Flow Characteristic Analysis of CFD-based Air Filter Forward Intake Pipe

Hongliang Wang¹, *, Pidong Shi²

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

*Corresponding author: hongliangwang@sdvcst.edu.cn

Abstract. Based on one type of heavy vehicles with air filter forward intake pipe as the research object, the flow field characteristics of air filter forward intake pipe is simulated. The original model of the flow characteristics such as pressure and velocity fields are analyzed based on the simulation results, and an improved model is proposed. Through the numerical simulation analysis of the improved model, it is proved that the internal flow characteristics have been significantly improved, and the intake resistance has been reduced, and the improved model structure is in line with the flow characteristics of the gas better.

Keywords: Air Filter Forward Intake Pipe; Numerical Simulation; pressure fields; velocity fields.

1. Introduction

As part of the air filter system, the front intake pipe of the air filter is located in front of the air filter. It is mainly used in the engine of vehicles such as heavy vehicles with poor working conditions. At present, numerical simulation analysis of research objects by Fluent and other software has been widely used in various fields, but for the intake pipe is little research before air filter, which makes the application on the market before the intake pipe, single function, dirt and prevention, separating the rain ability is insufficient, bring burden to the downstream air filter, reduces the working efficiency and service life [1].

As is known to all, the filtration capacity of air filters has a direct impact on the dynamic performance and economic efficiency of vehicle engines. How to improve the uniformity of fluid flow in the air filter system and reduce its intake resistance while ensuring the filtration efficiency is the key to improve the performance of air filters [2-3]. In addition, the total pressure loss of the intake system is an important factor affecting the performance of different engines. If the total pressure loss is too large, it will definitely affect the dynamic performance and fuel economy of the engine [4]. Therefore, under the condition of ensuring the filtration efficiency, the total pressure loss should be reduced as far as possible. In this paper, for the purpose of improving the flow uniformity of the internal flow field and reducing the intake resistance, the flow field characteristics of the original model were numerically simulated based on Fluent software, and the flow field characteristics such as pressure field and velocity field of the original model were analyzed. From the perspective of structure, the reasonable degree of the intake
pipe was judged, an improved model was proposed, and the rationality of the improved model was verified.

2. Geometric model construction and grid division

2.1. Simplification and establishment of geometric models
The geometric model of the front intake pipe of the air filter was established by using Pro-E. Some structures are simplified in the modeling process. For example, the outer screen at the upper entrance is not taken into account because it has little influence on the simulation process. In addition, due to the long feature length of the original model, the model is divided into upper and lower parts, and the main object of numerical simulation in this paper is the upper part. After simplifying the upper part, the 3d geometric model of the calculation area is obtained, as shown in Figure 1.

![Figure 1. Computational region geometry model.](image)

2.2. Grid division of geometric model
The model is imported into the pre-processing software Gambit for mesh generation. The specific grid division steps are as follows:

1. In order to obtain a more precise grid, the model is divided into two parts: the upper irregular region and the lower cuboid regular region;
2. Generate surface grid: generate surface grid for the side surface of the upper irregular region, and generate surface grid for the side surface of the lower cuboid regular region;
3. Volume grid generation: The upper region is divided by Cooper Scheme type, that is, the side of the generated surface grid and its corresponding surface are taken as the source surface and scanned according to the specified node mode to generate volume grid; The lower region is divided by regular grid (Map).

![Figure 2. Computes the regional grid model.](image)
According to the model structure, the generation of model mesh is simplified and refined appropriately to improve the calculation speed and accuracy. The specific implementation is as follows: Since the upper area is the research focus area, the upper Interval Size is set to 3, and the lower Interval Size is set to 4. FIG. 2 shows the generated computational area grid model, in which the total number of cells and nodes is 414,060 and 437,172.

3. Mathematical Model

3.1. The governing equation of three-dimensional viscous incompressible turbulent flow

In the analysis of flow field, the model should be simplified and assumed as follows:

1. In flow field analysis, single-phase flow model is adopted for simulation, that is, only air is considered as a medium, and other impurities such as particles or water are ignored;
2. At normal temperature, the temperature change is not obvious, so the temperature change is ignored and the whole process is considered adiabatic;
3. As mentioned above, if the air temperature change is ignored, its relative density change can also be ignored, and the fluid is considered to be incompressible flow.

In addition to the above assumptions, the flow state of the fluid needs to be determined by Reynolds number. The calculation formula of Reynolds number is:

\[
Re = \frac{\rho U d}{\mu}
\]  

(1)

Known conditions: inlet velocity \( U = 15 \text{m/s} \), equivalent diameter \( D = 209.5 \text{mm} \), standard pressure \( P = 101325 \text{Pa} \), when temperature \( T = 298.15 \text{K} \), the density of air is \( 1.185 \text{kg/m}^3 \), aerodynamic viscosity is \( 1.86 \times 10^{-5} \text{pa} \cdot \text{s} \).

After calculation, it can be concluded that Reynolds \( Re = 2.01 \times 10^5 \) belongs to turbulence with high Reynolds number, so its flow state is three-dimensional turbulent flow. Therefore, the standard K- two-way model with high Re is adopted to simulate turbulence. Therefore, the governing equation of the three-dimensional viscous incompressible turbulent flow in the front intake pipe of the air filter can be obtained as follows:

Mass conservation equation:

\[
\frac{\partial u_i}{\partial x_j} = 0 \quad (j = 1, 2, 3)
\]  

(2)

Where \( u \) is the velocity component.

Momentum conservation equation:

\[
\frac{\partial}{\partial x_j} (u_j u_i) = -\frac{\partial p}{\partial x_j} + \frac{\partial}{\partial x_j} \left( \frac{\mu_{\text{eff}}}{\sigma_k} \frac{\partial u_i}{\partial x_j} - \rho u_i u_j \right)
\]  

(3)

Where, \( \rho \) is the air density; \( \mu_{\text{eff}} \), \( \mu_1 \), \( \mu_t \) is the effective vortex viscosity coefficient, laminar viscosity coefficient and turbulent viscosity coefficient; \( k \), \( \varepsilon \) is the turbulent kinetic energy and dissipation rate; \( k \) equation:

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

(4)

\( \varepsilon \) equation:

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

(5)
Where, the empirical value of model constant is \[^5\]: \(C_1=1.44, \quad C_2=1.92, \quad k=1, \quad \varepsilon=1.3, \quad C_\mu=0.09\).

3.2. Calculation method and boundary condition setting

In this paper, the finite volume method (FVM) is adopted to establish the discrete equation for the three dimensional incompressible turbulent flow field in the front inlet pipe of the air filter; the convection term is adopted the first-order upwind difference scheme \[^6\]; the underrelaxation factor of each calculated physical quantity is adopted the default value; SIMPLE algorithm is adopted to solve the equation iteratively; and the standard K-two-way path with high Re is adopted to simulate the turbulent flow field in the front inlet pipe of the air filter. See Table 1 for setting the boundary conditions.

| Area    | The boundary types | The boundary conditions |
|---------|--------------------|-------------------------|
| Inlet   | velocity Inlet     | inlet velocity \(U:15\)m/s |
| Outlet  | Free outlet        | Turbulence intensity \(I:0.035\) |
| Wall    | Wall               | Equivalent diameter \(D:0.2095\)m |
|         |                    | The relative pressure \(P\) is zero |
|         |                    | Adiabatic, no slip       |

In this calculation, the number of iterative steps is set as 500, and the convergence accuracy is 0.001. When the actual iteration reaches 187, the residual of each calculated physical quantity has reached the specified convergence accuracy. The iterative process is finished, and the simulation results are analyzed.

4. Analysis of simulation results

4.1. Analysis of flow field characteristics of the original model

The velocity field and pressure field were obtained by numerical simulation using Fluent software on the front inlet pipe of the original air filter. In order to better explain the flow characteristics of the internal flow field of the model, it is necessary to select a section that can represent the internal overall flow characteristics for analysis. FIG. 3 and FIG. 4 show the velocity field and pressure field of the XZ section when \(y=100\)mm, respectively.

![Figure 3](image1)  
**Figure 3.** \(y=100\)mm, velocity field of XZ section

![Figure 4](image2)  
**Figure 4.** \(y=100\)mm, pressure field in XZ profile

By analyzing the simulation results of the original model, it can be seen that:

1. The change of velocity inside the flow field is relatively obvious: It can be seen from FIG. 3 that the velocity increases gradually in the inlet area, and the maximum velocity appears in the right area of
z=320mm. This is because the flow cross-sectional area of the internal flow field is suddenly changed, resulting in the increase of velocity. Low speed zone exists below the inlet corner and local backflow is generated.

(2) It can be seen from Figure 4 that the total pressure loss is mainly concentrated in the upper region of Z =320mm, which is mainly due to the local resistance loss caused by the structural mutation of the model. Among them, the pressure in the upper right corner is the largest, causing a strong impact on the wall surface; The lower side of the corner appears local negative pressure, resulting in greater local resistance loss.

4.2. Analysis of structural optimization and Improvement

4.2.1. Structural optimization and improvement. Based on the analysis of the results of the model of the front intake pipe of the original air filter, this paper proposes an improvement scheme for the structure of the original model in order to improve the flow uniformity of the internal flow field and reduce the intake resistance:

(1) Change the external structure at the entrance of the model to improve the internal flow uniformity and make it conform to the airflow flow characteristics;

(2) Adjust the inlet Angle of the model velocity inlet to reduce the impact on the wall;

(3) Adopt arc transition treatment for right Angle corners to reduce right-angle geometry structure and reduce local resistance loss as far as possible. The improved model is shown in Figure 5.

![Figure 5. Simplified geometric model after improvement](image)

4.2.2. Analysis of optimization and improvement results. The velocity field and pressure field are obtained by numerical simulation of the improved model. In order to better compare and analyze the difference in flow characteristics of the internal flow field before and after the improvement, the XZ profile at y=100mm was also selected for comparative analysis. FIG. 6 and FIG. 7 show the velocity field and pressure field of the XZ profile when y=100mm, respectively.
Figure 6. Y =100mm, velocity field of XZ section  Figure 7. Y =100mm, pressure field in XZ profile

It can be seen from the comparison between Figure 3 and Figure 6 that the improved internal flow field has better flow uniformity and the flow characteristics of the airflow are improved. The maximum velocity value in FIG. 6 is significantly lower than that in FIG. 3, thus reducing the impact on the wall. FIG. 6 A large range of velocity variation at the inlet corner generates strong local reflux.

By comparing and analyzing the profile of the pressure field of the model before and after the structural improvement (FIG. 4, FIG. 7), the pressure gradient of the model after the improvement is significantly lower than that before the improvement. The pressure drop of the model before the improvement was 1880Pa, and the pressure drop of the model after the improvement was 444Pa. It can be seen that the pressure drop of the model after the improvement was significantly reduced and the pressure distribution was more uniform. The local negative pressure at the corner still exists, but it is obviously lower than before the improvement.

5. Conclusions
Based on Fluent, numerical simulation was carried out on the front intake pipe of the air filter to comprehensively evaluate its internal airflow flow characteristics, and an improved model was proposed.

(1) After the improvement, the flow characteristics of the internal flow field of the model are significantly improved: the pressure drop is significantly reduced, the pressure distribution is more uniform, the intake resistance is reduced, and the internal flow movement is more in line with the flow characteristics of the gas;

(2) After structural optimization, there is a strong backflow at the entrance corner of the model, and local negative pressure still exists, which will still produce a certain resistance loss. Therefore, it is necessary to further optimize the model structure to improve the flow characteristics and reduce the intake resistance.

References
[1] Wang Enhao, Li Shiwei et al. Flow Characteristics analysis of new high inlet pipe [J]. Internal Combustion Engines and Power Plants, 2012(2):17-20.
[2] Fu Haiming, Yin Feng. Research on multivariate Correlation Of Filter Resistance of Air Filter [J]. Filtration and Separation, 2008(3):27-30.
[3] Zhang Hui, Fu Xuguang. CFD Analysis and Air Flow Optimization of Air Filters [J]. Fluid Transmission and Control, 2011(5):34-37.
[4] Ramasamy D et al. Design Optimization of Air Intake System (AIS)of 1.6L Engine by Adding Guide Vane. Proceedings of the International Multi-Conference of Engineer Sand Computer Scientists 2010. March 17,2010-March 19, 2010. Hong Kong :News Wood Ltd. 2010.999-1003.
[5] Wang Fujun. Computational Fluid Dynamics Analysis -- PRINCIPLE and Application of CFD Software [M]. Beijing: Tsinghua University Press, 2004.

[6] Chen Zuobin. Computational Fluid Dynamics and Its Application [M]. Beijing: National Defense Industry Press, 2003.