Commercial Vehicle Intake System Flow Field Mechanism Analysis

Sheng Li¹, Xingjun Hu² and Cunfu Chen¹

¹ Faw Jiefang Qingdao Automobile Co., Ltd, Qingdao, China
² State Key Laboratory of Automotive Simulation and Control, Jilin University, Changchun, China

Abstract. The numerical simulation and fluid dynamics simulation of the internal flow field and its influence mechanism of a commercial vehicle intake system were carried out aim at studying the flow field characteristics of the intake system. The internal and surface pressure, velocity, and turbulent kinetic energy distribution of intake systems under the rated operating conditions and three high torque conditions of the engine were analyzed. The simulation results showed that with the increase of the mass flow of the intake air, the pressure distribution on the surface of the intake system increased, and the pressure drop of the inlet and outlet increased rapidly. With the increase of the mass flow, the scale and number of vortexes in the intake system increased sharply. The distribution of turbulent kinetic energy is more related to the velocity field distribution. However the internal turbulent energy distribution of the intake system is concentrated under one condition. As the intake air mass flow rate drops from 1,500 kg/h to 764 kg/h, the pressure drop of the intake system drops from 2,858,566 Pa to 721,411 Pa; the average turbulent energy decreases from 5.41 m²/s² to 0.896 m²/s². In this paper, the research on the flow field of the engine intake system aims to provide research directions for the early stage of the structural design of the intake system of large agricultural vehicles.

1. Introduction

During engine operation, rapid and sufficient intake mass flow can ensure that the engine always maintains optimal combustion efficiency, thereby providing stable power for the vehicle.

In the case of vehicle air intake systems, foreign scholars have done a lot of research using 3D simulation methods, and through flow field analysis, numerical model derivation and mechanism exploration of the air intake system have been performed [1-5]. To improve the efficiency of air intake systems, Saravanan and Gokhale et al. A new double suction manifold system was designed and developed. In this way, there is no need to use a valve to guide filling, and no external actuation mechanism and moving parts are required. This further improves the reliability of the system. As a result of the experiment, it was found that the torque and power of the engine increased by more than 47% under POT conditions [6]. The difference in in-cylinder flow with and without the intake valve was studied, and many characteristics of the in-cylinder flow structure and differences in local velocity and turbulence at rotational speed, vortex and spark plug positions were discussed [4]. Simoles proposed an unsteady-state simulation method in which qualitative results can provide direction for
research at an early stage in the design process, discussing the implemented examples and actual results to help designers develop intake manifolds [7].

Sun et al. developed a turbulence model that can be used to calculate the flow field at the intake valve of the engine and compared the calculation results of the standard model and the modified model with experimental data. The modified model for predicting the flow field in the airway has been further improved [8]. Yang and Jiang also investigated the effect of the new turbulence model on the simulation accuracy of the inlet [9-10]. As a system for studying the vortex inside the cavity, the carburettor, suction pipe and in-cylinder flow field of a 4-stroke gasoline engine were used [11]. Wang et al. studied the effect of the vortex ratio on the suction system. A study on combustion characteristics through experiments and simulations [12]. Results of numerical simulations on the flow field of the intake system showed that the optimization of the structure reduced the flow loss of $\tau$, Li improved the engine's torque, power and fuel economy through the intake system [13-14]. Liu optimized the flow field structure of the suction manifold by combining one-dimensional and three-dimensional analysis methods. The local resistance was significantly reduced, the inhomogeneity of the intake air was reduced, and the design standard of the intake manifold was reached [15].

In this paper, the actual model of a commercial vehicle air intake system is scaled, and then the flow of the air intake system is simulated and analysed based on commercial CFD software. Comprehensive exploration of the flow characteristics of commercial vehicle suction systems. Pressure distribution, velocity field, and turbulent kinetic energy distribution identify the causes and key locations that affect the intake efficiency of the intake system and provide designers with design and optimization guidelines.

2. Mathematical Model

2.1. Continuity Equation

The mass conservation equation is the continuity equation [16]. If it is steady state, the density does not change with time. The continuity equation is as follows:

$$\frac{\partial (\rho u)}{\partial x} + \frac{\partial (\rho v)}{\partial y} + \frac{\partial (\rho w)}{\partial z} = 0$$

(1)

2.2. Momentum Equation

$$\frac{\partial (\rho u)}{\partial t} + \text{div}(\rho uU) = \text{div}(\mu \text{grad} u) - \frac{\partial P}{\partial x} + S_u$$

$$\frac{\partial (\rho v)}{\partial t} + \text{div}(\rho vU) = \text{div}(\mu \text{grad} v) - \frac{\partial P}{\partial y} + S_v$$

$$\frac{\partial (\rho w)}{\partial t} + \text{div}(\rho wU) = \text{div}(\mu \text{grad} w) - \frac{\partial P}{\partial z} + S_w$$

(2)

where $\mu$ is the dynamic viscosity, $u$, $v$, $w$ is velocity in three direction, $P$ is the pressure, $S_u$, $S_v$, $S_w$ is the generalized source term of the momentum conservation equation.

Equation (2) is the momentum conservation equation, also known as the Navier-Stokes equation [17].

2.3. Energy Equation

If there is heat exchange in the fluid flow process, the law of energy conservation must be satisfied. The energy conservation equation of the following formula is based on $T$:
\[ \frac{\partial \rho T}{\partial t} + \text{div}(\rho UT) = \text{div}\left( \frac{k}{c_p} \text{grad}T \right) + S_T \]  

(3)

where \( k \) is the thermal conductivity, \( T \) is the temperature, \( C_p \) is the specific heat capacity, \( S_T \) is the viscous dissipation term [17].

In order to close the system of equations, we need to add an equation of state:

\[ P = P(\rho, T) \]  

(4)

For ideal gases: \( P = \rho RT \), \( R \) is the molar gas constant.

### 3. Numerical Simulation

#### 3.1. Geometric Model

Figure 1 shows a picture of a commercial vehicle air intake system. The air intake system studied in this article is a three-dimensional geometric model of PROE provided by a company based on a prototype and a certain proportion. As shown in figure 2a, the original intake system model contains all the details of the actual model. In order to meet the requirements of fluent flow field analysis, the model needs to be partially simplified.

![Intake system](image)

(a) Before simplified  

(b) After simplified

**Figure 1.** Intake system.

**Figure 2.** Geometry model.

#### 3.2. Mesh

As shown in figure 3, in order to accurately name the structure of each part of the intake system, the names of each part of the intake system are marked in the figure. Because the structure of the air intake system is very complicated, even if simplified, to achieve the fluent calculation accuracy requires the grid size of the entire system to be at least 1 mm. In order to reduce the requirements for computing resources and improve the accuracy of calculation, the processed model is subjected to surface mesh partitioning, boundary layer partitioning, volume mesh partitioning, and mesh size settings in table 1. Regarding the thickness of the first layer of the boundary layer, due to the inconsistent pipe diameters of the various parts of the air intake system, the hydraulic diameter of the inlet system is 0.17 m as the hydraulic diameter of the entire system. Taking the rated conditions as an example, the first layer thickness is calculated using Y + Wall Distance Estimation to 1.5 mm. Finally, the irrelevance of the grid is verified by comparing the calculation results of different grid numbers, and a suitable grid generation scheme is selected. The number of grids after determining the optimal scheme is 18 million.
3.3. Boundary Conditions
The process simulated in this paper is that the external airflow enters from the inlet of the air intake system, and flows freely through the entire air intake system and exits from the outlet. Due to the existence of the entrance grille, the inlet wind speed is not the direction of the entrance normal, but forms a certain angle with the normal. The specific settings are shown in Table 2.

| Boundary Conditions | Conditions |
|---------------------|------------|
| Inlet               | Velocity inlet (Magnitude, direction), Orientation is set at a 37.523° angle to the direction normal to the entrance plane |
| Outlet              | Pressure outlet, 0 Pa |
| Other boundaries    | Wall |

Based on the engine performance data sheet provided by the enterprise, four operating conditions of the engine air intake system were determined, including one rated operating condition and three high torque operating conditions. The intake air mass flow rate of each working condition is shown in Table 3. The intake air mass flow of each operating condition was inconsistent.

| Working condition        | Inlet mass flow (kg/h) | Velocity (m/s) |
|--------------------------|------------------------|-----------------|
| Rated working conditions | 1500                   | 7.402           |
| High-torque condition 1  | 1201                   | 5.927           |
| High-torque condition 2  | 1026                   | 5.063           |
| High-torque condition 3  | 764                    | 3.77            |
4. Analysis of Flow Characteristics of Intake System

4.1. Velocity Field Distribution

4.1.1. Rated Condition Velocity Field. Analysis of the main cross-section velocity cloud diagram of the air intake system in figure 4 shows that when the airflow flows from the inlet to the air intake system in a certain direction and size, the change in the speed is mainly caused by the change of the cross-sectional area of each part of the air intake system. The larger the pipe radius shrinks, the faster the speed increases; moreover, the presence of the wavy tube causes a local increase in speed. The presence of the vortex directly affects the air intake efficiency of the air intake system and reduces the amount of air intake, thereby affecting the cooling performance of the entire heat dissipation system.

![Velocity Field Distribution](image)

(a) Air filter and outlet pipe section  
(b) Section x = 0.87 m

*Figure 4. Main section velocity distribution under rated working conditions.*

4.1.2. Analysis of Velocity Field under High Torque Conditions. As shown in figure 5, compared with the rated operating conditions, the intake air mass flow rate is reduced under the three high torque operating conditions. The speed field distribution at 1201 kg/h, 1026 kg/h and 764 kg/h are basically the same. When the airflow passes through the wavy tube, the cross-sectional area of the pipe decreases, and the speed rises to a peak. Subsequently, the airflow hits the pipe wall, generating two vortices in opposite directions, causing a decrease in the airflow velocity and an increase in the overall pressure drop of the pipe. After the airflow flows through the right-angle pipe, the speed becomes slower, and the flow field appears as a more uniform flow field. The sharp increase in the speed of the outlet tube was caused by the rapid contraction of the nozzle.

4.2. Joint Analysis of Pressure Drop and Turbulent Kinetic Energy

Looking at figure 6 and table 4, the kinetic energy loss when the airflow flows in the pipeline is divided into the pressure loss along the path and the local pressure loss. The local pressure loss is directly related to the turbulent kinetic energy. In the flow field distribution of the air intake system, the turbulent kinetic energy is concentrated at the angle between the inlet airflow and the tube wall, the side of the wavy tube near the tube wall, the corner of the right-angle pipe, and the junction of the air filter and the outlet pipe. That is to say, the change of the shape of the pipe causes the flow velocity and direction of the air flow to change, thereby causing a backflow phenomenon.

With the increase of air flow velocity, the turbulent kinetic energy of the air intake system shows a rapid increase. The pressure drop also increases. Therefore, as the operating conditions change, the influence of the intake air mass flow rate on the intake efficiency of the entire intake system needs to
be carefully considered, so as to improve the heat dissipation performance of the entire cooling system.

(1) High-torque condition 1  
(2) High-torque condition 2  
(3) High-torque condition 3

(a) Air filter and outlet pipe section

(1) High-torque condition 1  
(2) High-torque condition 2  
(3) High-torque condition 3

(b) Section x=0.87 m

**Figure 5.** Velocity field distribution of intake system in high torque condition.

(1) Pressure drop  
(2) Turbulent kinetic energy

**Figure 6.** Intake system pressure drop and field average turbulent kinetic energy curve.

**Table 4.** Inlet system flow field pressure drop and turbulent kinetic energy simulation data.

| Mass flow (kg/h) | Pressure drop (Pa) | Change rate (%) | Turbulent kinetic energy (m²/s²) | Change rate (%) |
|------------------|--------------------|-----------------|----------------------------------|-----------------|
| 1500             | 2858566            | 0               | 5.413031                         | 0               |
| 1201             | 1834292            | 35.83           | 3.472659                         | 35.85           |
| 1026             | 1356896.2          | 26.03           | 2.899082                         | 16.52           |
| 746              | 721411             | 46.83           | 0.8962519                        | 69.08           |
5. Conclusions
This paper studies and analyzes the internal flow field characteristics of a commercial vehicle air intake system based on the FVM method. Specifically explored the pressure distribution on the surface of the air intake system under rated conditions and high torque conditions, the characteristics of the velocity streamline of the main section, and the characteristics and changes of the turbulent kinetic energy distribution. Based on the simulation results and research analysis, the following conclusions are drawn:

(1) The number and size of vortices in the internal flow field of the air intake system increase significantly with the increase of the intake air flow. The existence of vortices is the main reason for the local pressure concentration in the internal flow field of the air intake system. Therefore, when the operating conditions change, the pressure drop at the inlet and outlet changes significantly with the sharp increase in the number and size of vortices.

(2) The change of turbulent kinetic energy can reflect the generation and development of eddy current in the flow field. Turbulent kinetic energy is often concentrated in the area where turbulence begins to develop, and eddy currents are distributed in larger turbulent kinetic energy. When analyzing the flow of the internal flow field and judging the development of the flow, it can be analyzed by combining the distribution of the two.

Acknowledgments
This work was sponsored by National Natural Science Fund of China (11102070).

References
[1] Kuo T-W 1992 Multidimensional port-and-cylinder gas flow fuel spray and combustion calculations for port-fuel injection engine SAE.
[2] Peters B and Gosman A D 1993 Numerical simulation of unsteady flow in engine intake manifolds SAE.
[3] Gosman A D and Ahmed A M Y 2007 Measurement and multidimensional prediction of flow in an axiymmetric port/valve assembly SAE pp 649-661
[4] Khalighi B, Haworth D C and Huebler M S 1994 Multi-dimensional port-and-in-cylinder flow calculations and flow visualization study in an internal combustion engine with different intake configuration SAE.
[5] Choi C R and Huh K Y 1994 Laser tomographic visualization and pressure measurement of premixed flame propagation in a constant-volume chamber SAE.
[6] Saravanan D, Gokhale A and Karthikeyan N 2015 Design and development of a flow based dual intake manifold system SAE pp 68-74.
[7] Simoles T A and Rui B D C 2011 The use of dynamic simulation of flow in the design of intake manifolds SAE 2011-36-0083.
[8] Sun J, Mou Y, Dong Y and Lu X 1989 Simulation and experimental study of gas flow field in the inlet of engine model Journal of Internal Combustion Engines (02) 117-123.
[9] Yang Y and Wu C 1998 Numerical analysis of three-dimensional flow characteristics in the inlet steady flow test device Internal Combustion Engine Engineering (3) 67-71.
[10] Jiang Y and Fan W 2000 Large-scale computerized program for calculating the motion of spiral inlets and in-cylinder air in direct injection diesel engines: IPIC-CFD (I) Combustion Science and Technology 6 (2) 150-153.
[11] Jiang Y, Chen X, Zhang Y and Chen G 2003 Three-dimensional numerical simulation of carburetor-intake pipe-in-cylinder system Journal of Wuhan University of Technology (10) 68-70.
[12] Wang G, Yu W, Li X, Su Y, Yang R and Wu W 2019 Study on dynamic characteristics of intake system and combustion of controllable intake swirl diesel engine Energy 180.

[13] Li Z 2008 Improved Design and Flow Field Characteristics Analysis of FSAE Racing Engine Intake System (Hunan University).

[14] Hu D 2018 Optimization Design and Research of Inlet Port of a Gasoline Engine Based on CFD (Guangdong University of Technology).

[15] Liu P 2018 Research on Parameterized Design Method of Intake Manifold Based on Flow Uniformity (Jilin University).

[16] Veersteg H K and Malalasekera W 1995 An introduction to computational fluid dynamics: The finite volume method Pearson Schweiz Ag 20 (5) 400.

[17] Wang Y, Fan X and Yang R 2007 Numerical computational meshing and independence assessment of complex geometric models Journal of Zhongyuan Institute of Technology.