Numerical simulation and optimization of the flow field in the passenger cabin of a vehicle

Bingheng Wu*, Lin Chen and Xiayi Yuan

Automotive Engineering Institute, Guanzhou Automobile Group Corporation Limited, Guangzhou, China

*Corresponding author e-mail: wubingheng@gacrnd.com

Abstract. The thermal comfort of the car passenger cabin directly affects the feelings of the driver and passengers. In the case of high ambient temperature in summer, air conditioning cooling is an important means to improve the thermal comfort of the passenger cabin. If the design of the air conditioning outlet is not reasonable, the thermal environment inside the car will not be improved, the passengers will feel uncomfortable and affect the safety of driving, and it will also cause the waste of energy. In this paper, we calculate the flow field distribution in the vehicle by numerical simulation method and optimize the air conditioning vent arrangement of a model, which greatly improves the thermal environment of the vehicle and enhances the thermal comfort of passengers.

Keywords: Numerical simulation; Optimization of the flow field; The passenger cabin of a vehicle.

1. Introduction

With the development of cities and the improvement of living standards, car ownership is increasing, more and more people choose to travel by car, and the time spent in the car is increasing, especially the residents of large cities often commute for more than one hour each way, so people's demand for the thermal comfort of the car passenger cabin is also increasing.

In the case of high ambient temperature in summer, air conditioning cooling is an important means to improve the thermal comfort of the passenger cabin [1]. If the air conditioning outlet design is not reasonable, the thermal environment in the car is not improved, the passengers will feel uncomfortable and affect driving safety. If the air outlet direction and air speed are unreasonable, the passengers in the car can not be adjusted in place through the air conditioning paddles, and can only increase the cooling gear to increase the cooling capacity to improve the thermal comfort of passengers through the overall cooling of the passenger cabin, resulting in low efficiency in the use of air conditioning system cooling, resulting in a great waste of energy.

If the subjective evaluation of human body is used to evaluate the thermal comfort of the passenger cabin, the cost is high, the development time is long, the subjective influence of the evaluation object is large, and the individual difference is large. If CFD (Computational Fluid Dynamics) simulation method is used to make quantitative comparison and optimization in the flow and temperature fields, the development cycle can be visually shortened and the cost reduced [2].

This paper uses computational simulation of the flow field of the passenger cabin to optimize the arrangement and flow distribution of the air conditioning vents so that the cold air in the passenger cabin can cover a larger area of the human body, while taking into account the demand for cold air wind speed in different positions of the human body to optimize the flow field, in allowing the human body to be in a comfortable state.

2. CFD calculation basic equations

We assume that the air in the passenger cabin is an incompressible fluid with a density independent of time and space, and the turbulence model uses Standard k-epsilon model [3].

Continuity equation:

$$\frac{\partial u_i}{\partial x_i} = 0 \tag{1}$$

Momentum equation:

$$\rho u_j \frac{\partial u_i}{\partial x_j} = -\frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j}\left(\mu_t \frac{\partial u_i}{\partial x_j}\right) \tag{2}$$

Turbulent energy equation:

$$\rho u_j \frac{\partial k}{\partial x_j} = \frac{\partial}{\partial x_j}\left(\frac{\mu_t}{\mathrm{Pr}_k}\frac{\partial k}{\partial x_j}\right) + \frac{\mu_t}{2}\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i}\right)^2 + \rho\varepsilon \tag{3}$$

Dissipation rate equation:

$$\rho u_j \frac{\partial \varepsilon}{\partial x_j} = \frac{\partial}{\partial x_j}\left(\frac{\mu_t}{\mathrm{Pr}_\varepsilon}\frac{\partial \varepsilon}{\partial x_j}\right) + C_1 \frac{\mu_t}{2}\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i}\right)^2 \frac{\varepsilon}{k} + C_2\rho\frac{\varepsilon^2}{k} \tag{4}$$

Where u_i is the velocity; ρ is the air density; p is the pressure; k is the turbulent pulsation kinetic energy, ε is the dissipation rate of turbulent pulsation kinetic energy; μ_t the turbulent viscosity coefficient, defined as $\mu_t = C_\mu k^2 / \varepsilon$; C_1, C_2, C_μ, Pr_k, Pr_ε are constants.

3. CFD simulation results

In this paper, the object of study is the internal flow field of the passenger cabin of a model. The geometry of the passenger cabin is processed and the surface mesh is generated using HyperMesh software, and then calculated using Star-CCM+ software. In the Figure 1, part 1 is the roof air outlet. In the Figure 2, part 2 is the central air outlet, part 3 is the air outlet on both sides, and the boundary is designed as a flow inlet, and the optimized solution is to add a compensated air outlet which is part 4 in the panel.

Simulation results of figure 3 and figure 4 show that for the front row dummy, the air volume before optimization is mainly distributed on the outside of the dummy's head and arms, the direction of the air outlet is relatively low. The dummy's head can not blown by cold wind, the subjective feeling of the human body is not good. The optimized solution by adding compensated air outlets above the screen of the panel, the simulation results show that for the front dummy, the air volume is mainly distributed on both sides of the dummy's head and arms, the head can blown by cold wind, and the human comfort is improved. After optimization, the cold air coverage of the air conditioner becomes larger, the flow field of the whole occupant cabin is more uniform, and the cooling rate will be improved.

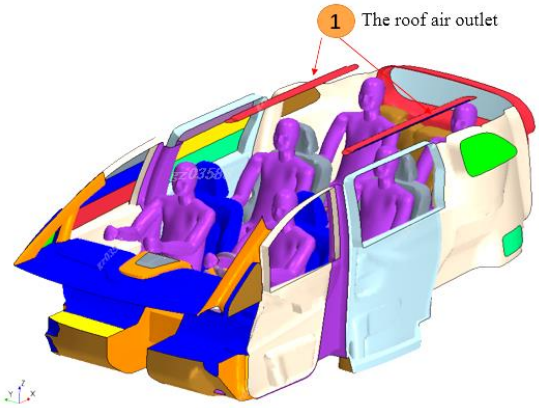


Figure 1. Layout diagram of the passenger cabin

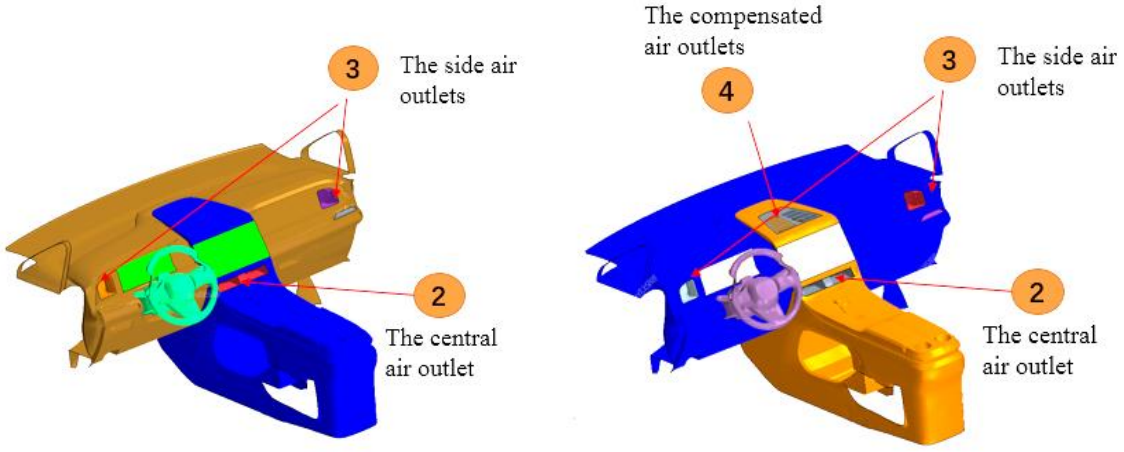


Figure 2. Schematic diagram of the air outlet arrangement (left is before optimization, right is after optimization)

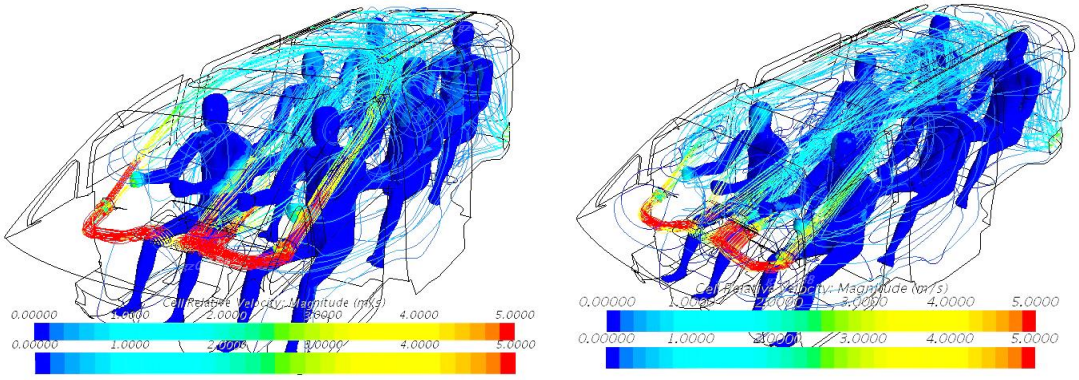


Figure 3. Passenger cabin flow line distribution diagram (left is before optimization, right is after optimization)

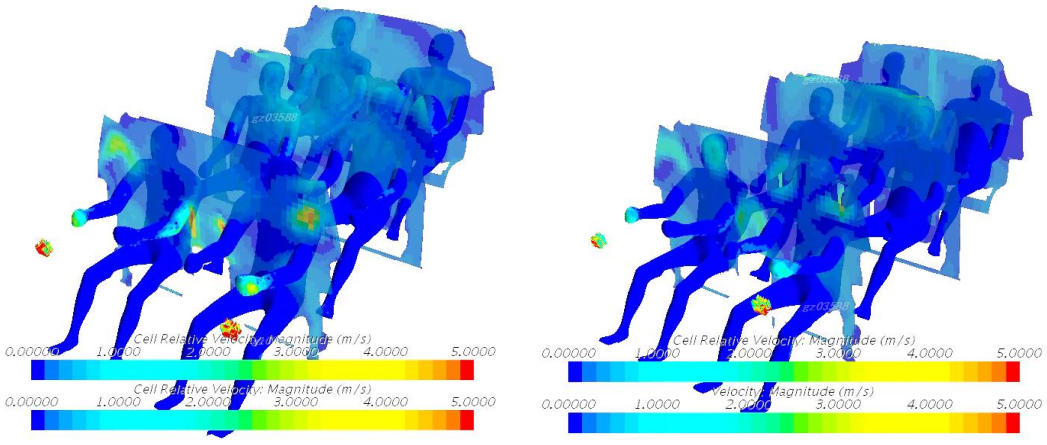


Figure 4. Crew cabin wind speed cloud map (left is before optimization, right is after optimization)

By quantitatively measuring the wind speed in the face, chest, abdomen, left hand and right hand areas of the main driver, as shown in table 1. It can be seen that the cold air of the air conditioner is in the main driver's area, before optimization, mainly concentrated in the main driver's chest, the direction is too low, after optimization, the cold air direction goes up, the wind speed blowing to the head increases, improving the thermal comfort of the main driver.

Table 1. Two Scheme comparing.

Measurement position	Wind speed before optimization(m/s)	Wind speed after optimization(m/s)
head	0.16	0.25
chest	0.36	0.19
abdomen	0.22	0.27
left hand	0.12	0.10
right hand	0.32	0.24

4. Conclusion

In this paper, the flow field of the passenger cabin of a model is modeled and simulated by HyperMesh and Star CCM+ software, comparing the flow field before and after adding compensated air vents on the panel. After optimization, the front passenger's head can blow cold air and human comfort is improved, while the cold air coverage of the air conditioner becomes larger, the flow field of the whole passenger cabin is more uniform, and the cooling rate will be increased, energy will be saved.

References

[1] Y.X. Helian, X.L. Zhang, J. Y. Liu and K.M. Yang. Study on the influence of cooling method on the thermal comfort of passenger car cabin. Journal of shangdong jianzhu university.2020,35(3), pp. 49-55.

[2] X.J. Zheng. The analysis on occupant dynamic thermal response and thermal comfort in vehicle cabin.2018.

[3] A. Parente, C. Gorle, J. van Beeck and C. Benocci. Improved k-e model and wall funtion formulation for the RANS simulation of ABL flows. Journal of wind engineering and industrial aerodynamics.2011,99(4) , pp.267-278.