Jurnal
Teknologi

Full Paper

EFFECTS OF VENTILATION SETUPS ON AIR FLOW VELOCITY AND TEMPERATURE FIELDS IN BUS PASSENGER COMPARTMENT

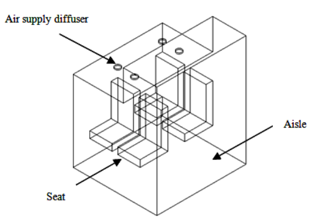
Noor Emilia Ahmad Shafie, Haslinda Mohamed Kamar*, Nazri Kamsah

Faculty of Mechanical Engineering, Universiti Teknologi Malaysia, 81310 UTM Johor Bahru, Johor, Malaysia

Article history
Received
5 August 2015
Received in revised form
21 November 2015
Accepted
28 November 2015

*Corresponding author
haslinda@mail.fkm.utm.my

Graphical abstract



Abstract

A bus compartment requires a good ventilation system to provide sufficient fresh air and a uniform air flow to passengers. This article presents a numerical study using CFD method to investigate the effects of using different ventilation setups on the air flow velocity and temperature distributions inside a passenger bus. Fluent software was used to develop a simplified three-dimensional model of a quarter section of a bus passenger compartment. Turbulent flow simulation was carried out based on a standard k-epsilon model to predict the distributions of air temperature and velocity inside the passenger compartment. The effects of two ventilation setups, namely mixing and displacement ventilations on the air temperature and air flow velocity distribution were also examined. Results of CFD simulations show that the displacement ventilation setup results in more uniform distribution of air flow velocity and air temperature inside the bus passenger compartment.

Keywords: Mixing ventilation setup, displacement ventilation setup, bus passenger compartment, CFD flow simulation, turbulent flow analysis

1.0 INTRODUCTION

A bus passenger compartment requires a good ventilation system that can supply sufficient fresh air and uniform air flow velocity and temperature distribution. A poor ventilation system would elevate the concentration level of air contaminants which could affect the passenger's health especially for those who are travelling long distances [1]. A number of researchers have carried out studies to characterize the air flow velocity distribution inside bus passenger compartments. Zhu *et al.* [2] examined the transport of airborne particles, which can cause disease on passengers, for various ventilation systems. They used CFD method to simulate the air flow inside the bus passenger compartment. They found that a displacement ventilation system is more effective in providing cleaner air inside the passenger compartment thereby minimizing the risk of disease to passengers due to airborne particles. Zhu *et al.* [3] also investigated the air flow distribution inside a bus passenger compartment using the CFD method. They concluded that poor ventilation system leads to an increase risk for disease due to airborne particles in public buses. Li *et al.* [4] have investigated the level of airborne contaminants inside a bus passenger compartment using CFD method. They found that the layout of the air supply diffusers, seats arrangement and passengers clothing have significant effects on the level of concentration of the airborne contaminants inside the passenger compartment.

This article presents a numerical study using CFD method, to examine the effects of air supply diffuser placement on the air flow velocity and temperature distributions inside a passenger compartment of a university shuttle bus. Ansys Fluent software was used to develop a simplified three-dimensional model of a quarter section of the bus passenger compartment.

Turbulent flow analysis was performed using a standard k-epsilon flow model. The distribution of air flow velocity and air temperature inside the bus compartment model was observed. The effects of locations of the air supply diffusers on the air flow velocity and air temperature distributions were examined.

2.0 METHODOLOGY

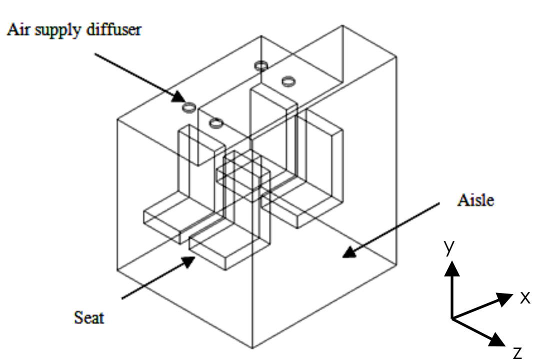
2.1 Field Measurement

Field measurements were conducted to measure the air temperature and air flow velocity at the air supply diffusers, passenger seat and floor level inside a passenger compartment of a university's shuttle bus. A digital anemometer (model V816B) was used to measure the air temperature and velocity data. The instrument is shown in Figure 1. The range and accuracy for air temperature and air velocity measurements are ±2°C and ±3%, respectively.

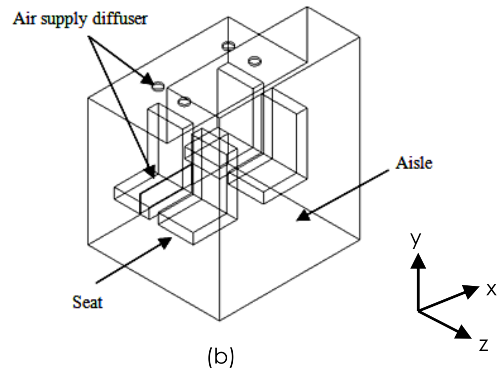
2.2 CFD Simulation Setup

Fluent CFD software was used to construct a simplified three-dimensional model of the quarter section of the bus passenger compartment. The CFD models are shown in Figure 2. In Figure 2(a), a mixing ventilation setup was incorporated into the model while a displacement ventilation setup for the passenger compartment is shown in Figure 2(b). The length (x-direction), width (z-direction) and height (y-direction) for both the CFD models are 1.5 m, 1.6 m and 2.4 m, respectively.

Figure 1 Digital anemometer



(a)



(b)

Figure 2 Simplified CFD model of quarter section of a bus passenger compartment. (a) Mixing ventilation setup and (b) Displacement ventilation setup

Four passenger seats are included into the models. For the mixing ventilation setup, two air supply diffusers are placed in a one row on the ceiling mounted duct work, above the passenger seats. For the displacement ventilation setup two air supply diffusers are placed in a one row on the ceiling mounted duct work and one air supply diffuser is placed at the side walls of the passenger compartment. The CFD models were meshed using tetrahedral elements. The meshed CFD model is shown in Figure 3. The total number of elements used was 110578. This number was chosen based on a simple grid independent test carried out on the CFD models. This will be briefly described in the following section 2.3.

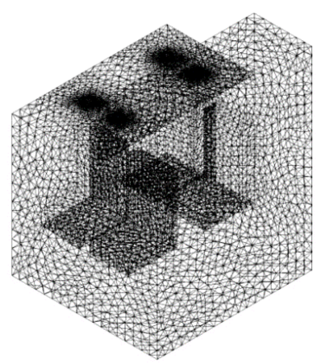


Figure 3 The meshed CFD model of the bus passenger compartment

The CFD analysis on the simplified models was carried out with the following boundary conditions. Cool air enters the compartment from the air supply diffusers located above the passenger seats. The temperature and flow velocity of the air were set at 23°C and 2.9 m/s, respectively, based on the values obtained from the field measurement. The temperatures of the floor and wall surfaces were set at 26°C and 25°C, respectively. These are illustrated in Figure 4. The same air flow boundary conditions were prescribed on both the CFD models.

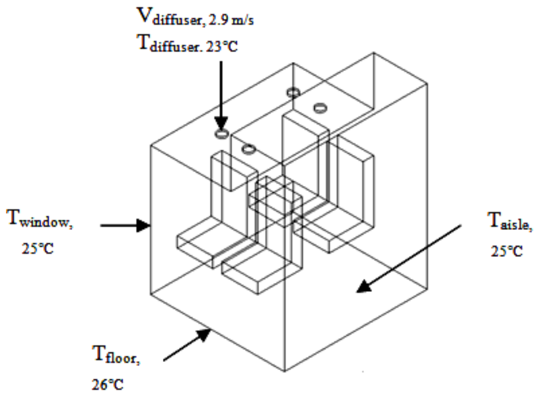


Figure 4 The boundary conditions prescribed on the CFD model for turbulent flow analysis

A turbulent flow analysis was carried out using the standard *k-ε* turbulent model. The turbulent intensity was set to 10 %. A no slip condition was prescribed at the walls and the floor surfaces of the passenger compartment [6]. The air was assumed to behave as an incompressible fluid. Ideal gas properties were assumed for the air. The reference pressure of the air inside the passenger compartment was set to atmospheric pressure, which is 101325 Pascal. The turbulent flow simulations were carried out at steady-state condition, employing a pressure-based segregated solver with a SIMPLE, second order upwind discretization scheme. The convergence criterion for all equations was set at 10^{-4} except for the energy equation, which was set at 10^{-6} [5].

2.3 Grid Independence Test

A grid independence test (GIT) was performed on the CFD models to ensure that the size of the meshing will have negligible effects on the results of the CFD analysis. This test was done by carrying out on the CFD model with the boundary conditions as described in the previous section. The number of elements used was varied for each case. They were 50000, 100000, 150000 and 200000 tetrahedral elements. For each case, the average value of air temperature in-front of the left front passenger seat was observed. The average air temperature values were then plotted against the number of elements used. This is shown in Figure 5. It can be observed form the figure that the value of average temperature saturates as the number of elements used was roughly about 100000 elements. Hence the exact number of elements of 110578 was used for both the CFD models.

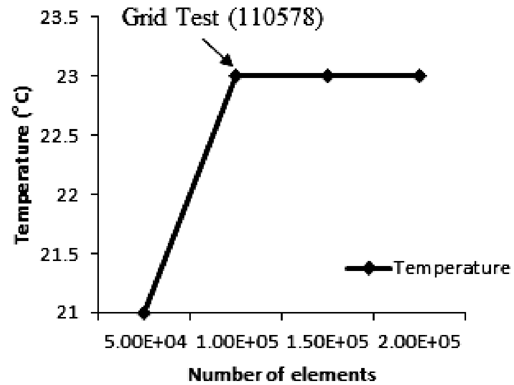


Figure 5 Average air temperature versus number of elements used on the CFD model

2.4 Validation of CFD Model

The CFD simulation model was validated by comparing the average air temperature at the air supply diffusers, at the vicinity of the front left passenger seat and at the floor surface obtained from the turbulent flow simulation, with corresponding values obtained from the field measurements. The comparison is summarized in Table 1. The percent difference at the air supply diffusers and the floor surface is obviously 0% since the same values of air temperature were used for the boundary conditions at these locations. The difference in the average air temperature in the vicinity of the left front seat is only 0.9%. This result suggests that the CFD model is capable of predicting the temperature of the air inside the passenger compartment with a fairly good accuracy.

Table 1 Comparison of the average air temperature between predicted and measured values

Locations	Avg. Air Temperature (°C)		Percent difference (%)
	CFD Result	Measured	
Air supply diffusers	23	23 ±9	0
Left front seat	23.2	23 ±3	0.9
Floor surface	26	26 ±6	0

3.0 RESULTS AND DISCUSSION

Figure 6 shows a contour of air temperature distribution on a vertical plane that passes through the left passenger seats, when the mixing ventilation setup was employed. The corresponding contour of air temperature distribution for the displacement ventilation setup is shown in Figure 7.

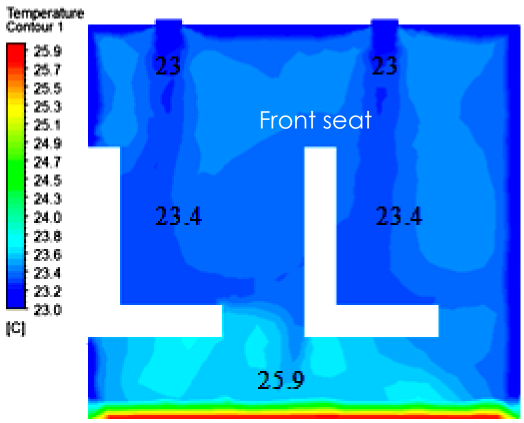


Figure 6 The contour of air temperature distribution inside the passenger compartment for the mixing ventilation setup

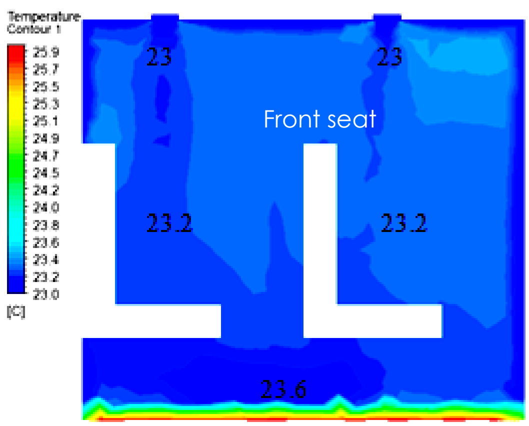


Figure 7 The contour of air temperature distribution inside the passenger compartment for the displacement ventilation setup

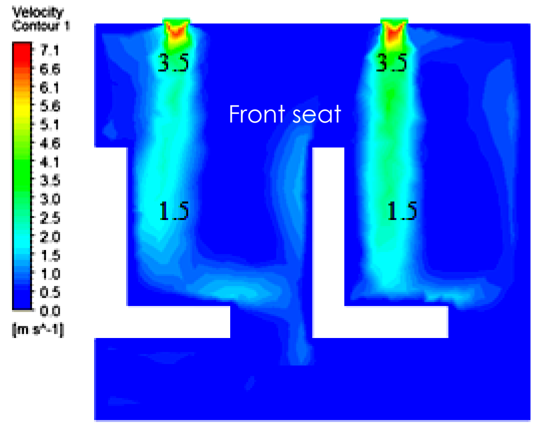


Figure 8 The contour of air flow velocity distribution inside the passenger compartment for the mixing ventilation setup

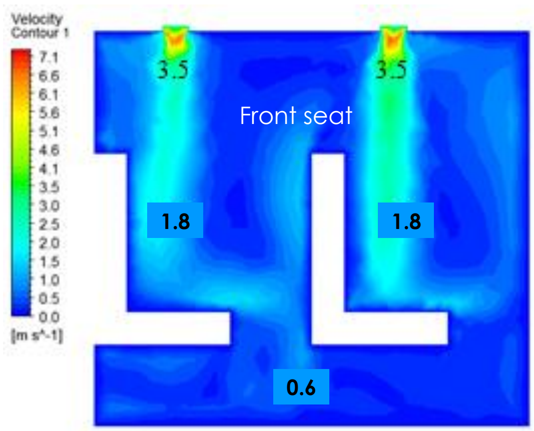


Figure 9 The contour of air flow velocity distribution inside the passenger compartment for the displacement ventilation setup

It can be observed in Figure 6 that the average air temperature in the region at the front of both passenger seats is about 23.4°C. The air temperatures at supply diffusers and the floor surface are 23°C and 25.9°C, respectively, as prescribed in the boundary conditions. The air temperature above the floor surface is seen to vary from about 24°C to 25.3°C. The air temperature above the passenger seats varies from about 23.2°C to 23.5°C. With the displacement ventilation, the air temperature above the floor surface is seen to vary between about 23.1°C and 23.3°C, as seen from Figure 7. The air temperature above the passenger seats is seen a lot more uniform at a value of about 23.1°C. Hence the use of displacement ventilation setup results in a more uniform temperature distribution inside the passenger compartment. Slightly lower air temperature can be attained inside the passenger compartment when the displacement ventilation setup is employed.

Figure 8 shows a contour of air flow velocity distribution on a vertical plane that passes through the left passenger seats, when the mixing ventilation setup was used. The corresponding contour of air flow velocity distribution for the displacement ventilation setup is shown in Figure 9. The CFD simulation result shows that the air flow velocity at the vicinity of the air supply diffuser is about 3.5 m/s. The air velocity in the region immediately in-front of the passenger seats is ranging from about 1.5 m/s to 2.0 m/s when the mixing ventilation setup was employed. In the region above the floor there is almost no air movement. It is observed from Figure 9 that the air flow velocity distribution is more distributed inside the passenger compartment when the displacement ventilation setup was used. The air flow velocity at the air supply diffuser is about 3.6 m/s. The average air flow velocity in the vicinity of the passenger seats is about 1.8 m/s. In the region above the floor, the average air velocity is about 0.6 m/s. Hence the use of displacement ventilation results in a distributed air flow velocity inside the passenger compartment. The average air flow velocity at several locations inside the compartment is slightly higher than the values when mixing ventilation setup was used.

4.0 CONCLUSION

A numerical study using CFD method was carried out to examine the effects of using different ventilation setups on the distributions of air flow velocity and air temperature inside a bus passenger compartment. It was found that the displacement ventilation setup is capable of providing more uniform distributions of air flow velocity and air temperature inside the passenger compartment. Higher air flow velocity is obtained in the vicinity of the passenger seats and lower air temperature in the region is also achieved.

Acknowledgement

The authors are grateful to the Universiti Teknologi Malaysia for providing the funding on this study, under the vot number 06H75.

References

[1] Awbi, H. B. 2003. *Ventilation of Buildings*. London: Taylor & Francis.

[2] Zhu, S., Srebric, J., Spengler, J. D. & Demokritou, P. 2012. An Advanced Numerical Model for the Assessment of Airborne Transmission of Influenza in Bus Microenvironments. *Building and Environment*. 47: 67-75.

[3] Zhu, S., Demokritou, P. & Spengler, J. 2010. Experimental and Numerical Investigation of Micro-Environmental Conditions in Public Transportation Buses. *Building and Environment.* 45(10): 2077-2088.

[4] Xiang, L. & Wang, H. 2011. Numerical Simulation of Pollutant Transport within an Vehicle Cabin. *In Advances in Computer Science, Environment, Ecoinformatics, and Education*. Springer Berlin Heidelberg.

[5] Bari, S. & Naser, J. 2010. Simulation of Airflow and Pollution Levels Caused By Severe Traffic Jam in a Road Tunnel. *Tunnelling and Underground Space Technology*. 25(1): 70-77.

[6] Fluent, A. N. S. Y. S. 2012. *14 User Manual.* ANSYS Inc. New York.