NUMERICAL SIMULATION AND EXPERIMENT RESEARCH OF AIR ORGANIZATION IN AIR-CONDITIONED PASSENGER CAR*

Dengchun ZHANG[1, 2], Peifen WENG[1]
[1]Shanghai Institute of Appl. Math. and Mech., Shanghai University, Shanghai 200072, China;
[2]School of Energy and Safety Engineering, Hunan University of Science and Technology, Xiangtan, 411201, China

（*Project Supported by the Excellent Youth Foundation of the Education Bureau of Hunan Province, China）

ABSTRACT

By $k-\varepsilon$ turbulence model, the indoor air fluid flow and heat transfer conjugated programs in the YZ_{25K} air-conditioned passenger car are solved with SIMPLE algorithm. The effect of solar radiation and body heat is taken into account and these auxiliary heat flows are the source terms of the energy equation. Regarding the push flow and the shape of passenger car and obstacles as an integer, the three dimension distributions of velocity and temperature field in the chamber are numerically simulated, and the effects of supply air mode and supply air velocity on airflow and temperature field are investigated. An experimental program was also performed to validate the airflow and temperature distribution. The experiment is carried out in the YZ_{25K} air-conditioned passenger car. The agreement between the computed and measured results of temperature and velocity field is quite well. It is also shown that the prediction results of airflow and temperature field will give valuable information to be used for evaluating the thermal comfort in the design procedure.

KEYWORDS

Passenger car; $k-\varepsilon$ model; Air organization; Numerical simulation

INTRODUCTION

The velocity and temperature field in air-conditioned passenger car are the basis of investigating the air organization and evaluating thermal comfort in the design procedure. In the last decades, the air organization design in air-conditioned passenger car was usually referred to indoor air organization design, regarding the push flow as jet [1]. Then the velocity and temperature distribution of some certain sections in the car could be obtained through solving empirical formulae of jet. So the location and size of supply air orifices and supply air velocity could be adjusted according to the conformity between the velocity and temperature distribution and the standard of design. Only recently, a new development tool has gained increased interest namely computational fluid dynamics （CFD）. Several authors have applied numerical simulation successfully for the optimization of different components of the HVAC system [2-4]. Also the flow in the air-conditioned passenger car has been calculated by different authors [5-7]. All these research were focused on the airflow simulation in vehicle passenger car. But the research about air distribution in railway passenger car is less.

In the present paper, by $k-\varepsilon$ turbulence model, the air flow and heat transfer problems in air-conditioned passenger car are solved with SIMPLE algorithm. The effect of solar radiation is taken into account and calculated with Monte Carlo method. The auxiliary heat flows of body heat and solar radiation are the source term of the energy equation. Regarding the push flow and the shape of train and obstacles as an integer, the three dimension distributions of velocity and temperature field in the chamber are numerically simulated, and the effects of supply air mode and supply air velocity on airflow and temperature field are investigated. An experimental program was also performed to validate the airflow and temperature distribution. The experiment is carried out in the YZ_{25K} air- conditioned passenger car. The agreement between the computed and measured results of temperature and velocity field is quite well. It is also shown that the prediction results of airflow and temperature field in air-conditioned passenger car will give valuable information to be used for evaluating the thermal comfort in the design procedure.

COMPUTATIONAL MODEL

2.1 Physical model

In this paper, half of the YZ_{25K} air-conditioned passenger car is investigated. The part is 9 600 mm long, 2 900 mm wide and 2 500 mm high（net height）. The left of the car is fixed twelve chairs, the uniform distance of every other chair is 600 mm, and the width of each chair is 1 350 mm（three persons）or 900 mm（two persons）, the layout of the right is similar to the left. Two racks are fixed on the two upside wall respectively. There are six supply air orifices（400 mm×400 mm）in the roof of the passenger car. The door（850 mm ×2 070 mm）is used to return air. The computational model is shown by **Fig.1**.

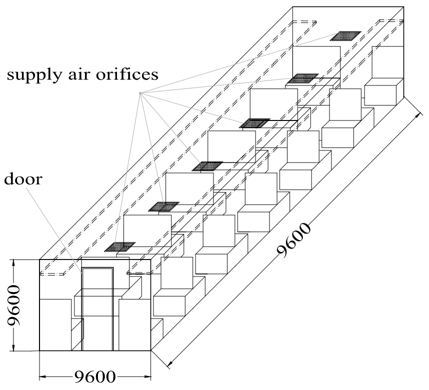


Fig.1 Computational model of YZ_{25K} air-conditioned passenger car

2.2 Governing equations

In this paper, $k-\varepsilon$ turbulence model is used. In order to simplify the problem, some assumptions are made [8-9].

a. Air fluid in the car is incompressible.
b. Air fluid is stable flow.
c. Emission heat transfer between solid walls is ignored, and air in the car is looked as transparent media.
d. High Reynolds number and the same turbulence viscosity in different directions.
f. Good air- tightness.

According to the assumptions, the universal governing equation of air flow and heat transfer in the air-conditioned passenger car is described as [10]:

$$\mathrm{div}(\rho\vec{V}\Phi) = \mathrm{div}(\Gamma_\Phi \mathrm{grad}\Phi) + S_\Phi \qquad (1)$$

Where ρ is the air density, $\vec{V}$ is the air velocity vector, Φ denotes the universal variables, 1, u , v , w , T , k and ε respectively, Γ_Φ is the effective diffusion coefficient, and S_Φ represents the source term for each of the variables. Φ , Γ_Φ and S_Φ are listed in **Table 1**.

Tab.1 Coefficients of general governing equations

Equation	Φ	Γ_Φ	S_Φ
Continuity	1	0	0
Momentum	U_i	$\nu+\nu_t$	$-\frac{\partial}{\partial X_j}\left\lvert\frac{P}{\rho}+\frac{2}{3}k\right\rvert-\beta(T-T_0)g_i$
Energy	T	$\nu/P_r + \nu_t/\sigma_T$	$q/\rho C_p$
k-Equation	k	$\nu+\nu_t/\sigma_k$	$\nu_t+G-\varepsilon$
ε-Equation	ε	$\nu+\nu_t/\sigma_\varepsilon$	$\varepsilon(C_1\nu_t S - C_2\varepsilon + C_3 G)/k$

COMPUTATIONAL METHOD

3.1 Computational method

On the basis of the YZ_{25K} air-conditioned passenger car model, we meshed the computational model in Descartes' right-angle coordinate system using 112×42×35 uniform grid. Length is loaded on x-orientation, width on y-orientation and height on z-orientation. The joint of the sidewall and the bottom of door is regarded as coordinate initial point. The governing equations are discretized by control volume method on staggered grid and solved by SIMPLE algorithm [12-13]. To ensure the stability of the computation, under-relaxation technique is applied to all the equations [14]. Overall continuity corrections for velocity and pressure at the section of door opening are added to assure the mass balance across the door opening [15]. The air fluid flow and heat transfer conjugated programs are solved as an integer. The viscosity coefficients of obstacles in the car such as chairs and racks are infinite. So we could make it rational that the air velocity close to the obstacles is zero. The flow properties at the grid nodes near solid surface are computed through the wall function method [16-17]. The temperature on the obstacles wall is calculated according to adiabatic condition. **Fig.2** shows the control volume and nodes site.

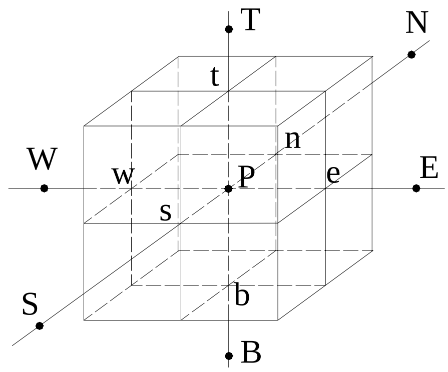


Fig.2 control volume and nodes site

3.2 Boundary conditions

The airflow and temperature field in YZ_{25K} air-conditioned passenger car is to be investigated and simulated. The six supply air orifices are calculated as inlet boundary, the door for returning air is calculated as outlet boundary. As air distribution of the entire car in length direction is symmetric, half of the car is investigated for making problem simpler, and the middle section in the car is calculated as adiabatic boundary.

Inlet boundary: $u = v = 0$， $w = 1.0$ m/s， $T = 291$ K， $k = 0.002$， $\varepsilon = 0.0008$；

Outlet boundary: $P = P_{out}$， k 、 ε： Free slip.

Wall boundary: The roof and two sidewalls of the car are constant heat flow, and the synthesis heat transfer coefficient K=1.16 W/（m^2·℃）；The synthesis temperature out of the car is 303 K. The floor, inner wall, chairs, racks and the symmetric section are adiabatic；The roof, two side walls, the floor, inner wall, chairs and racks are no slips ，

$u = v = w = k = \varepsilon = 0$; The supposed middle section is symmetric, and the other boundaries have no streamlines to penetrate [18].

The heat sources in the car are mainly body heat. The heat from each body is 115 W, and we scatter the heat on chairs uniformly. The heat of solar radiation from glass windows is calculated according to Monte Carlo method. Both the heat is regarded as the additional source term of the energy equation.

EXPERIMENTAL METHOD

In order to validate the simulation results of airflow and temperature distribution, an experiment was performed in the YZ_{25K} air-conditioned passenger car. **Fig.3**～**Fig.4** show the measured points on the section x =1.9 m and x =2.9 m separately. A data logger that can measure temperature and velocity instantaneously was used to record the data from each measured point. Measurements of air velocity field were taken using omnidirectional hot-wire anemometers reading the scalar velocities. At each measurement in the test chamber, air velocity and temperature were measured at two, five, and three points along x , y and z direction, respectively. The measurement errors of the velocity and temperature field were estimated to be within ±0.01 m/s and ±0.1 ℃ of the measured values, respectively.

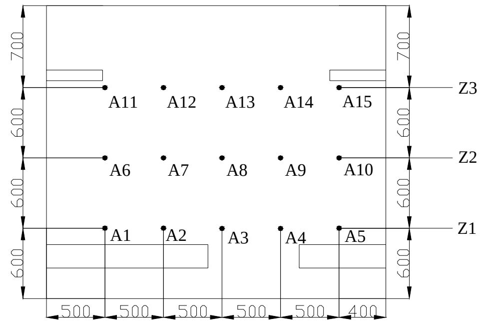


Fig.3 Measured points on the section x =1.9 m

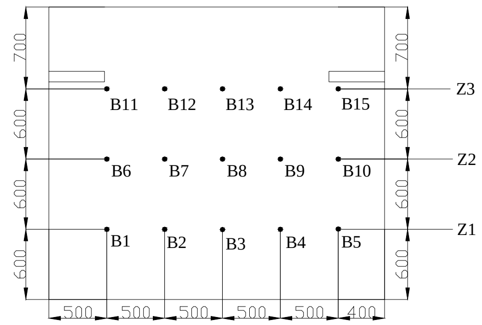


Fig.4 Measured points on the section x =2.9 m

RESULTS AND ANALYSIS

The computation of the temperature and velocity field was done in the YZ_{25K} air-conditioned passenger car. **Fig.5**～**Fig.6** show the distributions of velocity vector at different sections, **Fig.7**～**Fig.8** show the temperature distributions at the corresponding sections respectively.

The location of supply air orifices has an obvious effect on the distribution of airflow and temperature field. As supply air orifices locate at the middle of ceiling of the car, the flow intensity in the central of the car is bigger, and the temperature in the central of the car is lower. As the buoyancy effect of body heat, the temperature in the chair zone is higher, and two big eddies are formed at the two sides of the car. As the effect of solar radiation on the right sidewall of the car, the temperature near the right sidewall in the chair zones is higher than that of other zones in the chamber. The maximum temperature difference between them is 2.3 ℃.

Supply air velocity in some rang（υ=1.0 m/s～2.0 m/s）has little effect on air distribution of airflow field. Velocity of each section varies with the supply air velocity, and the velocity at the upper section is always higher than that at the lower section in the car. When the air supply velocity is 1.0 m/s, the mean velocity on the section located at z =0.6 m is about 0.13 m/s, While the air supply velocity is 2.0 m/s, the mean velocity on the section located at z =0.6 m is about 0.15 m/s.

The obstacles such as chairs, racks have obvious effect on air distribution of velocity and temperature field. The airflow field is dissymmetric because of different layout of chairs. A big eddy emerged upper the left chairs, and the big eddy upper right chair leave to the left clearly. The temperature near the chairs and racks is higher because the heat near them is not easily removed, and could not form intensive convection.

The comparisons of numerical and experimental results of velocity field on the section x =1.9 m and x =2.9 m are illustrated quantitatively in **Fig.9**～**Fig.10**. In order to match the same locations measured points, the simulation results of on the section x =1.9 m were taken for comparisons as shown in **Fig.9**. The subfigures（**a**）-（**c**）are presented the results that three different constant z -directional points were taken for comparing the velocities. As shown in subfigures （**a**）-（**c**）, the comparison between predicted and experimental results is quite well. The average deviation is 11% and the maximum one is about 20%. **Fig.10** illustrates the comparisons results on the section x =2.9 m using three different z -directional points to comparing y -directional velocity between measures and predictions. The average deviation is about 8% and the maximum one is about 20%. The rather high discrepancies can be found in subfigures（**b**）and（**c**）in **Fig.10**. The comparisons of numerical and experimental results of temperature field on the section x =1.9 m and x =2.9 m are illustrated quantitatively in **Fig.11**～**Fig.12**. **Fig.11** shows the comparison between the temperature results measured and simulated on the section x =1.9 m with three different z -directional points. Notice the agreement covers most data points. The average discrepancy is under 2% approximately for all test points. As body heat regarded as a heat source and solar radiation regarded as an auxiliary heat flows, they have great compact on temperature field. The temperature close to chairs is higher than that in the central of car because passengers always crowed at chairs. The temperature of chair zone is higher than that of passageway, and the maximum temperature difference among them is 1.0 ℃. **Fig.12** shows the temperature comparison results on the section x=2.9 m using three different z-directional points（0.6 m, 1.2 m, and 1.8 m）. As the supply air orifices locate at the center of the train roof, the temperature difference between the center and sidewall is obvious. The maximum temperature difference among them is 1.3 ℃（**Fig.12c**）.

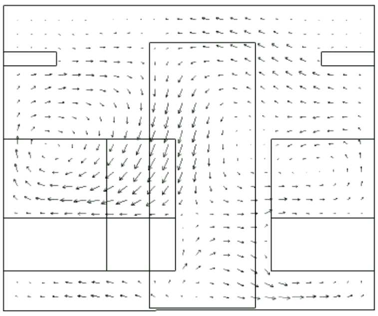


Fig.5 Velocity vectors on the section x=1.9 m

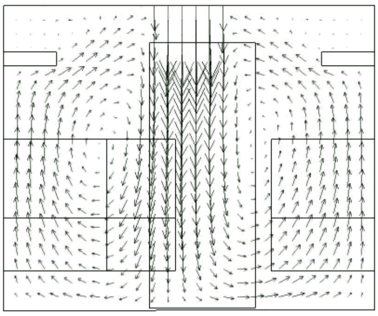


Fig.6 Velocity vectors on the section x=2.9 m

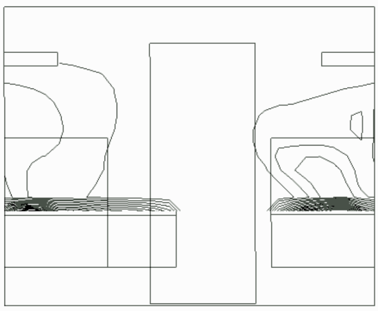


Fig.7 Temperature contours on the section x=1.9 m

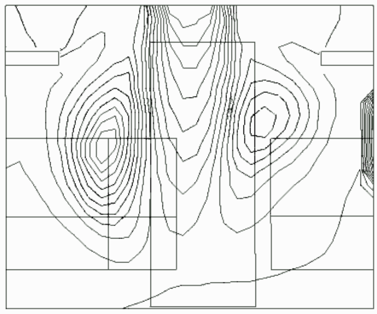
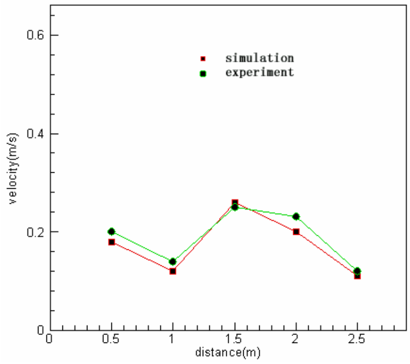
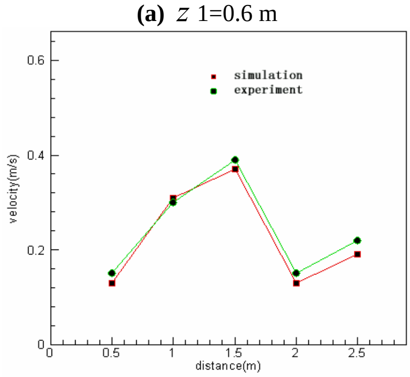


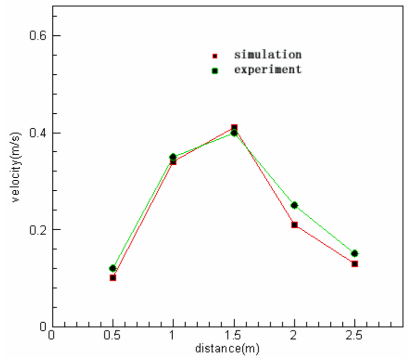
Fig.8 Temperature contours on the section x=2.9 m



(a) z 1=0.6 m

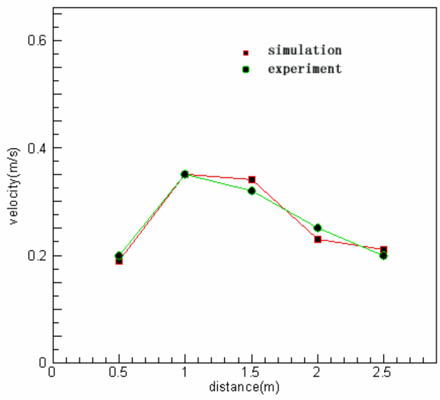


(b) z 2=1.2 m

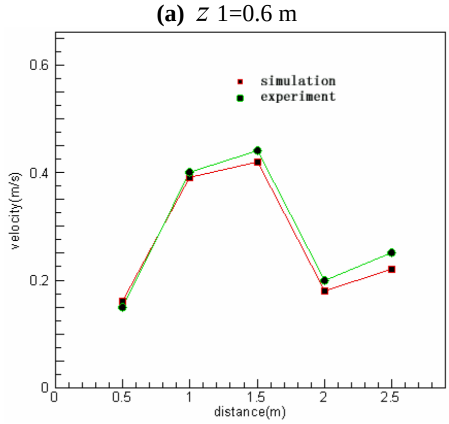


(c) z 3=1.8 m

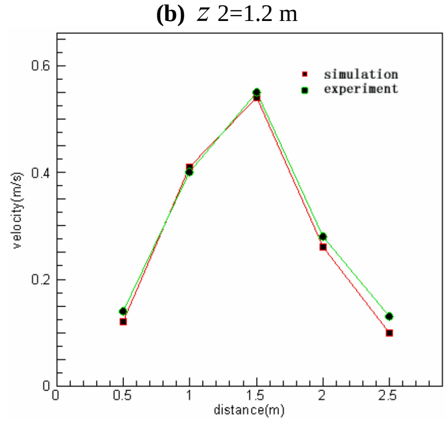
Fig.9 Comparison of measured and predicted velocity on the section x=1.9 m



(a) z_1=0.6 m

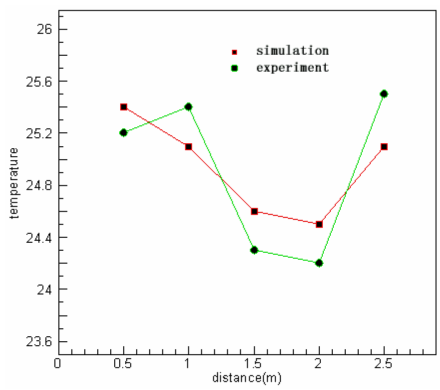


(b) z_2=1.2 m

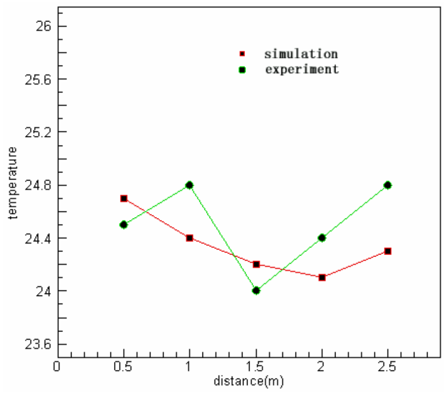


(c) z_3=1.8 m

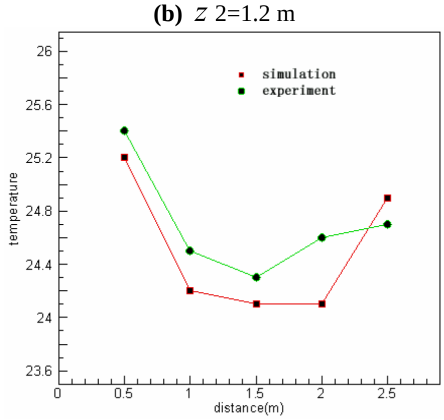
Fig.10 Comparison of measured and predicted velocity on the section x=2.9 m



(a) z_1=0.6 m

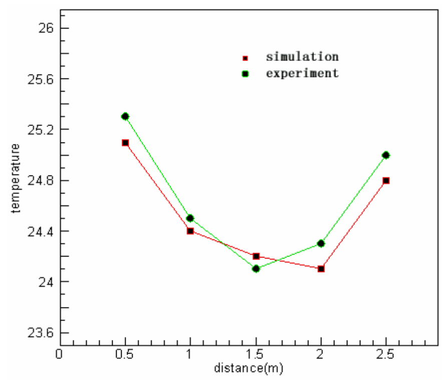


(b) z_2=1.2 m

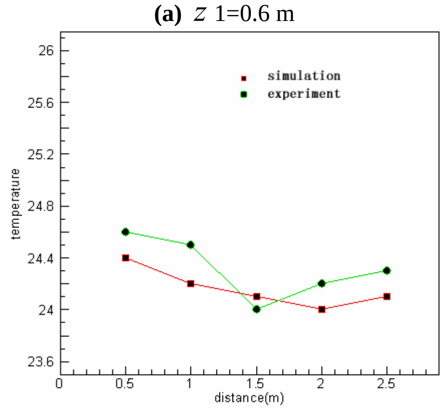


(c) z_3=1.8 m

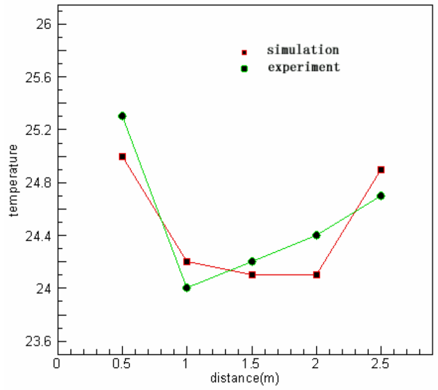
Fig.11 Comparison of measured and predicted temperature on the section x=1.9 m



(a) z_1=0.6 m



(b) z_2=1.2 m



(c) z 3=1.8 m

Fig.12 Comparison of measured and predicted temperature on the section x =2.9 m

CONCLUSIONS

A computational model of air flow and heat transfer in air-conditioned passenger car was established. The distribution of airflow and temperature field was successfully predicted by $k-\varepsilon$ turbulence model and validated through an experiment in an air-conditioned passenger car. The maximum discrepancy of the velocity distribution between experiment and simulation is 20% approximately. However, the average overall discrepancy is under 10% approximately between measured and predicted velocity. The agreement between the computed and measured results of temperature field is quite well. The average discrepancy is under 2% approximately for all test points.

The investigated results show that the layout of supply air system has great effect on air conditioning in the passenger car, and the location of supply air orifices has large effect on airflow and temperature fields, but the supply air velocity has little effect on airflow field in the rang of υ=1.0 m/s～2.0 m/s, two big eddies emerge at the two sides of the car. With the supply air velocity increasing, the eddies move upwards.

The temperature distribution varies greatly in the height direction, and the temperature of the upper zones is lower than that of the lower zones. As the effect of solar radiation on the right sidewall of the car, the temperature near the right sidewall in the chair zones is higher than that of other zones in the chamber. The maximum temperature difference between them is 2.3℃.

REFERENCE

[1] SHASE Handbook. Japan: Society of Heating, Air-conditioning and Sanitationing Engineers, 1996

[2] K. C. Chung. Three-Dimensional Analysis of Airflow and Contaminant Particle Transport in a Partitioned Enclosure. Building and Environment, 1999, 34(3): 7～17

[3] J. S. Zhang, G. J. Wu, L. L. Christianson. A New Similitude Modeling Technique for studies of Nonisothermal Room Ventilation Flows. ASHRAE Trans, 1994; 100(2): 129～138

[4] Lu W. Z, Andrew T, Alan P. Prediction of Airflow and Temperature Field in a Room with Convective Heat Source. Building and Environment, 1997; 32(10):541～550

[5] Han T. Three-Dimensional Navier-Stokes Simulation for Passenger Compartment Cooling. Int J of Vehicle Design, 1989; 10(2): 223~235

[6] Tohru Komoriya. Analysis of vehicle passenger compartment ventilation using experimental and numerical models. SAE Trans, 1991, 98(6):392～400

[7] Lin C H, Lelli M A, Han T, et al, A experimental and computational study of cooling in a simplified gm-10 passenger compartment. SAE paper 910216, 1991

[8] Komoriya T, kobayashi T, Tanihuchi N. Numerical simulation of the flow in a vehicle passenger compartment using general co-ordinate system with finite volume method. JSAE Review, Vol. 12, No.3, 1991

[9] Joachim Currle. Numerical simulation of the flow in a passenger compartment and evaluation of the thermal comfort of the occupants. SAE paper 970529:806～816, 1997

[10] Nielsen P. V. The Selection of Turbulence Models for Prediction of Room Airflow. ASHRAE Transaction, 1998; 104(1):1119～1126

[11] Lander B E and Spalding D B. The Numerical computation of turbulent flows. Comp. Mech. in Mech. Eng. 1974; 3:269～289

[12] Patanker S V. Numerical heat transfer and fluid flow. New York: Hemisphere Obliging Corporation, 1980

[13] J C S Lai and C Y Yang. Numerical Simulation of turbulence suppressions of the performance of four K-ε turbulence models. Int J Heat and Fluid Flow. 1997; 18:575～584

[14] Patanker S V and Spalding D B. A calculation procedure for heat, mass, and momentum transfer in three-dimensional flow. Int J Heat Mass Transfer, 1972; 15:1787～1806

[15] W K Chow. Application of computational fluid dynamics in building services engineering. Building and Environment, 1996; 31(5):425～436

[16] Blosch E, Shyy W, Smith R. The role of mass conversation in pressure-based algorithms. Numerical Heat Transfer, 1993; 24:415～429

[17] Gartling D K. A test problem for outflow boundary conditions-flow over a backward-facing step. Int J Number Mech Fluids, 1990; 11:953～967

[18] Awbi H B. Application of computational fluid dynamics in room ventilation. Building and Environment, 1989, 24(1):73～83