Optimization of duct structure and analysis of its impact on temperature inside the shelter

Jiawei Lu1, Tao Wang1,*, Liangmo Wang1, Wei Chen2 and Yandong Chen2

1Nanjing University of Science and Technology, Nanjing, China
2Suzhou Jiangnan Aerospace Mechanical and Electrical Industry Co., Ltd., Suzhou, China

*Corresponding author e-mail: Wangtao-me@njust.edu.cn

Abstract. The flow field of the air duct in a shelter was studied using the computational fluid dynamics (CFD) and experiment methods. The improvement methods of vortexes at the bends were analysed. It was found that the corner chamfer and the installation of guide plates have significant effect on improving the vortex. The optimized scheme expanded the inlet and outlet area of the air duct and chamfered the inner bends, then the uniformity of velocity at outlets on both sides of the straight duct was increased by 47% and 42% respectively. The duct was placed in the shelter for simulation. The results showed that the total air volume of the air duct has increased, so the cooling effect is better. The temperature distribution in the shelter is more uniform after optimization, especially in the front part due to the improvement of vortexes.

1. Introduction
The temperature in the vehicle has a very important impact on the comfort of passengers, and the duct undertakes the task of uniform air supply. The reasonable design of the duct structure directly affects the uniformity of air velocity at outlets, then affects the temperature distribution in the shelter. The uniformity is also one of the key factors to be considered in the development of duct products.

Vortexes are easy to occur at bends or sudden changes of cross section in ducts, especially at bends. The flow separation is serious due to inertia, which will cause great pressure loss [1] and destroy the uniformity of the flow field. Tao et al. [2] has proved that adjusting the radius of the bend can reduce the large-scale vortexes. Cheng et al. [3] proposed that the structure with small local drag coefficient, such as divergent or shrinking pipes, should be used to avoid vortexes at variable cross section. In addition to the structure size of the duct itself, it is also an effective method to install a guide plate in the appropriate position. Li Ming [4] and Jianping [5] adjusted the flow path of the airflow using the guide plates, which improving the uniformity effectively. Zhu et al. [6] pointed out that if the guide plate is not well set, it will produce many vortexes and cause the unreasonable distribution of airflow in the duct. Therefore, it is necessary to design the layout of the guide plates carefully.

Aiming at improving the uniformity of air velocity at outlets, the flow field of the duct was studied by CFD and experiment methods, and the effect of bend chamfer and the guide plate on vortexes was analyzed emphatically. The optimized scheme was proposed to optimize the structure of the air duct, and the performance of the new duct was also analyzed. Finally, the air ducts before and after
optimization were placed in the whole shelter to verify the optimization, so the temperature distribution inside the shelter was studied.

2. Numerical Simulation Method
The steady analysis method was used to calculate the flow field, and the Standard k-ε model was chosen, which is the most widely used turbulence model in engineering [7]. The model is a semi-empirical formula. Turbulence energy k and turbulence dissipation rate \( \varepsilon \) are introduced and their corresponding governing equations are:

\[
\frac{\partial}{\partial t}(\rho k) + \frac{\partial}{\partial x_i}(\rho k u_i) = \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_t}{\sigma_k}\right)\frac{\partial k}{\partial x_j}\right] + G_k - \rho \varepsilon \tag{1}
\]

\[
\frac{\partial}{\partial t}(\rho \varepsilon) + \frac{\partial}{\partial x_i}(\rho \varepsilon u_i) = \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_t}{\sigma_\varepsilon}\right)\frac{\partial \varepsilon}{\partial x_j}\right] + C_1 \varepsilon \frac{k}{k} G_k - C_2 \rho \varepsilon \frac{\varepsilon^2}{k} \tag{2}
\]

In the formula, \( \rho \) is density, \( \mu \) is dynamic viscosity coefficient, \( \mu_t \) is turbulent viscosity coefficient, \( G_k \) is turbulent energy generation term, constants \( C_1 = 1.44, C_2 = 1.92, \sigma_k = 1, \sigma_\varepsilon = 1.3 \).

The analysis of the duct only studied the internal airflow, so the temperature change was not considered. However, the energy conservation equation should be calculated to obtain the temperature field of the whole shelter. In this paper, air was regarded as an incompressible fluid whose density is constant, so the equation of energy conservation can be written as follows:

\[
\frac{\partial T}{\partial t} + \text{div}(UT) = \text{div}\left(\frac{k}{\rho c_p} \text{grad}T\right) + \frac{S_T}{\rho} \tag{3}
\]

In the formula, \( U \) is the velocity vector of the fluid, \( T \) is the thermodynamic temperature, \( k \) is the heat transfer coefficient of the fluid, \( C_p \) is the specific heat capacity, \( ST \) is the viscous dissipation term.

The pressure-velocity coupling method here adopted SIMPLE algorithm. The default relaxation factor was used in the initial calculation. If instability occurs, the relaxation factor can be adjusted appropriately to improve the convergence. The convergence conditions were set as follows: residuals of the velocity, continuity, \( k \) and \( \varepsilon \) were less than 10-3 and the residual of energy was less than 10-6. Furthermore, several variables related to the required calculation results were selected as monitoring variables. When the monitoring variables are basically stable, the calculation can be finished.

3. Establishment of CFD Model

3.1. Physical Model
Due to the complexity of the physical structure, simplification was made in the modeling. The effect of the adjusting devices at the outlets on the air volume was taken into account, so the actual equivalent area of the outlet was used. The influence of the inner wall materials on the airflow was ignored. Only the passenger seat was reserved in the shelter. In CATIA software, the 3D model of the fluid area was established directly, including the air duct, which is shown in Figure 1.
There is a guide plate at the inlet of the duct. The number of the outlets is 24, so they are numbered for easier identification, which is shown in Figure 2.

3.2. Mesh Generation
In order to fit the geometry better, a tetrahedral mesh was generated in the Meshing module of ANSYS software. The mesh of the duct has five boundary layers with a growth rate of 1.1, as is shown in Figure 3. The height of the first layer was determined by y+ value [8], and the empirical value is between 30 and 300[9]. Finally, it was calculated to be 2 mm.

The volume of the shelter is larger and the structure is more complex. Boundary layers can easily cause worse grid quality. Besides, this analysis cares little about the airflow near the surface, so the boundary layer was no longer generated here. Finally, a tetrahedral mesh was used for the whole shelter, as is shown in Figure 4.
The quality of the grid mainly considers the skewness, which has the greatest impact on the convergence. Skewness of the most grid ranges from 0 to 0.5, and very few of them are more than 0.9. Therefore, the grid quality of the CFD model can meet the calculation requirements.

3.3. Boundary Conditions

3.3.1. Boundary condition of the duct. The airflow inside the duct is a typical internal flow problem. The boundary conditions were mainly set for inlet and outlet. The inlet was set as a velocity inlet. The air volume is 2240m³/h under cooling condition, so the velocity was calculated to be 7.98m/s; the outlet was set as a pressure outlet and the pressure was set to 0Pa. The default wall condition is non-slip wall.

3.3.2. Boundary condition of the shelter. The velocity at the inlet was the same, while the outlet was changed to the return air outlet of the shelter, so the corresponding size was changed accordingly. The outlet was still set as a pressure outlet, and the pressure was set to 0Pa. In addition, it is necessary to set the initial temperature on the boundary when calculating the temperature field in the shelter. In summer, the ambient temperature is relatively high, so the temperature of the return air outlet was set to 40 degrees Celsius. The initial temperature inside the shelter should be slightly lower than the ambient temperature, so the temperature of the wall was set to 37 degrees Celsius. The inlet temperature of the air conditioner should be set to 20 degrees Celsius according to the reference [10]. The heat transfer between the wall and the outside environment was not considered here.

4. Flow Field Analysis and Experimental Verification of the Duct

4.1. Flow Field Analysis of the Duct
First of all, the flow field of the air duct was studied alone. As is shown in Figure 5, the pressure at the bends is high outside and low inside, and there is even a negative pressure area on the inside of the left bend. Combining with the streamline in Figure 6, it is found that a large-scale vortex appeared in this area. The change of cross section at the left bend is more complex than that at the right, so the sudden change of velocity is more severe. There is an obvious high-speed area on the outside of the left bend, and the flow path is irregular. The vortexes at the bend cause pressure loss, which seriously affect the velocity at the close outlets, especially at outlet 3 and 4.

4.2. Air Velocity Experiment
The above simulation results are based on the data of flow field, so it is necessary to test and verify the air velocity. When the duct is working normally, the velocity at each outlet can be measured by QDF-6 anemometer. Then it was compared with the simulation data, as is shown in Figure 7. The velocity at outlet 3 and 4 close to the large vortex on the inside of left bend is significantly influenced, which is much smaller than the velocity at other outlets.
Figure 7. Comparison of simulation and experiment data

The simulation conditions are more ideal, and there are some errors in the actual measurement, such as the flow field’s fluctuation, so there will be some deviation in the measured data. However, the deviation of the results is not large and the general trend of velocity at each outlet is basically consistent. In conclusion, the established CFD model is close to reality, which can meet the needs of subsequent analysis and ensure the reliability of simulation results.

5. Research on Improvement Method for Vortexes

5.1. Effect of chamfers at the Bend on Vortexes
As is shown in Figure 8(a), when the inside and outside of the bend are right angle, the flow transition at the bend is extremely unnatural. Two large-scale vortexes are formed at the right angle. However, the vortex on the outside is not located in the central line of the airflow, so its effect on the overall airflow is relatively small. On the contrary, the vortex on the inside has a large impact on the flow field, which should be eliminated or reduced. The existence of the right angle makes the direction of the airflow change greatly. The airflow completely separates from the inner wall of the bend.

The vortex on the outside disappears in Figure 8(b), after chamfering the outside of the bend, but the flow field in the remaining areas is not improved. Especially, the large-scale vortex still exist on the inside of the bend. Compared to Figure 8(c), after chamfering the inside of the bend, the vortex on the outside has no change, but the vortex on the inside has improved significantly. Although there is still some flow separation, the overall flow field is more uniform. The existence of inner chamfer makes the flow path more close to the inner wall, and the large high-speed zone disappears. Velocity change in small area occurs at the inner wall of the bend due to sudden change of the direction.

Finally, the inside and outside of the bend were all chamfered, as is shown in Figure 8(d). The results show that the two large-scale vortexes are almost eliminated, and the uniformity of the flow field inside the duct is greatly improved. This proves that the inner and outer chamfers at the bend, especially the inner chamfer, have a very important influence on the uniformity of the flow field.
5.2. Effect of the Guide Plate on Vortices

In addition to chamfering the inside and outside of the bend, it is also an effective method to install guide plates at the bend. It can be seen in Figure 9(a) that the high-speed area at the outside of the bend is reduced and the vortex is also improved after installing a guide plate while the bend is right-angle. Furthermore, the velocity distribution is more uniform and the vortices are better controlled when two guide plates were installed in Figure 9(b), but a vacuum area still exists.

Obviously, the vortices problem cannot be well solved just by installing guide plates. Therefore, on the basis of chamfering the bend, then the guide plate was also installed, which is shown in Figure 10. It proves that the flow path of the air is more reasonable, the transition is more natural, and the sudden change of the direction is reduced. In Figure 10(a) and 10(b), the flow separation at the inner wall is weakened, and the vacuum area is reduced to a smaller size, which can almost be ignored. Of course, the number of guide plates depends on the specific situation.
In summary, through the analysis of an ordinary 90 degree rectangular bend, it is found that the chamfers on the inside and outside of the bend and the guide plates at the bend play an important role in eliminating the vortexes and improving the uniformity of the flow field. Considering the duct in this paper, it already has chamfers on the outside of the bend, but the inside of the bend is right-angle, so the design is obviously unreasonable. The inner chamfer of the bend should be the object of optimization. However, the change of cross section at the bend is complex, so it is not convenient to install the guide plates in this case.

6. Optimization of the Duct Structure

For a more comfortable experience, the velocity at outlets should be reduced, so combining with engineering design experience, outlet radius, inner chamfer of bend and inlet area were selected as optimization variables. These three variables make little change to the overall structure of the duct and are easy to process. The final optimized scheme is shown in Table 1. The change of the inlet area was based on the original center, and the size of all sides were scaled equally.

| Variables               | Original | Optimal |
|-------------------------|----------|---------|
| Outlet radius (mm)      | 32       | 34      |
| Inner chamfer of bend (mm) | 0       | 50      |
| Inlet area (10^4mm^2)  | 7.8      | 9.2     |

As is shown in Figure 11, the airflow at the bend is obviously more uniform. Especially, the vortexes on both sides have been effectively improved. The chamfers make the flow path change slightly gentle and the direction change is more natural.

![Figure 11. Streamline of the optimized duct](image-url)
The velocity at outlet 1 and 2 is quite different from other outlets, which is not taken into account here. The maximum air volume difference between the outlets at two straight ducts is shown in Table 2. The uniformity of velocity at outlets on both sides of the straight duct has increased by 47% and 42%, respectively.

| The maximum air volume difference Original Optimized Improvement |
|---------------------------------------------------------------|----------------|----------------|----------------|
| Left side (m³/h)                                             | 59.30          | 31.47          | 47%            |
| Right side (m³/h)                                            | 28.37          | 16.37          | 42%            |

7. Analysis of Temperature Field in the Shelter

7.1. Air Distribution in the Shelter
As is shown in Figure 12, the airflow will change its path when it encounters obstacles, so the flow path inside the shelter is irregular and there are vortexes due to the existence of the walls and chairs. The two schemes use the same number of sample points to draw the streamline, but the airflow of the optimized scheme in the shelter is obviously more than that of the original scheme. Although the average velocity at outlets of the optimized duct decreases, the total air volume increases from 104.78 m³/h to 107.03 m³/h with the expansion of the area of each outlet, so streamlines of optimized scheme are more intensive.

![Streamline of the shelter](image)

Figure 12. Streamline of the shelter

7.2. Temperature Field in the Shelter
Two sections in the vertical and horizontal directions at the passenger area are selected to analyze the temperature distribution inside the shelter, especially the passenger area. Figure 13 and 14 show the temperature field of the vertical and horizontal sections before and after optimization, respectively. The temperature of each area on the two sections of the optimized scheme is significantly lower than that of the original scheme, which proves that the new duct has better cooling effect, because the new duct can send more air volume into the shelter at the same time.

It can be seen from the vertical section that the temperature is lower along the slender forward path of the airflow at each outlet than that of the adjacent area in Figure 13(a) and 13(b). The closer the airflow is to the outlet, the more obvious the phenomenon is. The space of the front shelter is smaller than that of the rear one, so the cooling effect at the front part is better.
The horizontal section in Figure 14(a) and 14(b) is located at the height of the louver between the front and rear shelter, so there will be more air exchange near the louver. The airflow with higher temperature in the rear shelter flows to the front shelter through the louver, which makes the temperature change at two sides of the louver. The air in the back is near the corner, so the air volume is less, which makes the temperature in the front of the rear shelter slightly lower than that in the back.

The temperature of the front shelter in the horizontal section shows the phenomenon of low left and high right, especially in the horizontal section of the original scheme. The velocity at the outlet 3 and 4 near the left bend of the original duct is much smaller, so the air blowing to the right side of the shelter is less than the left side, which leads to higher temperature on the right side. After optimization, the vortex is weakened and the influence on the velocity at outlets is reduced. Therefore, the temperature imbalance between the left and right sides of the front shelter has been improved.

8. Conclusion

The flow field of a duct is analyzed by CFD and experiment methods. The duct structure is optimized to improve the uniformity of velocity at outlets. Furthermore, the temperature distribution in the shelter is more uniform. This can provide guidance for the development of duct products and greatly shorten the development cycle.

Through the simulation and optimization, the following conclusions can be drawn:

1) The right-angle transition at the bend will produce severe vortexes, which will cause pressure loss. Chamfers at the bend can smooth the flow path and eliminate the vortexes well. The guide plates at the bend can be added according to the need to achieve better performance. The number of guide plates should be determined according to the width of the duct and the simulation results.

2) The total air volume of the optimized duct has increased, leading to better cooling effect. The uniformity of temperature distribution has been improved with the optimized scheme, especially when the vortex at the left bend is eliminated, the uniformity of temperature in the front shelter is improved obviously.

Acknowledgments

This work was financially supported by Research and Application Demonstration Project on Key Technologies of Emergency Medical Rescue Support Complete Equipment (2017YFC0806405) and Fundamental Research Funds for the Central Universities (No.309181B8809).
References

[1] Sun T, Zhang Y, Wang Z. Research on flow in 90° curved duct with round section, Physics Procedia. 24 (2012) 692-699.

[2] Tao QM, Xu ZB, Xia GF. Analysis and structure optimization for auto HVAC defrosting duct. Journal of Hefei University of Technology (Natural Science), 33 (2010) 498-500.

[3] Cheng MH, Liu G. Effect of air-conditioning duct design on cabin inside noise[J]. Journal of Shenyang Institute of Aeronautical Engineering, 22 (2005) 37-39.

[4] Li M, Li GD, Zhao WB, et al. An Analysis on the Modification Effects of Outlet Duct Structure on Vehicle Defrosting Performance, Automotive Engineering, 40 (2018) 1364-1369.

[5] Yang B M, Cho H M. Flow analysis of air intake duct for noise reduction in automobile, Contemporary Engineering Sciences, 9 (2016) 989-995.

[6] Zhu JJ, Su XP, Chen JP. A Study on Structural Optimization for Defrosting Duct of Moblie Air-conditioner. Automotive Engineering, 26 (2004) 747-749.

[7] Hu K, Li ZB. ANSYS ICEM CFD Engineering Example, first ed., People's Posts and Telecommunications Publishing House, Beijing, 2014.

[8] Bonitz S, Larsson L, Löfdahl L, Sebben S. Numerical Investigation of Crossflow Separation on the A-Pillar of a Passenger Car, ASME. J. Fluids Eng. 140 (2018) 111105-111105-9.

[9] Blejchař T, Michalcová V. Flow in Atmospheric Boundary Layer in the Surrounding of Coal Stockpile–Effect of Air Direction on Spontaneous Ignition, Archivum Combustionis, 30 (2010) 115-124.

[10] Li YW, Xing X, Hu ST. CFD Simulative Evaluation for Air Distribution and Heat Comfort in one Air Conditioning Office, Journal of Qingdao Technological University, 26 (2005) 103-106.