CFD Analysis of a Gasoline Engine Exhaust Pipe

Pengyun Xu*, Haiyong Jiang and Xiaoshun Zhao
Mechanical and Electronic Engineering College, Agriculture University of Hebei, Baoding 071001, P.R. China

Abstract
The exhaust pipe is an important part of gasoline engine. Its structure and performance have a direct impact on the engine power, economy and emissions, and it is one of the key technologies of multi-valve engine development. In order to test the theoretical design of a 1.5 L gasoline engine exhaust pipe, Solidworks Flow simulation was used to analyze the exhaust pipe. Pressure and velocity of the position near the three-way catalytic converter and the oxygen sensor were selective analyzed. CFD Simulation results show that the internal flow is laminar flow state, and the sensor position is reasonable. The design is reasonable, and can achieve the design goal.

Keywords: Exhaust pipe; Catalytic converter; Carrier component; Fluid uniformity; CFD analysis

Introduction
The exhaust pipe is an important part of gasoline engine. It connects all parts of the automobile exhaust system, and prevent the leakage of waste gas. Its structure and performance have a direct impact on the engine power, economy and emissions, and it is one of the key technologies of multi-valve engine development.

Automobile exhaust pipe is a space curved surface geometry, it has a certain difficult to manufacture, and the exhaust pipe is worked in a bad condition, and some problems will occur in the process of production and use. The traditional design method of the exhaust pipe is in steady flow test stand experiments, to obtain or test shape parameter, this method takes long time and cost. However, using computer fluid analysis technology (CFD), it is convenient and intuitive to analyze the three-dimensional model of the exhaust pipe, the analysis process is visual, and easy to adjust the parameters, the analysis results are intuitive, determine whether the structure meets the design requirements quickly. In this paper, SolidWorks Flow Simulation is used to simulate the structure of a certain type of engine. Pressure and velocity of the position near the three way catalytic convert-er and the oxygen sensor were selective analyzed. The design goals are verified by simulation analysis (CFD) results.

In order to reduce the concentration of CO, NOx and CxHy in the exhaust gas, there are (mostly) two catalytic converters installed in the exhaust pipe system. The crucial quantity to control the efficiency of a catalytic converter is the temperature in the catalytic converter. Due to this reason, one is interested in how to ensure a sufficient high temperature in the catalytic converters in a short time after the engine start. A special method of heating after the engine start is the combustion of unburnt gas in the catalytic converters. Modern cars can control the ratio of oxygen and fuel in the combustion chamber of the engine. By choosing a ratio with more fuel and less oxygen some unburnt fuel gets to the catalytic converters and can be used there for an exothermic reaction.

In order to test the theoretical design of a 1.5 L gasoline engine exhaust pipe, carrying out a natural experiment in respect gas flow hydrodynamics. To estimate the spectrum of possible technological innovations in the existing structure expedient, first carry out numerical simulations using fluid dynamics software packages solidworks flow simulation. This paper presents the numerical simulation of classical structure of exhaust pipe; the results are compared with the calculation for the case of a conical nozzle on the exhaust pipe [1-5].

Material and Methods
The calculation was carried out in gas dynamics software package Solidworks Cosmos FloWorks, which uses a finite volume method, the movement of the fluid is modeled by the Navier Stokes equations, the Reynolds averaged. Their closures are used for the transport equation and its kinetic energy dissipation within the k-ε turbulence model.

During the calculation was the condition of the grid and the iterative convergence, which were determined by the self-similarity of the final result of the number of cells and iterations.

Calculation of the motion of particles in the flow was conducted under the following assumptions:
• Uniform air and standard atmospheric pressure;
• The particles have a spherical shape;
• The drag coefficient of the particle is calculated by Henderson’s equation;

A series of pertinent indoor and outdoor experiments are carried out to know the engineering properties of exhaust pipe, and provide theoretical references for engineering quality evaluation and improvement scheme selection.

3-D model analysis of exhaust pipe
Using SolidWorks software to establish the 3D model of the exhaust pipe, as shown in Figure 1. The model parameters are shown in Table 1.

In order to control the automobile harmful gases such as NOx, HC and CO, the three-way catalytic converter must be installed in the exhaust pipe. The harmful gases can be converted into harmless carbon dioxide, water and nitrogen by oxidizing and reducing gases. The carrier component of the three-way catalytic converter is a porous ceramic material, which is installed in the specific position of the
exhaust pipe, and is the most important equipment in the automobile exhaust system. The three-way catalytic converter and oxygen sensor are generally installed in the exhaust manifold (natural gas engine) or after the turbocharger (turbocharged gasoline engine). The vector parameters and grid partition are shown in Tables 2 and 3. Grid partitioning result is shown in Figure 2.

**Boundary conditions setting**

Fluid computational domain is shown in Figure 3. Boundary conditions are shown in Table 4. The velocity in the inlet section was set as a fully developed turbulent flow in a pipe, its mean value of 0.13 kg/s; in the outlet section was set at ambient pressure 101325 Pa; on the walls of all the components of the velocity zeroed (condition “sticking”) [6-9].

Additional information:

(a) fixed wall boundary
Adiabatic slip free, fixed temperature wall 293 K, the boundary layer is treated by turbulent wall law.

(b) import and export boundary
Use the parameters in steady flow test, the values shown in Table 4.

**Analysis target**

The inhomogeneous flow of the front end face of the catalytic converter can produce the phenomenon of vortex flow and air separation, which cause the temperature distribution no uniform, also cause the carrier component damage, and then affect the engine’s work. Therefore, it is necessary to analyze the flow uniformity index $\gamma$, the range is between 0 ~ 1, and the 1 means completely uniform. When $\gamma$ is more than 0.9, the flow uniformity of the cross section is better.

If the position of the oxygen sensor in the exhaust pipe is not suitable, the oxygen sensor cannot measure the oxygen concentration accurately [10-15]. It will affect the air fuel ratio of the ECU calibration, and directly affect the engine’s performance and emission performance. So it is necessary to use CFD to analyze the flow field around the exhaust pipe.

The CFD analysis process is shown in Figure 4.

**Results and Discussion**

The computed result of all design variables were analysed and discussed in detail to identify the optimum performance of exhaust pipe. The computed result obtained at different design variables; effects of one design variable on other variables were assessed. Effects of flow rate, pressure, uniformity index were discussed in detail [16].

**Flow field analysis**

The whole pressure field shows that the fluid pressure decreases along the axis of the tube, and the pressure gradient is obvious in the position of the expanding port and the shrink port. The result is shown as Figure 5.
Figure 4: CFD analysis process.

Figure 5: Pressure field analysis.
Pressure field distribution

As shown in Figure 6, near the inlet elbow outside wall, the radial pressure of pre-catalytic converter carrier component is relatively large. Near axis center, the axial pressure of main-catalytic converter carrier component is relatively large [17].

Pressure drop analysis

The main pressure drop detection position of the air flow in the pipe is shown in Figure 7, and the pressure loss value is shown in Table 5.

Velocity field analysis

The Figure 8 shows that when the inlet flow is 0.13 kg/s, the highest flow rate can reach 104.8 m/s, and the velocity mutation mainly occurred in the import (export) conical surface. Fluid through the catalytic converter, Reynolds number Re<2000, is laminar flow state.

As is shown in Figure 9, the maximum speed of the pre-catalytic converter end face is 34.92 m/s, the maximum speed of the main catalytic converter end face is 48.91 m/s, which is less than 100 m/s, that is in accordance with the design requirements.

Oxygen sensor position CFD analysis

The front oxygen sensor is located in the main flow area, which is in accordance with the design requirements; The rear oxygen sensor is located in the main flow area, which is in accordance with the design requirements. The analysis result is shown as Figure 10.

Fluid uniformity analysis

In general, the calculation of the fluid uniformity coefficient γ is only for the end face of catalytic converter. When γ<0.9, it is necessary to optimize the import.

After calculation, the fluid uniformity coefficient γ of the main catalytic converter end face is 0.97, and the precatalytic converter is 0.696. The results are shown in Figure 11.

Velocity Index is a criterion for judging the radial force of the carrier component, as shown in Figure 12, the calculation method is shown in formula (1~3). Under normal circumstances, when the velocity index ε ≤0.7, that is in accordance with design requirements, but when the fluid uniformity γ ≥ 0.94, without considering the influence of Velocity Index [18].
### Table 5: Pressure drop.

<table>
<thead>
<tr>
<th>Position</th>
<th>Pressure drop data (KPa)</th>
</tr>
</thead>
<tbody>
<tr>
<td>P1</td>
<td>12.57</td>
</tr>
<tr>
<td>P2</td>
<td>9.09</td>
</tr>
<tr>
<td>P3</td>
<td>21.65</td>
</tr>
</tbody>
</table>

**Figure 8:** Velocity field analysis.

**Figure 9:** Section of velocity distribution.
Figure 10: Oxygen sensor position CFD analysis (pre & main).

Figure 11: Fluid uniformity analysis.

Figure 12: Velocity index.
conclusions can be drawn:

**Conclusion**

Take integrated analysis of the above test results, and the following conclusions can be drawn:

1. The exhaust pipe pressure loss of the catalytic converters is 12.57 kPa, 9.09 kPa, 21.65 kPa, which is in accordance with the design requirements.
2. The fluid uniformity coefficient of the inlet end face of the catalytic converter is satisfied.
3. The velocity index of the main and the pre catalytic converter meets the design requirements.
4. The maximum flow velocity of the pre and main catalytic converter is less than 100 m/s, which meets the design requirements.
5. The flow velocity of the oxygen sensor is higher and the oxygen sensor place is more reasonable.

**Acknowledgement**

The authors thank the Science research project of Hebei Province, Youth Science and Technology Fund of agriculture university of Hebei, and Baoding science and technology research and development plan for support.

**References**