Journal of Computers Vol. 29 No. 2, 2018, pp. 34-44
doi:10.3966/199115992018042902004

Analysis of Automobile Air Conditioner's Flow Field and Improvement of Air Inlet

Qing-Hui Zhou[1], Li-Yan Liu[1*], Zhan Chen[1]

[1] School of Mechanical-Electronic and Vehicle Engineering, Beijing University of Civil Engineering and Architecture, Xicheng District, Beijing, ROC
1542695721@qq.com

Received 24 July 2017; Revised 19 September 2017; Accepted 19 October 2017

Abstract. The design of the modern automotive air mostly turns to the operating parameters, such as the air supply temperature, the relative humidity of the supply air and the air flow. There is a lack of research on the influence of the different number and location of air outlets on the driving room temperature and humidity field. Based on the numerical analysis of the cab simulation model and the experimental analysis of the cooling performance of the air-conditioner, the velocity field and temperature field distribution of the different air outlets are studied. First, the temperature field model of the cab is established and then use Fluent to simulate, finally, confirmatory tests carried out. By analyzing the distribution of flow field and temperature field of different sections, the distribution characteristics of air flow in the cab are obtained, and the accuracy of the model is proved. After adding two air vents on both sides of the B column, the average temperature of the front part of the cab is reduced by 35K. The average temperature difference between the front and rear of the cab is decreased by 25K, the cooling effect of air conditioning is improved.

Keywords: air conditioner, air inlets, flow field, fluent, temperature field

1 Introduction

Nowadays, cars have become the means of transportation for people's lives. With the rapid development of automotive technology, science and industry, people's demand for warm environment of vehicle cab is also increasing by [7]. The indoor thermal environment directly affects the mental state of drivers and passengers. A good car compartment thermal environment can improve people's comfort, reduce driving risk coefficient, and meet the national energy saving and emission reduction needs.

In the traditional design of automobile air conditioner system, due to its complicated structure, it is difficult for the air flow to reach the uniform distribution, and the ventilation depends on the continuous operation of the air conditioner system, in addition, the traditional design only consider the overall cooling (heat) and air supply, however, it didn't do the precise design of the cab's distribution of air flow. Sun Yining used the weighted PMV index to analyze the effect of the different layout of the air outlet on the thermal comfort of the human body, with to partition the vehicle passenger compartment space [8]. Based on the analysis of typical working conditions in summer, Yue Lei used comfort control method to reasonably combine air supply parameters, and improved the thermal comfort of passenger compartment without increasing energy consumption [1]. Zhang of South China University of Technology set up the evaluation method of thermal sensation and thermal comfort based on the skin temperature of the human body, and carried out the field environment research for the crew cabin [9]. Wang used Fluent to simulate the influence of different air supply temperature and speed on the indoor speed and temperature field during the cooling process of automobile air conditioning [2].

This paper takes a sedan as the research object, initial model is established according to the real vehicle dimensions, using FLUENT to solve the model and analysis, through the experiments, the

* Corresponding Author

accuracy of model is validated. Considering the solar radiation wall heat transfer, natural convection heat transfer and fluid solid coupling heat transfer, we studied the influence of different air outlets on the temperature field and flow field in the cab. For the design of automobile air conditioning, choose the appropriate number of air supply outlets for reference.

2 The Establishment of the Original Structure Model

Taken a sedan as the research object, a model based on the parameter of the real automobile is established. Because the temperature field and flow field in the cab are mainly studied, in the process of modeling, the front view mirror, the rearview mirror, wheels, door handles are ignored, the dash board, pedal and other automotive interior components are ignored, simplified the structure of the interior of the cab, only retaining seats, which has a great influence on the internal flow structure. In the model, there are three air inlets installed on the front of the instrument panel and two air outlets under the head of the cab. The main dimensions of the cab are shown in Table 1. The main dimensions of the model are shown in Fig. 1.

Table 1. The main dimensions of the cab parameters

	Size (mm)
External dimensions (length*width*height)	2550*1350*1250
Left side inlet (width * height)	70*90
Central inlet (width * height)	90*70
Right side inlet (width * height)	70*90
Outlet (width * height)	120*100

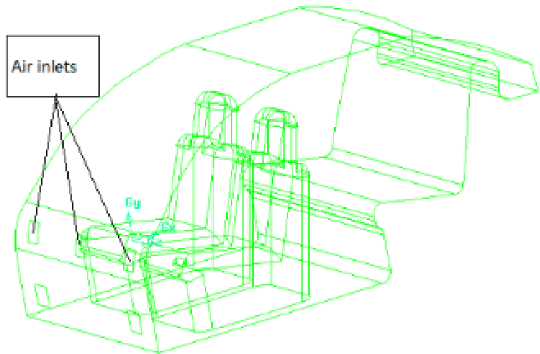


Fig. 1. The original structure model

2.1 Grid Division

In this paper, the cab model is a complex three-dimensional model, so the method of dividing the grid by using the method of semi-structured grid is selected, then, appropriate adjustments to the grid, such as the position of the air inlets and the air outlets, region of geometric catastrophe, the area around the seat, the processing of the near wall region. The grid division is shown in Fig. 2, the number of grids is 1891042892.

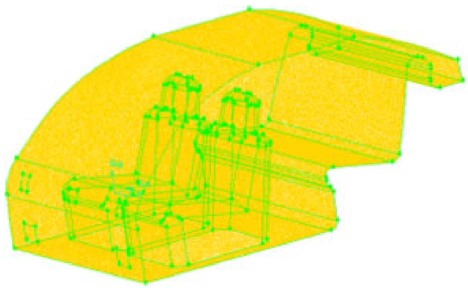


Fig. 2. Cab model for semi structured mesh generation

2.2 Selection of Turbulence Model

Due to the complexity of the turbulence itself, unless the DNS technology is realized, the turbulent flow problem can not get the exact numerical solution. Among the numerous turbulence models proposed by the academia, no model is suitable for various kinds of flow phenomena. When choosing turbulence model, the physical problems in the flow, the limitation of computational resources, the requirement of accuracy, and the model solving time constraints should be taken into account. The flow in the driving room is three-dimensional turbulent flow, with the vortex. The k-ε model equation in the simulated driving indoor velocity field and temperature field test results are very accurate, RNG k-ε turbulence model was selected. By comparing with the standard k-ε model, the model considers the rotation and rotational flow in the mean flow. The RNG k-ε mode can handle the large high strain rate and streamline bending degree of flow better, which is more accurate than the standard k-ε model [3, 10].

2.3 The Boundary Condition of Inlet

Define the air inlet as velocity-inlet, which is used to define other scalar type flow variables related to the flow velocity in the flow entrance, and the wind speed is 2m/s, the air supply temperature is 16°C (289.15k). Turbulence intensity by the following modified empirical formula (1):

$$I = \frac{\mu'}{\mu_{avg}} = 0.16(R_{eD_H})^{-1/2} . \quad (1)$$

In the formula: u' is the root mean square value of the velocity fluctuations, u_{avg} u_{avg} is the average velocity, R_{eD_H} is the Reynolds number when the equivalent diameter is calculated [4].
Hydraulic Diameter (D) calculation formula for the equivalent diameter of the air inlet is as follows:

$$D = 4A/S . \quad (2)$$

In the formula: A is the flow area of the air inlet, is 6300 mm^2 and the S is the wet week length, is 320 mm.

According to the above formula, the diameter of each inlet can be calculated, and the diameter is 0.07875 m.

2.4 The Boundary Condition of Outlet

In this paper, the boundary of the outlet is defined as pressure-outlet, the outlet pressure is the external atmospheric pressure, and that is, the relative pressure is zero. The parameters of the outlets are set to be processed by the local individual. In order to use this method in the calculation, there should be a certain distance between the exit section and the calculation area, and there is no reflow in the outlet section.

The equivalent diameter calculation method is used to avoid the backflow phenomenon [5]. The calculation formula is shown in (3):

$$d_i = 2ab/(a+b) . \quad (3)$$

In the formula, d_i is the equivalent diameter, a is the length of return air inlet, is 120 mm. b is the width of return air, is 100 mm. According to the formula (3), the hydraulic diameter of the return air outlet is 0.1090909 m.

2.5 Wall Condition

Due to the complex of the structure in the cab, simulate each wall is a big difficulty. It is necessary to sort the walls so that we can put the same wall together. And it can simplify the heat transfer. Although there is a certain difference between the simplified walls is different from the actual situation, the effect is close; therefore, we can obtain the ideal simulation results. The wall surface is simplified and the following wall are obtained and set to the wall. The wall heat transfer mode and wall temperature settings are shown in Table 2.

Table 2. The cab wall boundary conditions

Wall	Mode of heat transfer	Wall temperature
roof of the cab	fixed	338K
floor of the cab	fixed	338K
side of the cab	fixed	338K
surface of the front seat	conduct	338K
surface of the rear seat	conduct	338K

2.6 Numerical Iterative Calculation

In this paper, it is assumed that the fluid in the cab is unsteady flow, each time step update will lead to a large residual, so the residual curve is bound to be a curve of repeated shocks. Because the model is explicit iteration, the time step is smaller, the precision is higher, and not easy to spread, but the total calculation time will be longer. The inner iteration may not need too much, the default iteration step number is 20. The time step is set to 0.02s, the velocity field and temperature field of the whole model in 200s are calculated for 72 hours. The residual diagram of simulation as shown in Fig. 3. The residual error of the momentum equation to reach the order of 10-3 magnitude, the residual error of continuity equation (mass conservation equation), energy equation and DO equation are less than 10-3, then, can be regarded as convergent.

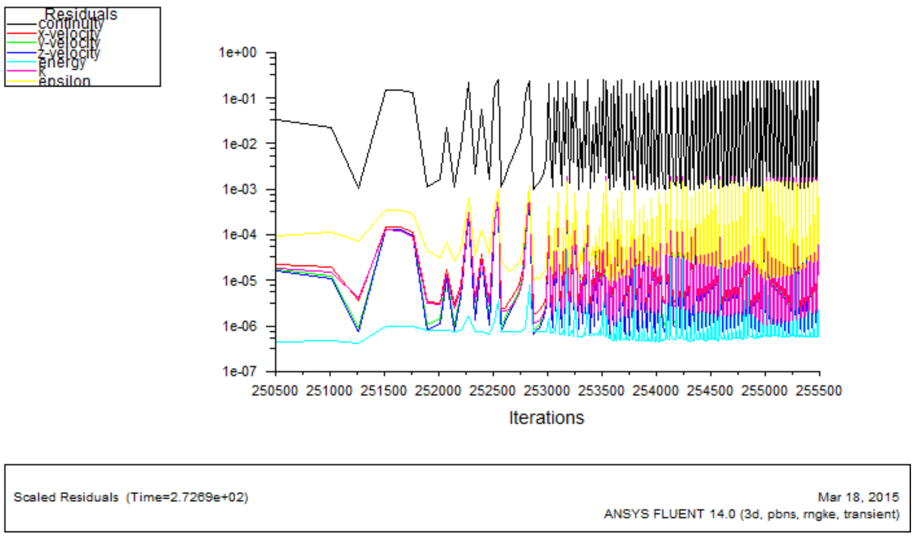


Fig. 3. Residual diagram of simulation

3 The Solution and Verification of Model

3.1 Solution of the Model

The temperature changes directly affects the driver's comfort, since the driver's head is the most sensitive to the changes in heat or cold [6], we selected the position of driver's head as the research point, and then obtained the temperature change curve of the research point in 250s, as shown in Fig. 4. The temperature of this point begins to decrease by 338K, the cooling rate is relatively fast, this is because the air inlet changes the airflow characteristics in the interior of the car and rapidly reduces the air temperature near the air outlet. At 50s, there is almost no heat and mass transfer with the air flow in the rear of the car, because the temperature is basically balanced, so the temperature fluctuation is about 320K. With the increase of time, the air flow in the inlet of the air conditioner and the air and air flow in the front and rear parts of the car have heat and mass transfer with each other. The temperature fluctuates greatly and the cooling rate is slow, eventually close to 315K in 200s.

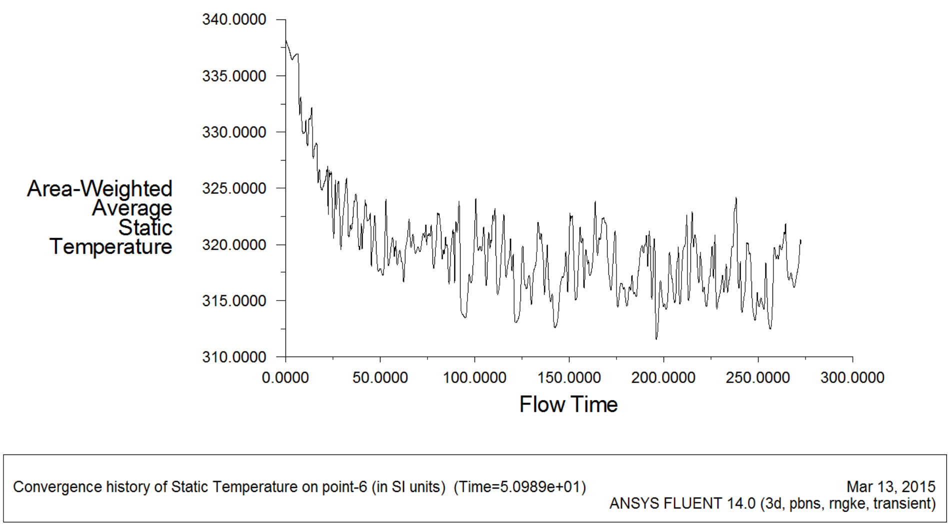


Fig. 4. The temperature change curve of the research point in 250s

3.2 Validation of the Model

According to GB/T12782-2007 Car Heating Performance Requirements and Test Methods, under the provisions of the standard, we should layout corresponding measuring points in the driver's seats, CO pilot and the back seat. Meanwhile, the test should be operated in sunny days with little wind. For the procedures of the test, the engine should be cooled. Before the test, the driving room must have good ventilation. After the test, the doors and windows must be closed, that is the vehicles are under idle condition. After opening the air conditioner, set a fixed value for the interior temperature, at the same time, record the test data. After the test, the experiment data should be analyzed. A black sedan is selected as the experiment object, and the cab's seats are dark cortex to minimize the impact of color and material on vehicle temperature and cooling. Meanwhile, the test vehicle should be cleaned before the test, including the cleaning of the windows, body, and removal of flammable and explosive materials in the car. Firstly, we checked the vehicle sealing performance, and then test the vehicle to preheat temperature, the specific methods and requirements of preheating stage as shown in Table 3.

Experimental environment: open-parking ground; outdoor temperature is 35°C; air humidity is 57%.

Experimental instruments: Ipetronik temperature measurement module; CAN/LIN Vector data recorder; data acquisition; Rossel temperature sensors and other instruments.

Table 3. The specific methods and requirements of the heating up stage

Stage name	Vehicle condition	Requirement
Preheating stage I	Door status: fully open Window status: fully open Evaporator fan status: off Engine status: off Car personnel: no	10 minutes
Preheating stage II	Door condition: close Window status: fully open Evaporator fan state: medium speed Engine status: idle Car personnel: no	10 minutes
Heating up stage	Door condition: close Windowstatus: close Evaporator fan status: off Engine status: off Car personnel: no	20 minutes

Test vehicle in accordance with the above method to complete the heating up, and to ensure that the head of the crew average temperature of 60°C to 65°C, then can do the experiment. In the process of cooling test, air conditioner system is required to adjust to the state of maximum cooling capacity, that is, air volume reached the maximum, and meanwhile, we set the temperature at 16°C on the air conditioner, and record the temperature of the vehicle air outlets and the driving position.

As for the simulation model, the temperature change frequently. So, there are many record points, however, it is not that significant to the real situation. It is necessary to select the average temperature instead of too many points to draw the curve. The test curve and the simulation model curve are shown in Fig. 5, the starting point of the simulation model and the test vehicle are 338K or so, the temperature curve of the two are similar in 250s, the overall trend of the curve is first fast and then slow in 250s, either the model or the test car, the driver's head temperature has been reduced to about 315K. However, there are still many differences in the change of temperature among simulation model and experimental results. The first reason is that, the time step in the simulation model is 0.01s, thus, the changes of temperature is more specific, and Fluent can record the extremely tiny changes of temperature, it is more sensitive to outside environment. However, in the experiment process, the time step was extended to 1s, which is 100 times to simulation. Thus, the data obtained in the test is even less than the simulation, and the sensor can not react to each small changes of temperature, that is, the sensitivity of sensor is lower than Fluent. Secondly, simulation model is an idealized assumptions and simplifications of real cab model, so, there are many difference between the simulation model and the real experiment results.

The results show that the simulation model can be used to analyze the flow field and temperature field of a certain car cab and the simulation model has high reference value.

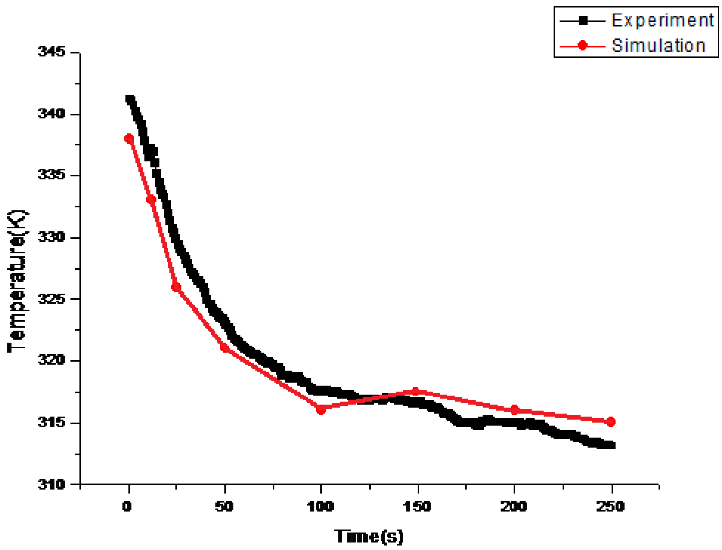


Fig. 5. The temperature change curve of experimental model and the driving in 250s

4 The Establishment of Improved Structure Model

In order to improve the airflow distribution, the cooling rate can be increased by increasing the number of air inlets. Because the front end of the cab cooling is faster than the back, therefore, we selected the choice of adding two symmetrical air inlets at the rear part, and the structure is easy to realize. Under the premise of the original model air inlets' size is unchanged, adding the size of 70*90 mm air inlets on the B column position on the left and right sides of the car, keep the size of the outlet to the same position, as shown in Fig. 6. The improved geometric model still adopts the method of semi-structural mesh to partition and adjust the model.

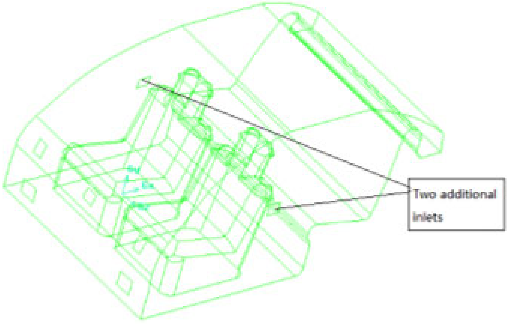
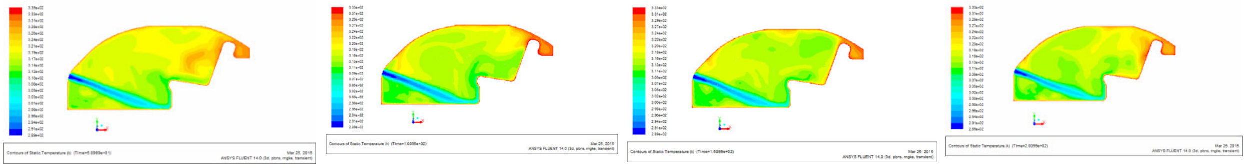


Fig. 6. Improved structure model

5 Comparative Analysis of the Flow Field between the Original Structure and the Improved Structure

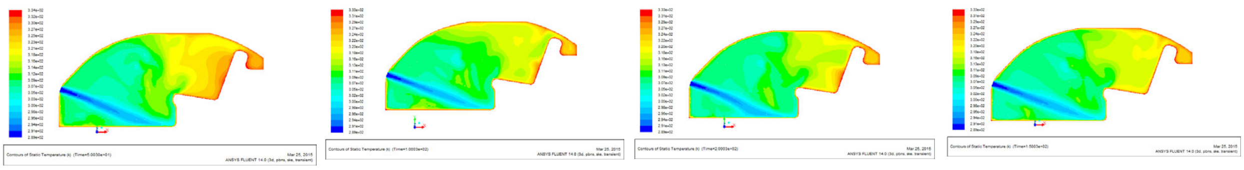
The model is simulated in the condition of both the wind speed and the air supply temperature are fixed value, the wind speed is set about 2 m/s and the supply air temperature is about 16°C (289.15k), so the distribution of flow and temperature field in the cab can be obtained. The 50s, 100s, 150s and 200s moments are selected, and then the flow field and temperature field distribution were analyzed respectively. To clearer show, we selected the Z=0mm plane of the XY plane, the Z=500mm plane of the XY plane (Close to the driver's position) and the X=450mm plane of the YZ plane (the vertical section of the front of the seat back) to analysis the flow field and temperature field [11]. Taken above plane as I, II, III plane to simplify the following description.

Fig. 7 and Fig. 9 is the distribution of temperature and velocity fields at different moments in the I section of the original structure. Fig. 8 and Fig. 10 is the distribution of temperature and velocity fields at different moments in the I section of the improved structure.



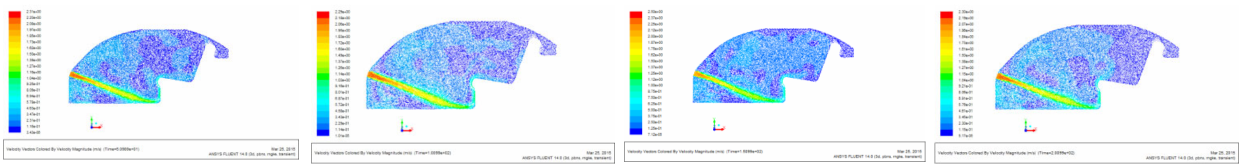
(a) t=50s (b) t=100s (c) t=150s (d) t=200s

Fig. 7. The I section's temperature field distribution of the original structure



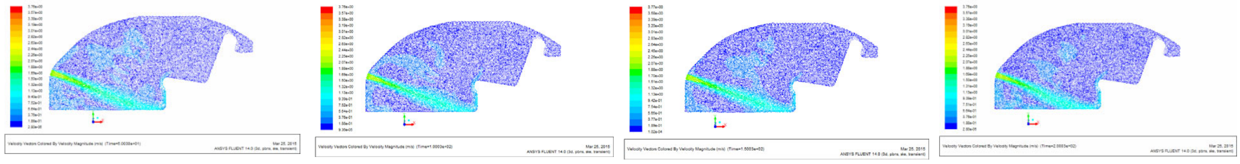
(a) t=50s (b) t=100s (c) t=150s (d) t=200s

Fig. 8. The I section's temperature field distribution of the improved structure



(a) t=50s (b) t=100s (c) t=150s (d) t=200s

Fig. 9. The I section's velocity field distribution of the original structure

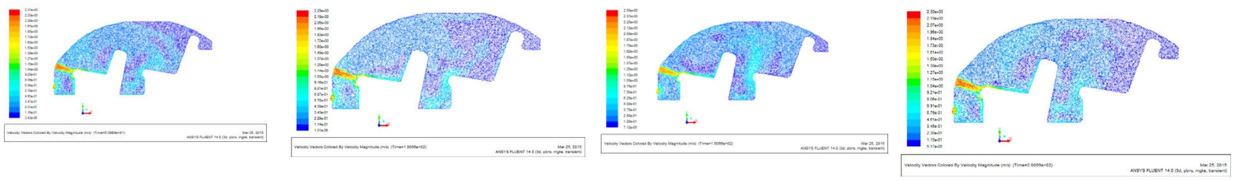


(a) t=50s (b) t=100s (c) t=150s (d) t=200s

Fig. 10. The I section's velocity field distribution of the improved structure

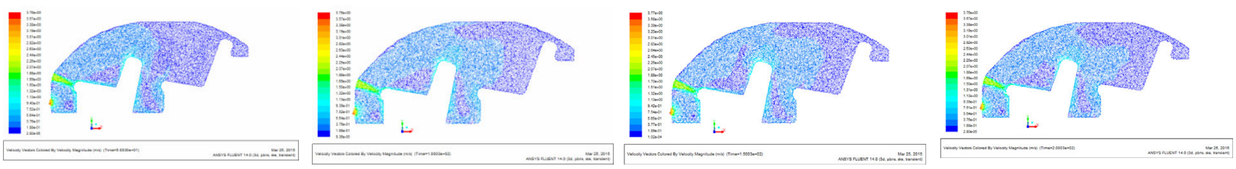
The air flow rate is 0 m/s in the whole cab at the time of 0s, the air flow rate of the air supply position is close to 2 m/s. Due to the setting of the model, the temperature of each position in the driving room is 338K. As shown in Fig. 7, the cab's temperature dropped by 20K, the temperature dropped to 318K in most of the regional in 200s. The air speed reduced gradually from front to back can be seen from Fig. 9, thus, the temperature of the chamber is also showing that the front is slightly lower, the rear is slightly higher, so that the temperature is always not less than 325K in the rear seat back area in 200s, the reason is that the three air inlets concentrated at the front part of the cab. The cold wind blows from the central air inlet, through the gap between the front two seats into the back seat of the foot position directly to speed up the cooling rate of the position. By comparing with the original simulation model, the fluid velocity is obviously increased through improved. The air flow rate at the front part of the cabins slightly higher, about 0.939m/s. Distribution of the flow field directly affects the distribution of temperature field. As shown in Fig. 8, the cab's front cooling situation is very well at 200s, and cool down to 305K in most areas of the front, and the back of the cab area to cool down to 316K, there is a certain temperature difference between the front and back of the cab. Compared to the original, in t=5s and t=200s, the improved cab 1 section temperature field and velocity field were significantly lower, the velocity field and temperature field in the rear compartment was good, the temperature decreased from 20K to 33K, this is because the increase in the B column of the two air inlets. The temperature field and the velocity field change in a relatively short period of time before and after the car room. At the same time, when the number of inlet air inlet is increased, the intake flow rate is also increased, so it can improve the comfort of the car room faster.

Fig. 11 and Fig. 13 is the distribution of velocity and temperature fields at different moments in the II section of the original structure. Fig. 12 and Fig. 14 is the distribution of velocity and temperature fields at different moments in the II section of the improved structure.



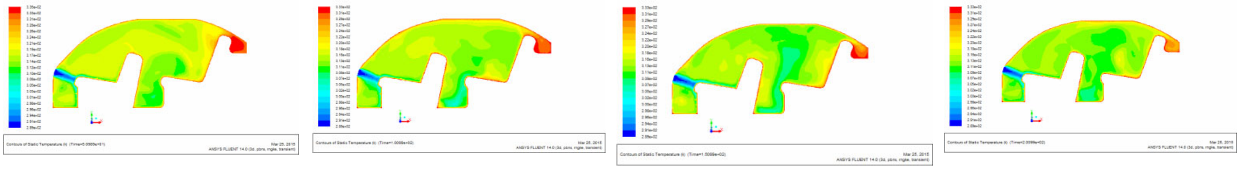
(a) t=50s (b) t=100s (c) t=150s (d) t=200s

Fig. 11. The II section's velocity field distribution of the original structure



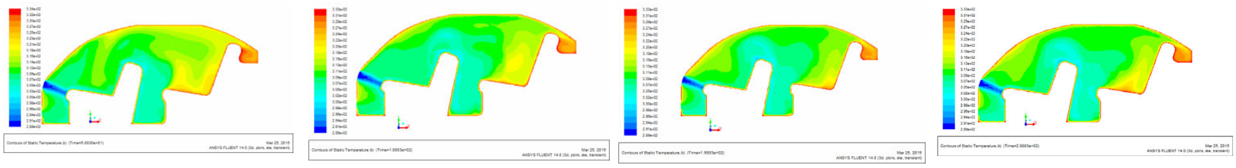
(a) t=50s (b) t=100s (c) t=150s (d) t=200s

Fig. 12. The II section's velocity field distribution of the improved structure



(a) t=50s (b) t=100s (c) t=150s (d) t=200s

Fig. 13. The II section's temperature field distribution of the original structure

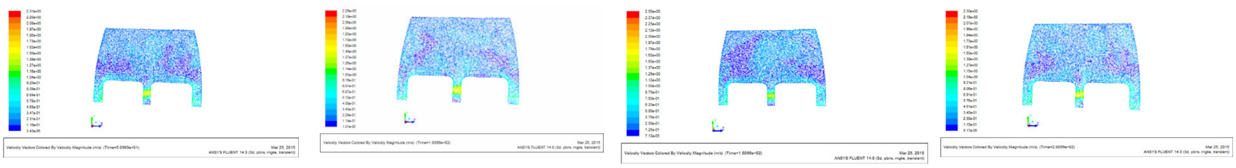


(a) t=50s (b) t=100s (c) t=150s (d) t=200s

Fig. 14. The II section's temperature field distribution of the improved structure

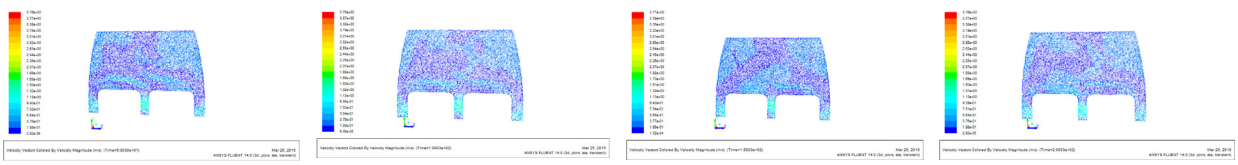
The air speed near the surface of the front seats is larger can be seen from Fig. 11, and the temperature is also relatively lower can be seen from Fig. 13, this is mainly due to the II section is located near the driver's position. Meanwhile, from the velocity field and temperature field distribution of the II section, it can be seen that the air speed in the rear seat is large and the temperature is low. Fig. 12 shows that the air flow rate in the front seat space reached 1m/s through improving, because the cold wind blowing in front of the air inlets are blocked by the seat, increasing the air flow around the seat and the two inlets at the left and right of the B column mainly winds the cold air to the front of the cab, the air flow rate of the area is larger. Therefore, the front space's temperature cooling well can be seen from Fig. 14, at 150s, the temperature dropped to 300K, while the rear space temperature only about 310K, that is, the temperature of the front and back is not balanced. Compared with the previous improvement, when t=100s and t=150s, the temperature field and velocity field near the front seat of the II section of the improved cab were lower, which improved the comfort of the cabin personnel in the cockpit.

Fig. 15 and Fig. 17 is the distribution of velocity and temperature fields at different moments in the III section of the original structure. Fig. 16 and Fig. 18 is the distribution of velocity and temperature fields at different moments in the III section of the improved structure.



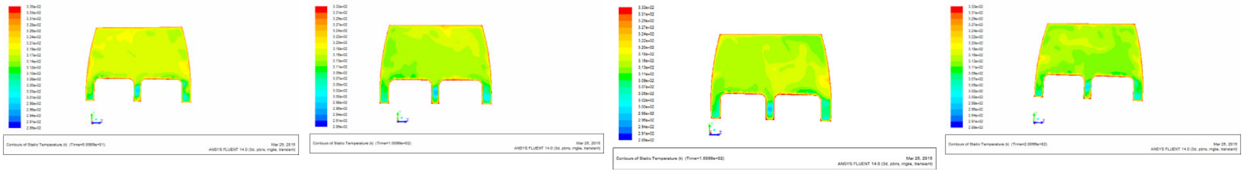
(a) t=50s (b) t=100s (c) t=150s (d) t=200s

Fig. 15. The III section's velocity field distribution of the original structure



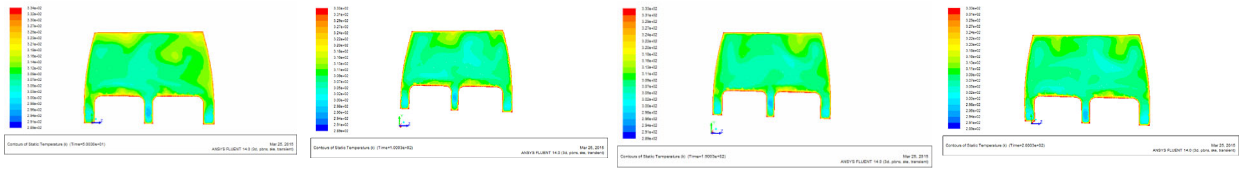
(a) t=50s (b) t=100s (c) t=150s (d) t=200s

Fig. 16. The III section's velocity field distribution of the improved structure



(a) t=50s (b) t=100s (c) t=150s (d) t=200s

Fig. 17. The III section's velocity field distribution of the original structure



(a) t=50s (b) t=100s (c) t=150s (d) t=200s

Fig. 18. The III section's temperature field distribution of the improved structure

Through the analysis of section III's velocity field and the temperature field, the air flow rate below the front seat was significantly higher than that of the other regions, so it makes the temperature lower than that of other regions. This is mainly up to the three air inlets send wind to this region directly, and the effect of cooling is obvious. As shown in Fig. 16, at the time of 150s, the air flow rate is about 1m/s through improved, the temperature has dropped to about 310K, as can be seen from Fig. 18, and the effect of cooling is very well. Similarly, the air flow rate below the two sides of the seat was significantly lower than that of the other regions, and it's also due to the three air inlets send wind to this region directly. After improvement, due to the addition of two air inlet on both sides of B column and the larger air flow part into the cockpit, the air velocity and temperature in cockpit are closer to the perceived comfort of human body.

6 Conclusion

Through the simulation and analysis, the following conclusions are drawn:

By comparing the experimental results with the simulation results, in the 0-250s time period, within the reasonable error range, the two all decreased from about 338K to 315K, and the trend of temperature change is the same. Therefore, the geometric model has a high reference value.

Under the condition of constant air supply speed, air supply temperature and air flow rate, two air outlets on both sides of the B column are built on the basis of the three air outlets on the front cab. The 1 plane between the main driving position and the side driving position, the 2 plane near the driving air outlet and the3 plane of the vertical section of the seat back to the back of the 50mm are selected. Through the analysis of flow field and temperature field, in the 0-200s time period, the average temperature of the front part of the cab is reduced by 35K, the average temperature of the rear part is decreased by 25K, there is still a certain temperature difference between the front part and the rear part of the cab, and the cooling range is obvious.

Acknowledgements

The research is a project supported by General Project of Science and Technology Project of Beijing Municipal Education Commission (SQKM201610016017).

References

[1] J-Y. Wang, C-T. Xue, X-J. Hu, Simulation of airflow in passenger compartment based on air-conditioning supply parameters, Journal of Jilin University 47(2017) 50-57.

[2] R. Wang, W. Li, B. Zhu, Impact of air supply parameters on velocity field and temperature field of vehicle interior, Agricultural Equipment and Vehicle Engineering 2(2016) 1673-3142.

[3] J-F. Jin, T. Zhu, Y-L. Li, Numerical simulation of inner flow for air-conditioned bus using computational fluid dynamics, China Mechanical Engineering 18(13)(2007) 1591-1594.

[4] C-Q. Zhang, X-M. Liu, G-Q. Meng, Analysis on resistance for bus/coach intake system with fluent, Bus & Coach Technology and Research 1(2015) 35-37.

[5] J-P. Chen, Y-M. Zhu, J-Y. Mu, Computational fluid dynamic analysis of the car air-conditioning duct, Automotive Engineering 24(2)(2002) 134-136.

[6] A. Alahmer, M. Abdehamid, M. Omar. Design for thermal sensation and comfort states in vehicles cabins, Applied Thermal Engineering 36(1)(2012) 126-140.

[7] H. L. Liu, Analysis and optimization on thermal comfort of vehicle cabin, [dissertation] Changchun: Jilin University, 2017.

[8] Y. N. Sun, Evaluation and analysis of thermal comfort on vehicle HVAC, [dissertation] Changchun: Jilin University, 2012.

[9] W. C. Zhang, Study on the key technology of thermal environment and occupant's thermal comfort analysis in vehicle cabins, [dissertation] Guangzhou: South China University of Technology, 2013.

[10] W. G. Zhang, Optimal design of vehicle air condition Base on thermal comfort of human body, [dissertation] Changsha: Hunan University, 2012.

[11] H. Y. Song, The matching and simulation analysis of electric vehicle air-condition system based on fluent, [dissertation] Wuhan: Wuhan University of Technology, 2012.