Numerical Simulation and Analysis of Air Duct of Microbar Air Quality Conditioning

Hongliang Wang1,*, Pidong Shi2

1Shandong Vocational College of Science and Technology, Weifang 261053, Shandong, China
2Zhongtong Bus Holdings Co. Ltd, Liaocheng 252000, Shandong, China

*Corresponding author: hongliangwang@sdvcst.edu.cn

Abstract. A microbar air conditioning duct was taken as the research object, and the flow field characteristics of the air conditioning duct were numerically simulated based on CATIA and ANASYS software. According to the numerical simulation results, the pressure field and velocity distribution of the air duct were analysed in detail. The analysis results show that the increase of the inlet velocity of the left air duct is conducive to the increase of the velocity of the air outlet, but the flow resistance is increased. By changing the inner wall structure and inlet area of the air duct on the right side, the wind speed of the rear air outlet can be improved effectively.

Keywords: Microbar air conditioning duct, Numerical simulation, Pressure field, Velocity field.

1. Introduction

Riding comfort, as one of the main performance of automobiles, has become a key index in the evaluation of vehicle performance by consumers. Its main role is to ensure that automobiles have good noise reduction and shock absorption effect, and to provide a high-quality riding environment [1]. It is mainly based on people's subjective evaluation, namely human comfort. With the continuous improvement of people's living standards, people's requirements for travel experience are gradually getting higher [2]. The comfort of vehicle air conditioning system is becoming more and more important in automobile design and gradually becomes the main means to improve the competitiveness of the automobile market [3]. Air conditioning duct is an important part of automobile air conditioning system. Air duct layout, air duct occupied space (cross-sectional area), wind speed, temperature distribution, flow resistance and other parameters [4, 5] will directly affect the performance of air conditioning system, and further determine the air flow field inside the car, thus determining ride comfort [6].

Currently barye vehicle air conditioning problems after the right side of the duct section of export wind speed is low, the specific conditions as follows: when the normal operation of air-conditioning fan (air conditioning) 75% of the maximum speed conditions, the passenger door at the back of the outlet wind speed is small, compared with the target performance of 3 m/s velocity is large, the serious influence passenger door at the back of the passenger area space temperature regulation, affect the passengers ride comfort. It is costly and time-consuming to explore the flow field characteristics of air conditioning duct.
duct by experimental methods. Therefore, computational fluid dynamics method is adopted to study the performance of air duct.

2. Geometric model construction and mesh division

2.1. Geometric model construction

The geometric model of this study includes the left air duct assembly and the right air duct assembly. CATIA and Anasys geometric modules are used to simplify and repair the model [7]. Among them, the actuating mechanism of the slide door in the right air duct is wrapped, and the influence of its shape structure on the air flow characteristics is briefly studied, as shown in Figure 1. Flow passage extraction was performed on the geometric model for numerical calculation. The fluid domain model is shown in Figure 2.

![Figure 1. Geometric model of air duct.](Image)

![Figure 2. Fluid domain model.](Image)

2.2. Mesh division

The research model is meshed, and the near wall surface is divided into five layers of grids. The total number of grids of all models is more than 700000. The grid independence verification shows that the number of grids meets the accuracy of engineering problem solving. When the number of grids is 300000, the difference between the outlet flow velocity and the total number of grids is small, so it meets the requirements of numerical simulation.

![Figure 3. Fluid domain grid.](Image)

3. Theoretical model and boundary condition setting

According to the three equations of fluid motion, the mathematical model of fluid flow in air conditioning duct is established. The continuous phase is air and is assumed to be an ideal incompressible fluid.
k equation:

\[
\rho \frac{\partial}{\partial x_j} (u_j k) = \frac{\partial}{\partial x_j} \left[ \left( \frac{\mu_t}{\sigma_k} + \mu_1 \right) \frac{\partial k}{\partial x_j} \right] + \frac{\partial u_j}{\partial x_j} \left( \frac{\partial u_i}{\partial x_i} + \frac{\partial u_i}{\partial x_i} \right) - \rho \varepsilon
\]  

(1)

\varepsilon equation:

\[
\rho \frac{\partial}{\partial x_j} (u_j \varepsilon) = \frac{\partial}{\partial x_j} \left[ \left( \frac{\mu_t}{\sigma_k} + \mu_1 \right) \frac{\partial \varepsilon}{\partial x_j} \right] + \frac{\varepsilon}{k} \frac{C_1}{C_2} \frac{\partial u_j}{\partial x_j} \left( \frac{\partial u_i}{\partial x_i} + \frac{\partial u_i}{\partial x_i} \right) - C_2 \rho \frac{\varepsilon^2}{k}
\]  

(2)

Where, \( \rho \) is the air density; \( \mu_t \), \( \mu_1 \) is the laminar viscosity coefficient and turbulent viscosity coefficient; \( k \), \( \varepsilon \) is the turbulent kinetic energy and dissipation rate, \( C_1 = 1.44 \), \( C_2 = 1.92 \), \( \sigma_k = 1 \), \( \sigma_\varepsilon = 1.3 \), \( C_\mu = 0.09 \).

The inlet velocity is set as 1.6m/s (90% air conditioning fan speed), 1.4m/s (75% air conditioning fan speed) and 1.2m/s (65% air conditioning fan speed) according to the research conditions. The inlet velocity of the right air duct is simulated according to the maximum speed of 75% air conditioning fan, i.e. 1.4m/s. The boundary of the air outlet is set as free outlet, and the rest of the wall is assumed to be smooth and no slip. The operating environment is standard atmospheric pressure, and the relative atmospheric pressure of the initial pressure setting is 0.

4. Analysis of simulation results

4.1. Analysis of flow field characteristics of left air conditioning duct

The flow field of left air conditioning duct is simulated under three conditions, which are inlet velocity 1.6m/s (90% air conditioning maximum speed), 1.4m/s (75% air conditioning maximum speed) and 1.2m/s (65% of the maximum air conditioning speed). The distribution of the flow field characteristics in the left air conditioner is obtained and compared with the experimental results.

4.1.1. Pressure field analysis. It can be seen from the pressure nephogram of the left air conditioning duct outlet in Figure 4 that with the decrease of inlet velocity, the overall static pressure level of the rear section of the air conditioning duct gradually decreases, and the pressure gradient dividing line moves backward. At the same time, it can be seen that the pressure gradient of the front section changes greatly under the three working conditions, indicating that the pressure potential energy decreases greatly. From the overall pressure nephogram of the left air conditioner under the three working conditions in Figure 5, it can be clearly seen that the static pressure decreases greatly from the inlet of the air duct to the front end of the straight pipe of the air duct, and in the front end of the installation slot of the lead screen, the static pressure shows an increasing trend, because the air flow from the front end moves to the installation slot, the air flow movement is blocked, the dynamic pressure decreases, and the static pressure increases. Due to the narrow passage at the rear of the installation slot, the dynamic pressure of the air flow increases, which leads to the further decrease of the static pressure. The air flow passage at the rear of the installation slot becomes larger, which leads to the increase of the static pressure of the air flow and the decrease of the dynamic pressure.
4.1.2. Velocity field analysis. It can be clearly seen from the middle surface velocity cloud picture of the air conditioning outlet on the left side of Figure 6 that with the increase of the inlet velocity, the overall velocity gradient change rate is small, and the velocity level in the middle of the section gradually increases, resulting in a relatively large air velocity at the middle of the section, which indicates that the increase of the inlet velocity is conducive to the increase of the outlet velocity.

The distribution of air flow in air conditioning duct can be seen through the trace diagram of the left air conditioning air duct main view (see Fig. 7) and the top view of the left air conditioning air duct (see Fig. 8). It can be seen from the main view that the trace shows spiral state at the branch of the pipeline, indicating that there is a large range of eddy current in the pipeline, and it continues to the front end of the air duct in the driving area; From the top view, it can be seen that with the increase of inlet velocity, there is a large area and large velocity reflux phenomenon in the upper part of the straight air duct, which will easily lead to the increase of the resistance of airflow movement. At the same time, the flow velocity at the corner also increases gradually, which leads to the increase of local pressure loss at the corner. Therefore, it is necessary to optimize the structure of the left air duct.
4.2. Analysis of flow field characteristics of right air conditioning duct
There is a sliding plug door actuator in the air duct of the right air conditioner, so the air flow in the air duct is complex. When the inlet velocity is 1.4m/s, the outlet velocity of the rear section of the air duct is small, which cannot meet the performance requirements. Through numerical simulation, the flow field of the right air conditioning duct in the fully closed state of the actuator is analyzed, and the factors influencing the air outlet velocity in the rear section of the air conditioning duct are explored.

4.2.1. Analysis of pressure field of fully enclosed actuator. It can be seen from the pressure nephogram of the right air-conditioning pipe in the fully closed state in Figure 9 that the overall pressure of the straight pipe presents a negative pressure state, and the pressure difference between the front end and the rear end of the plug door actuator is large, which indicates that the plug door actuator has a large local pressure loss in the overall pipe, resulting in a large overall pressure drop of the air-conditioning pipe.

4.2.2. Velocity field analysis of fully closed executive structure. Fig. 10 shows the middle velocity nephogram of the air-conditioning duct in the straight section and the velocity nephogram of three cross sections at different positions. Through the middle velocity nephogram, it can be seen that there is a large range of low-speed air flow area before and after the plug door actuator, and the air flow velocity distribution is uneven, which indicates that there is an air flow separation area, and it is easy to produce vortex, thus affecting the velocity at the outlet. It is easy to see that the front outlet velocity is less than the rear outlet velocity. From the following three cross-sectional velocity nephogram, it can be seen that the air flow can flow into the rear air conditioning duct from the narrow and long channel outside the actuator, and most of it flows from the upper right corner of the actuator. If the baffle area is properly reduced, it is more conducive for the air flow to enter the rear air conditioning duct. The velocity of the...
air outlet at the rear of the actuator is more than 3m / s, because the actuator compresses the air duct space of the air conditioner, at the same time, the number of outlets is relatively small, and the air flow area is small. Therefore, compared with the left air duct, the outlet velocity at the same inlet flow is relatively larger.

Figure 10. Velocity nephogram of right air conditioning duct in fully closed state.

Figure 11 shows the numerical simulation results of the outlet wind speed of the right air conditioning duct in the fully closed state. It is found that the overall outlet speed is larger, and the air outlet speed in the driving area is greater than 3.5m/s. The maximum wind speed is located at the front end of the air duct in the driving area, because the rear actuator is blocked and the air flow resistance is large. When the air flow moves to the intersection of the pipes, the flow into the air duct in the driving area increases, Thus, the speed of the air outlet in the driving area is increased.

Figure 11. Numerical simulation results of outlet wind speed of right air conditioning duct under fully closed condition.
5. Conclusion

Through the numerical simulation of the left air conditioning duct, it shows that the increase of the inlet velocity is conducive to the increase of the outlet velocity, but the flow resistance increases, which needs further optimization. Through the analysis of flow field characteristics and flow rate, it is concluded that by changing the inner wall structure and inlet area of the air duct, the air speed at the rear outlet can be improved. At present, although it is found that the structure of the outlet wind speed can be improved, the relationship between the specific wind speed and the structural change and the degree of correlation need to be simulated in detail. At the same time, whether the air volume of the evaporator itself is insufficient is being considered.

References

[1] Zhang Wen can, Liu Jun. flow field analysis and structure improvement of automotive air conditioning duct [J]. Highway and automobile transportation, 2015 (04): 17 - 20.
[2] Zhang Xian. Structural optimization design of air conditioning duct for bus [D]. Jilin University, 2019.
[3] Zhang Xiaobing, Yuan Yaxiong, Li Jun, Chen Daojiang. Numerical simulation of air conditioning duct and interior flow field of light bus [a]. China Association for science and technology. Science and education first and sustainable development in Western Development: Proceedings of 2000 Annual Conference of China Association for science and technology [C]. China Association for science and technology: China Society of civil engineering, 2000: 2.
[4] Wang Xianmin, Wang Xiaomei. Improvement design of air conditioning duct for light bus [J]. Journal of Hefei University of Technology (NATURAL SCIENCE EDITION), 2007 (03): 357 - 360.
[5] Yang Runze, an Zhengshun. CFD simulation and optimization of a car air conditioning warm air duct [J]. Journal of Hubei Automotive Institute, 2016, 30 (04): 1 - 4.
[6] Wang Dan, Liu Shuangxi, Mou Jiangfeng, Zhang Chuanwen. CFD analysis of automotive air conditioning duct system [J]. Automotive engineer, 2014 (06): 47 - 49.
[7] Gong Shuguang. Basic application and example analysis of ANSYS [M]. Beijing: China Machine Press, 2003: 167 - 175.