Flow analysis of engine intake manifold based on computational fluid dynamics

Jianmin Xu*
School of mechanical and automotive engineering, Xiamen University of Technology, Xiamen, China
*Corresponding author e-mail: xujianmin1020@163.com

Abstract. Intake manifold is a part of intake system, which has an important impact on intake uniformity of the multi-cylinder engine. Three kinds of intake manifold are designed and a three-dimensional flow field numerical simulation for them is conducted using software FLUENT. The flow characteristics of intake manifold are analyzed and the intake unevenness of intake manifold is calculated. According to the flow field contours, the reason of generating non-uniformity of intake system is analyzed. The results show that the inlet end arranged in the middle of the intake manifold can reduce the non-uniformity of intake system. And it can improve the engine intake and combustion quality. The calculation results provide a theoretical basis for designing the complex intake manifold

1. Introduction
Intake manifold should provide a uniform fresh charge for each cylinder of the engine, which is an important factor affecting the quality of engine combustion. Intake manifold is one of the main parts of the engine. Its main role is to provide uniform and stable intake air for the engine so as to ensure the intake uniformity of each cylinder. Uneven intake will produce unstable torque output, engine vibration, emissions increase and other problems. The rational design of the intake manifold not only can reduce the inlet pressure loss and increase the intake air quantity, but also can ensure the intake uniformity of each cylinder. Therefore, designing the intake manifold is one of the key technologies of ensuring the engine power, economy, and reliability and emission quality. The flow characteristics of air-fuel mixture flowing in various designs of manifold of IVECO 6 cylinder heavy-duty engine were studied [1]. Internal flow field analysis of engine intake manifold was conducted based on RE and CFD [2]. The internal flow characteristic in the intake manifold of a diesel engine was investigated computationally for the different intake manifold (Helical, Spiral and Helical-Spiral) configurations [3]. The transient behavior of air flow through the intake manifold of a heavy duty diesel engine was researched according to the step of valve opening [4]. A methodology for optimization of gas injector orientation for better thermal efficiency was emerged [5]. The flow distribution in an intake manifold under steady state turbulence conditions was studied [6]. A novel Dual Intake Manifold system was designed and developed which improves the torque at POT condition by making the charge to flow through the longer path and at WOT condition the charge flows through the shorter path[7]. The flow field of intake system of CW6200 gas engine was simulated [8-9]. And the intake uniformity of each
cylinder was studied. The gas liquidity and the gas flow quality in branched pipe can be improved by changing the air inlet direction and regulator chamber volume and other geometric parameters [10-13].

In this paper, the three-dimensional models of three kinds of new intake manifold for engine were established. The model was divided with tetrahedral mesh using software GAMBIT. Three-dimensional flow simulation was conducted using software FLUENT and the velocity contours of intake manifold were obtained. The influence of intake manifold structures on the intake non-uniformity was researched. The study provides the basis for the optimization design of the intake manifold.

2. Mathematical Model

2.1. Equations
For the stable dimensional compressible flow, there are mass and momentum conservation equations which are Reynolds-averaged.

\[
\frac{\partial}{\partial x_j} (\rho u_j) = 0
\]

(1)

\[
\frac{\partial}{\partial x_j} (\rho u_i \tau_{ij} - \tau_{ij}) = -\frac{\partial p}{\partial x_j} + s_i
\]

(2)

Where \( S_i \) is the source term, it represents resistance; \( \tau_{ij} \) is the stress tensor, and there are the following equations for Newtonian flow.

\[
\tau_{ij} = 2\mu(s_{ij} - \frac{1}{3} \frac{\partial u_k}{\partial x_k} \delta_{ij} - \frac{\partial u_i}{\partial x_j} \frac{\partial u_j}{\partial x_i})
\]

(3)

Where \( \mu \) is the molecular dynamic viscosity coefficient; \( \delta_{ij} \) is Kroneker number; \( \frac{\partial u_i}{\partial x_j} \frac{\partial u_j}{\partial x_i} \) is Reynolds stress tensor; \( s_{ij} \) is fluid strain rate tensor, which is determined by the following equation.

\[
s_{ij} = \frac{1}{2} \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right)
\]

(4)

2.2. Turbulence model of muffler flow field
The Reynolds stress is calculated using a standard -\( \kappa-\epsilon \) model to close the flow control equations:

\[
\rho u_i \mu_j = -2\mu_s s_{ij} + \frac{2}{3} (\mu_t \frac{\partial u_k}{\partial x_k} + \rho k) \delta_{ij}
\]

(5)

Where \( \mu_t \) is the turbulent viscosity coefficient, which is determined by the following equation.
\[ \mu_i = \frac{c_\mu \rho \kappa^2}{\varepsilon} \]  \hspace{1cm} (6)

Where \( \kappa, \varepsilon \) is turbulent kinetic energy and turbulent energy dissipation rate respectively. The transport equation is as follows.

\[
\frac{\partial}{\partial x_j} \left( \rho u_j \kappa - \frac{\mu_{\text{eff}}}{\sigma_{\varepsilon}} \frac{\partial \kappa}{\partial x_j} \right) = \\
\mu_{\text{eff}} \frac{\partial u_i}{\partial x_j} - \rho \varepsilon - \frac{2}{3} \left( \mu_i \frac{\partial u_i}{\partial x_j} + \rho \kappa \right) \frac{\partial u_i}{\partial x_j} \\
\frac{\partial}{\partial x_j} \left( \rho u_j \varepsilon - \frac{\mu_{\text{eff}}}{\sigma_{\varepsilon}} \frac{\partial \varepsilon}{\partial x_j} \right) = c_\varepsilon \frac{\varepsilon^2}{\kappa} \left( \mu_{\text{eff}} \frac{\partial u_i}{\partial x_j} - \frac{2}{3} \left( \mu_i \frac{\partial u_i}{\partial x_j} + \rho \kappa \right) \frac{\partial u_i}{\partial x_j} \right) + c_{\varepsilon 4} \rho \varepsilon \frac{\partial u_i}{\partial x_j} \]  \hspace{1cm} (7)

Where \( \mu_{\text{eff}} = \mu + \mu_i \); The empirical coefficients of \( C_\mu, \sigma_k, \sigma_\varepsilon, C_{\varepsilon 1}, C_{\varepsilon 2} \) and \( C_{\varepsilon 4} \) are determined according to Table 1.

| Table 1. The coefficient of experience |
|--------------------------------------|
| \( c_\mu \) | \( \sigma_k \) | \( \sigma_\varepsilon \) | \( C_{\varepsilon 1} \) | \( C_{\varepsilon 2} \) | \( C_{\varepsilon 4} \) |
| 0.09 | 1.0 | 1.22 | 1.44 | 1.92 | -0.33 |

3. Calculation model and boundary conditions

3.1. Geometric model
The three-dimensional models of 3 kinds of the intake system are shown in figure 1 to 3. The intake system is composed of an inlet end, regulator chamber and intake manifold. The role of intake manifold is to distribute the gas to each cylinder evenly. The structural parameters of the engine intake manifold are shown in Table 2. Three kinds of intake manifold structure are proposed. The inlet ends of design scheme 1 and 2 are located at the side of the regulator. The inlet end of design scheme 3 is located in the middle of regulator.

| Table 2. The parameters of intake manifold |
|------------------------------------------|
| The length of regulator cavity(mm) | 275 |
| The width and height of regulator cavity(mm) | 80 |
| Intake manifold diameter(mm) | 50 |
3.2. Meshing and boundary conditions
Meshing quality will directly affect the level of convergence speed, accuracy and computation time. It has a significant impact on the entire simulation. In order to make the final calculations close to the real situation, the quality of meshing should be improved. The tetrahedral mesh is used to divide the model of four intake manifolds, and the hexahedral mesh is applied in rest parts. Computing grid model of intake manifold systems is shown in figure 2. The SIMPLE algorithm is used to couple pressure and velocity field. The tube air flow is set to incompressible viscous turbulent flow. The standard turbulence model $k - \varepsilon$ is applied. Wall function method is used to describe the fluid velocity and pressure distribution of boundary layer. Turbulence intensity is set to 0.03, the turbulence length is set to 0.05. The front surface of the inlet end is set to mass flow inlet boundary, outlet face of each manifold is set to pressure outlet boundary and others are set to wall boundary.

4. Analysis of calculation results
4.1. Evaluation standard
Intake manifold flow coefficient of each outlet is usually the common method of evaluating intake uniformity. The maximum unevenness $E$ is adopted to represent engine intake unevenness. The formula is as follows.

$$E = \frac{(Q_{\text{max}} - Q_{\text{min}})}{Q_{\text{me}}}$$

(9)
Where $Q_{\text{max}}$ is the maximum flow rate of export quality; $Q_{\text{min}}$ is the minimum flow rate of export quality; $Q_{\text{ave}}$ is the average mass flow.

4.2. The results and discussion

4.2.1. Velocity distribution. Figure 3 shows the velocity contours of intake manifold for design scheme 1. The figure shows that the airflow velocity of the manifold which is far away from the inlet end is large and the airflow velocity near the entrance end is small. When gas flows into the intake manifold from the regulator chamber, the vortex is generated in the intersection of the regulator chamber and manifold because a sudden change in flow cross-sectional area. The air distribution of each intake manifold is uneven. The intake rate of the third and fourth manifold is big and the intake air is sufficient. Since the shaft of regulator manifold is perpendicular to each branch pipe axis, intake swirl is generated so as to result in feed gas shortage of the first and second manifold. According to the simulation result, the unevenness of the calculated intake is 9.31 percent. This shows that the intake uniformity of scheme 1 is poor. So the intake manifold is designed unreasonable.

![Image](a) The intake manifold 1 is open  
(b) The intake manifold 2 is open

![Image](c) The intake manifold 3 is open  
(d) The intake manifold 4 is open

**Figure 3.** The velocity contours of intake manifold for design scheme 1

Figure 4 shows the velocity contours of intake manifold for design scheme 2. The intake quality of intake manifold 3 and 4 is very good and the gas reflux phenomenon appeared in manifold 1 and 2. Reflux phenomenon causes an increase of flow resistance so as to result in reduction of flow coefficient. So the intake amount of entering the cylinder is reduced and the combustion process is affected. According to the simulation result, intake unevenness is 7.78 percent, so the intake uniformity of scheme 2 is poor.
The design scheme 3 is that the inlet end is located in the middle of the regulator. The velocity contours of each manifold are shown in figure 5. Intake air flow rate of each manifold is relatively uniform and no vortex phenomenon appears. According to the simulation results, the intake unevenness E is 4.716 percent, which shows that the intake uniformity of scheme 3 is very good.

4.2.2. Pressure loss analysis. Figure 6 shows the pressure loss comparison of 3 kinds of intake manifold. As can be seen from the figure 6 that the improving effect of the program 3 is the best. The pressure loss of each manifold is reduced and the pressure loss uniformity of the respective manifold is greatly increased. The intake air of each manifold is relatively uniform. The improving effect of the intake manifold in program 3 is feasible.
5. Conclusion
(1) Numerical Simulation of three kinds of intake system is conducted and flow field of intake manifold is analyzed. The results show that the inlet end arranged in the middle of the regulator can greatly improve intake uniformity so as to improve the intake efficiency of the engine and combustion quality.

(2) Based on computational fluid dynamics theory, three-dimensional numerical simulation of the engine's intake manifold system is carried out, which is an effective and feasible method of analysis.

(3) The results show that the pressure loss and flow uniformity of intake manifold are two major evaluation indicators, and the scheme 3 meets these two design specifications.

References
[1] Jemni M A, Driss Z, Kantchev G. Design and Modeling of Mechanical Systems, CA: Springer Berlin Heidelberg, pp.331-338, 2013.
[2] Wang C Y, He Z X, Wang Q. “Internal Flow Field Analysis of Engine Intake Manifold Based on RE and CFD”, Advanced Materials Research, vol. 1079, pp. 930-934, 2015.
[3] Rahiman A, AD M S, Ramis M K. “CFD Analysis of Flow Field Development in a Direct Injection Diesel Engine with Different Manifolds”, American Journal of Fluid Dynamics, Vol. 4, No. 3, pp. 102-113, 2014.
[4] Srisattha T, Issarakul S, Wechsatol W. “Transient Behavior of Air Flow through the Intake Manifold of a Heavy Duty Diesel Engine”, Applied Mechanics and Materials, Vol. 87, pp. 146-150, 2011.
[5] Chintala V, Subramanian K A. “A CFD (computational fluid dynamics) study for optimization of gas injector orientation for performance improvement of a dual-fuel diesel engine”, Energy, Vol. 57, No. 2, pp. 709-721, 2013.
[6] Thamaraikan R, Anish M, Kanimozhi B. “Design and Analysis of an Intake Manifold in an IC Engine”, Applied Mechanics and Materials, Vol. 766, pp. 1021-1027, 2015.
[7] Saravanan D, Gokhale A, Karthikeyan N. “Design and Development of a Flow Based Dual Intake Manifold System”, SAE International Journal of Engines, Vol. 8, No. 1, pp. 68-74, 2014.
[8] Zhang Q, Li N. “Numerical analysis of intake uniformity of biogas engine”, Transactions of the Chinese Society of Agricultural Engineering, Vol. 26, No. 8, pp. 145-149, 2010.
[9] Wang Han, Cai Yixi, Mao Xiaoping. “3D numerical simulation of intake unevenness for engines”, Small Internal Combustion Engine and Motorcycle, Vol. 36, No. 3, pp. 41-44, 2007.
[10] Fenzhu J, Keshuai G. “Numerical simulation of liquidity and uniformity of gasoline engine intake manifold” Journal of Beijing University of Aeronautics and Astronautics, Vol. 40,
No.2, pp.154-159, 2014.

[11] Ceviz M, Akim M. “Design of a new SI engine intake manifold with variable length plenum”, Energy Conversion and Management, Vol. 51, No 11, pp. 2239 – 2244, 2010.

[12] David C, Pascal C. “Fluid dynamic modeling of junctions internal combustion engine inlet and exhaust systems”, Journal of Thermal Science, Vol. 19, No. 5, pp. 410 – 418, 2010.

[13] Tong-qun H A N. “Design and Exploitation of Intake Pipe of Engine Equipped with EFI”, Chinese Internal Combustion Engine Engineering, Vol. 26, No. 3, pp. 22-26, 2005.