow duct (J). For the defrost-vent mode (Fig. 7(a)), the air flow passing through the narrow duct region moves directly to the defrost vent since the narrow duct and the defrost vent are aligned nearby on a straight line. As shown in Fig. 7, the maximum flow speed appears at the end of the narrow duct for the defrost-vent mode. The closed foot-vent door does not seem to influence the flow pattern of the defrost-vent mode. However, stagnation flow occurs in the region between the panel vent and the foot vent. As shown in Fig. 8(a), a pair of line-shaped vorticity contour exists at the narrow duct. After passing through the narrow duct, a strong vorticity rotating in the counterclockwise direction is formed, and then it moves toward the defrost vent. The counterclockwise vorticity at the vicinity of the temperature baffle results from the bumping of incoming air against the baffle plate. The strength of the pair of turning vorticity is the strongest for the defrost-vent mode as compared to the two other modes.

For the hybrid-vent mode (Fig. 7(b)), after air passes through the narrow duct, some of it moves toward the defrost vent directly, while the others go to the foot vent after rotating in the clockwise direction. The air flow is nearly stagnant at the inside (K) and outside (L) of the chamber in front of the panel vent due to this kind of flow characteristic. The momentum of incoming flow is decreased more or less due to two opened vents as compared to the defrost-vent mode. The flow pattern and vorticity contours (Fig. 8(b)) are similar to those of the defrost-vent mode.

For the foot-vent mode (Fig. 7(c)), the air flow passing through the narrow duct turns its flow direction once again in front of the panel-vent door (B). This flow has an inclined S-shaped path with a large radius of curvature. The stagnation flow region is formed in the large area near the defrost vent (M) due to the closed defrost-vent door. Therefore, the flow speed and momentum have the smallest value among the three ventilation modes tested in this study. The closed defrost-vent door seems to work as a flow resistance inside the HVAC module at warm mode. For the warm mode, since the length of the flow path is much longer than that of the cool mode, the momentum loss is considerably large. As shown in Fig. 8(c), the vorticity is relatively smaller than that of the two other ventilation modes.

4. Conclusions

The effect of ventilation mode on the variation of velocity field and vorticity contours of air flow inside an automotive HVAC module was investigated. The velocity fields were measured using a PIV velocity field measurement technique. Three different ventilation modes (defrost-vent mode, foot-vent mode, and hybrid-vent mode) were tested for two temperature modes.

The velocity fields inside the HVAC module were found to be largely affected by the ventilation mode and the temperature mode. For the hybrid-vent mode at cool mode, the flow resistance is decreased slightly due to two opened vent doors as compared to the defrost-vent mode. For the cool mode, the flow moving backward to the heater core occurs for all ventilation modes tested in this study. This implies that the structural layout of the interior ducts of the HVAC module should be improved to resolve this kind of flow phenomena.

For the warm mode, since the temperature baffle blocks the horizontal pathway connected with the panel vent, the air flow passes through the heater core and goes through a U-shaped curved duct. Since the length of the flow path is much longer than that of the cool mode, the momentum loss is considerably large. For the defrost-vent mode, the air flow passing through the narrow duct moves toward the defrost vent directly. The closed foot-vent door does not seem to influence the flow pattern of the defrost-vent mode. The strength of vorticity is strongest for the defrost-vent mode.

These experimental data would be used to validate numerical predictions. In addition, the present work highlights the usefulness of the PIV velocity field measurement technique as a tool for analyzing a real automotive HVAC system.

Acknowledgement

This work was supported by Automobile Core Basic Technology Development Project from Ministry of Commerce, Industry and Energy, KOREA.

References

Adrian, R. J., Particle-imaging Techniques for Experimental Fluid Mechanics, Annual Review of Fluid Mechanics, 23 (1991), 261-304.
Aroussi, A., Abdul Ghani S. A. A. and Rice, E., PIV Measurement and Numerical Simulation of Airflow Field in a Road Vehicle HVAC Cowl Box, SAE technical paper, (2001), 2001-01-0294.
Bennett, L., Dixon, C. W. S. and Watkins, S., Modeling and Testing of Air Flow in a HVAC Module, SAE technical paper, (2002), 2002-01-0506.
Cho, N. H. and Kim, M. R., Numerical Investigation of Fluid Flow in an Automotive HVAC Module, SAE paper, (1997), 971778.
Currie, J., Numerical Simulation of the Flow in a Passenger Compartment and Evaluation of the Thermal Comfort of the Occupants, SAE paper, (1997), 970529.
Hirota, M., Asano, H., Nakayama, H., Asano, T. and Hirayama, S., Three-dimensional Structures of Turbulent Flow in Mixing T-junction, JSME International Journal Series B, 49 (2006), 1070-1077.
Ishihara, Y., Shibata, M., Hoshino, H., Hara, J. and Kamemoto, K., Analysis of Interior Airflow in a Full-scale Passenger Compartment Model Using a Laser-light-sheet Methods, SAE paper, (1992), 920206.
James, L., Ken, H. and Karim, N., Design and Development of a Cylindrical HVAC Case, SAE technical paper, (2004), 2004-01-1385.
Jang, Y. I. and Lee, S. J., Visualization of Turbulent Flow around a Sphere at Subcritical Reynolds Numbers, Journal of Visualization, 10 (2007), 359-366.
Kitada, M., Asano, H., Kanbara, M. and Akaike, S., Development of Automotive Air-conditioning System Basic Performance Simulator: CFD Technique Development, JSAE Review, 21 (2001), 91-96.
Lin, C. H., Experimental and Computational Studies of Flow in a Simplified HVAC Duct, International Journal of Vehicle Design, 15 (1994), 147-165.
Shojaee, M. H., Tehrani, F. P. H., Noorpoor, A. R. and Adili, M. R., Analysis of Vehicle Passenger Compartment HVAC Using Simulation, SAE technical paper, (2004), 2004-01-1505.
Wan, J. W. and Kooi, J., Influence of the Position of Supply and Exhaust Opening on Comfort in a Passenger Vehicle, International Journal of Vehicle Design, 12 (1991), 588-597.
Yoon, J. H. and Lee, S. J., Velocity Field Measurements of Ventilation Flow in a Vehicle Interior, International Journal of Vehicle Design, 31 (2003), 96-111.

Author Profile

Jong Hoon Kang: He received his MSc and Ph.D. in Mechanical Engineering from POSTECH (Pohang Science and Technology) in 2004 and 2009, respectively. Now he is working in POSCO. His research interests are PSP technique, flow visualization, applications of PIV/PTV velocity field techniques, bluff-body aerodynamics, ventilation flow, and wind engineering.

Sang Joon Lee: He received his MSc and Ph.D. in Mechanical Engineering from KAIST (Korea Advanced Institute of Science and Tech.) in 1982 and 1986, respectively. In 1986 he worked as a senior researcher at KIMM. He joined the Department of Mechanical Engineering at POSTECH as an Assistant Professor in 1987, and in 1999 he became a full professor. His research interests include quantitative flow visualization (PIV, PTV, LIF, X-ray imaging and Holography), experimental fluid mechanics, aerodynamics, bio-fluid flows, microfluidics, and flow control.