Academic Journal of Science and Technology
ISSN: 2771-3032 | Vol. 9, No. 1, 2024

Gasoline Engine Inlet Structure Design and Performance Analysis

Zeyu Zhang[1, *], Jiaju Wang[1], Shiyun Wang[1]

[1] School of Mechanical Engineering, Tianjin University of Technology and Education, Tianjin 300222, China

[*] Corresponding author: Zeyu Zhang (Email: zhangzeyu990423@163.com)

Abstract: By creating a three-dimensional model of the intake pipeline, using AVL software to mesh and virtual simulation experiments, using the control variable method, changing the temperature conditions and pressure conditions one by one to observe the flow of the air pipe and the cylinder, and observing the numerical changes of the flow direction coefficient and tumbling flow ratio, the best solution was found. Through the virtual simulation experiment of the 3D model, the influence of external factors such as temperature and pressure on the air flow in the cylinder is obtained. The relationship between the characteristic parameters of the air intake tract was found through AVL numerical simulation, and the established three-dimensional model could accurately obtain the data after testing, which provided theoretical proof for the subsequent optimization scheme. The experimental results show that the tumbling flow ratio is increased as much as possible without changing the flow coefficient and the fuel economy of the engine is improved. But it also affects the flow coefficient, making it go down. Temperature has little effect on the flow in the cylinder. The flow velocity in the cylinder changes significantly under different pressures, and the area of the vortex area increases.

Keywords: Air intake ducts; Pressure; Flow direction coefficient; Tumble flow ratio.

1. Introduction

Since the 1980s, virtual software simulation experiments have been continuously promoted among major automobile manufacturers, and the United States, the United Kingdom and other countries have become more and more widely used in automotive research and development. The UK and France have conducted in-depth research on CFD technology, and have made remarkable achievements in the development of automotive air intake ducts.

Gosman et al. [1] used a three-dimensional computational model based on the k-ε two-way model to explore the influence of the air flow inside the engine on the shape of the combustion chamber. Taylor W et al. [2] proposed a calculation method for predicting the loss of the intake area of an internal combustion engine. J.N.KIM and H.Y.KIM used a full 3D model to investigate the optimal value of the intake valve lift of a CAI engine. Homogeneous mixing in the engine is a key parameter affecting the reliability and thermal efficiency of engine spontaneous combustion. The way the supply air is inlet often determines the homogeneity of the fuel-air mixture [3]. Kang Y H found that the in-cylinder flow characteristics of fuel injection and subsequent interaction with fuel spray and combustion are fundamental considerations for diesel engine performance and exhaust emissions [4]. Yi J W invented a new engine, EcoBoost, and combustion system performance is key to the success of the EcoBoost engine [5]. Godrie et al. [6] simulated steady-state flow in two typical inlet channel shapes: straight inlet and vortex inlet. The engine torque predictions based on the emission factor calculations are in good agreement with the test bench results in various situations. Schurov et al. [7] used a computer to simulate the flow and heat transfer phenomena in the intake tract of a spark-ignition engine. The resulting forecast provides a detailed picture of the characteristics of the air-fuel mixture along the intake tract. Bailly, Buchou, and Floch [8] developed a computational code for a hybrid finite element tetrahedral finite element method for engine simulation. A comparison was made between the calculation and measurement of the engine intake and compression stages.

With the advancement of computer technology abroad and the development of related research fields, the popularization of computer applications in China, and the progress and improvement of related research fields, China has correspondingly developed the application technology of numerical simulation of engine intake systems. Sun Jimei et al. [9] established a flow field analysis model for the engine intake system and used a modified k-ε The model calculated the operating conditions, and by comparing the experimental results with the steady-state calculation results, it was found that the corrected k-ε The prediction accuracy of the engine inlet flow field of the model is much higher than the pre correction prediction accuracy. Huang Zhen and Romoji [10] proposed a new numerical model mesh generation method, which combines the Euler method and Lagrange method to perform transient numerical simulation on the intake system of a certain engine. Pei Pucheng et al. [11] conducted several sets of steady-state flow experiments, in which they found that the flow direction and velocity of the airway were greatly changed when the airway structure was observed in the longitudinal and transverse planes with different structural planes. Wang Qiao et al. [12] found that the simulation results obtained by using FLUENT simulation software can accurately predict the test value of an engine, so as to verify the accuracy of CFD numerical simulation. Lu Yong et al. [13] developed a new cylinder, which has a different tumbling ratio from previous products. This combustion chamber optimizes the vortex currents in the cylinder and has proven itself in practice. Yang Xiaofeng [14] found that the gas flow is very closely related to the valve lift, and when the air flow is relatively stable, the valve lift is about in the upper middle position. The tumble flow ratio increases as the lift increases.

The testing methods for airway characteristics include AVL evaluation method, FEV evaluation method, and Ricardo evaluation method. Considering the advantages and disadvantages of the airway itself, and comparing various geometric structures of the airway, this paper uses the AVL evaluation method and applies the roll ratio and flow coefficient to evaluate the flow characteristics under different valve lifts.

We choose a more suitable speed as the actual operating speed of our engine, named N_{mot}. A vane anemometer is placed in the cylinder, and the speed of the anemometer when it rotates represents the speed of the gas flow in the cylinder, which is named N_{PADD}. When the gas enters the cylinder, we can observe a strong tumbling flow in the longitudinal direction. In the numerical simulation, assuming that the average velocity of the piston is equal to the average axial velocity of the flow field in the cylinder, the tumbling ratio TR is:

$$TR = \frac{n_{padd}}{n_{mot}} = \frac{\rho SA}{30m} n_{padd} \tag{1}$$

where S is the piston stroke;

$A = \frac{\pi \cdot D^2}{4}$ is the cross-sectional area of the cylinder;

m is the mass flow rate;

D is the radius of the cylinder.

The TR is integrated according to the AVL standard valve lift curve, which is called the integrated rollover ratio TR_{m}:

$$SR = \frac{n_d}{n_{mot}} = \frac{\rho SA}{30m} d\alpha \tag{2}$$

where SR is integrated according to the AVL standard valve lift curve, which is called the integrated vortex ratio SR_{m}:

$$SR_m = \frac{1}{\pi} \int_0^{\pi} \frac{n_d}{n_{mot}} \left(\frac{c_\alpha}{c_m}\right)^2 d\alpha \tag{3}$$

Because the screw inlet is mostly used in diesel engines, the vortex ratio is mostly the characteristic parameters of the inlet duct of the diesel engine, and there is also a small amount of vortex in the gasoline engine, but because its value is small and has little influence on the flow state in the cylinder, the vortex ratio is not used as its evaluation index in the gasoline engine.

2. Inlet Structure Design

2.1. Three-dimensional modeling

CATIA software was used to model the selected model in 3D, and the specific parameters are shown in Table 1. The geometric model was geometrically analyzed and meshed by the AVL software, and finally the model was imported into the AVL software and all parameters were set and the finite element virtual simulation was calculated.

Table 1. Model data parameters

Name	Data
Inner diameter	78.3mm
Stroke	86.4mm
Valve lift	6.1mm
Seat diameter	25.4mm
Connecting rod	158.0mm

Import the created geometry model into FIRE FAME Hexa for boundary condition editing and mesh generation, and set the maximum mesh size to 4 mm and the minimum mesh size to 1 mm. The grid was generated using a divided region, and approximately 420,000 cells were generated after processing. Airway model after treatment is shown in Figure 1.

The boundary condition pressure of the inlet surface is 100000Pa, the fixed temperature is 293.15K, and the boundary condition pressure of the outlet surface is 97500Pa, as shown in Table 2 and Table 3.

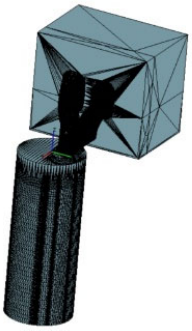


Figure 1. Airway model after treatment

Table 2. Parameters for setting boundary conditions for import surfaces

	Parameter	Numeric	Units
import	Sel. for BC	BC_inlet	
	Type of BC	Inlet/Outlet	
	Inlet/Outlet	Total Pressure	
	Pressure	100000	Pa
	Fixed temperature	Temperature 293.15	K

Table 3. Parameters for setting boundary conditions for exit surfaces

	Parameter	Numeric	Units
outlet	Sel. for BC	BC_outlet	
	Type of BC	Inlet/Outlet	
	Inlet/Outlet	Static Pressure	
	Pressure	97500	Pa

2.2 Data writing and simulation results and analysis

Simulation experiments are performed and the results are analyzed after the given conditions and data are written into the model. The maximum airflow at the valve exit is 85.113 m/s. As can be seen from Figure 2, a tumbling flow is formed in the cylinder when the gas enters the cylinder, and a part of the vortex formed in the upper right corner of the picture becomes an air resistance that hinders the gas from entering. At the entrance, the maximum flow velocity is present, and the mass flow on both sides simultaneously obstructs the intake process. This is an outcome that we don't want to see, so we need to start with structural optimization and improve to reduce the formation of resistance. As far as possible, it is possible to increase the tumbling flow ratio without reducing the flow coefficient, promote the inlet of the air flow and mix with the fuel, and improve fuel economy.

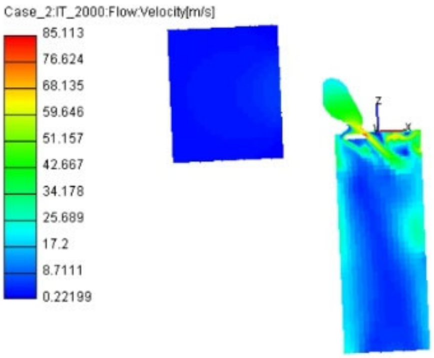


Figure 2. Scalar diagram of velocity

According to Figure 3 and Figure 4, the average dissipative energy grows rapidly in the first 250 iterations, grows slowly from 250 to 875 iterations, and tends to be flat in the back of 875. The average hybrid energy is growing rapidly in the first 500 iterations and tends to be flat in the future.

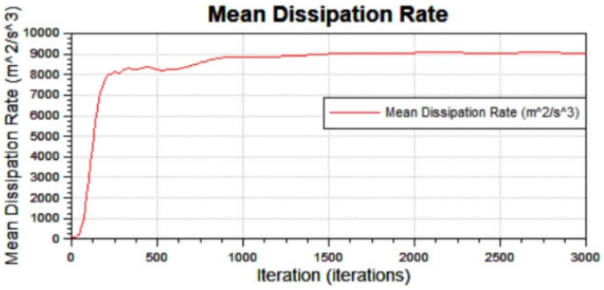


Figure 3. Average dissipated energy

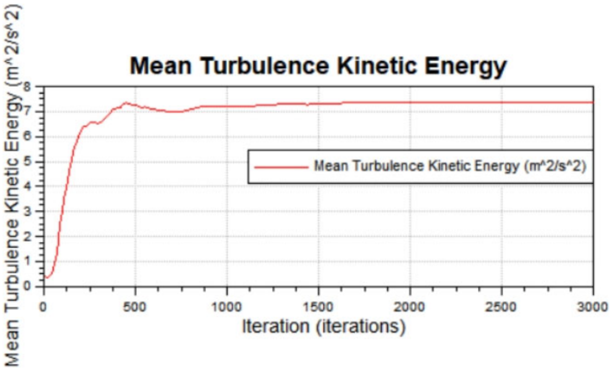
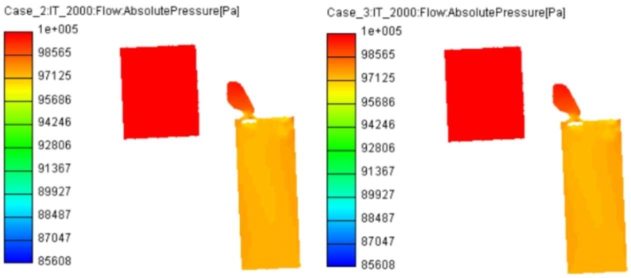


Figure 4. Average hybrid kinetic energy

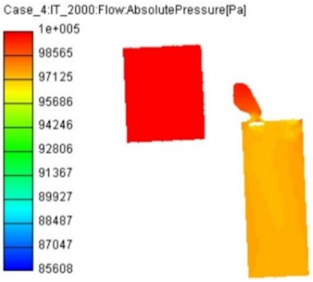
3. Intake Stroke and Intake Pipe Performance Analysis

3.1. The effect of temperature on the flow of gas in the cylinder

In order to clearly see the effect of temperature change on the experimental data, we designed an experimental group and two control groups to compare the air velocity and pressure in the cylinder, and the temperature was set to 293.15 K, 303.15 K and 313.15 K, respectively. They are shown in Figure 5 and Figure 6.

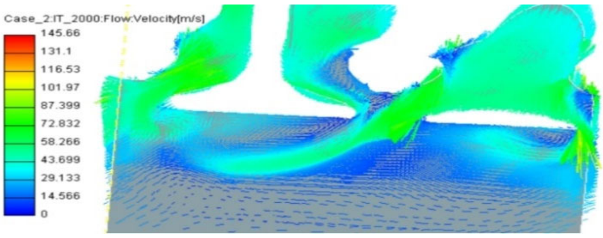


(a) Pressure chart at 293.15K (b) Pressure chart at 303.15K

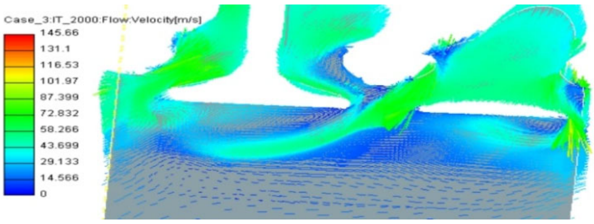


(c) Pressure chart at313.15K

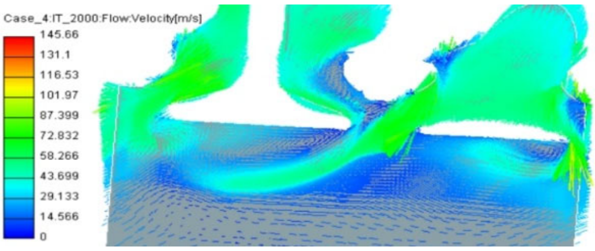
Figure 5. Comparison of pressure at different temperatures



(a) Enlarged view of the velocity graph at 293.15 K



(b) Enlarged view of the velocity graph at 303.15K



(c) Enlarged view of the velocity graph at 313.15K

Figure 6. Comparison of speeds at different temperatures

According to the simulation analysis results, it can be concluded that under the same other conditions, changing the temperature alone does not have a significant impact on the pressure change inside the cylinder. In order to rule out the possibility of a small temperature difference, a control group with a larger temperature difference was added in the following experimental comparison.

From Figure 7 and Figure 8, it can be seen that the effect of temperature on the flow velocity in the cylinder is not significant. To observe in more detail, we export a curve graph to observe the effect of temperature on other performance. The fourth line is a curve with an additional set temperature of 333.15 K. Observing the graph, it is found that the four curves are basically consistent, indicating that temperature has little effect on the airflow and pressure inside the cylinder. The impact on other performance parameters is not significant, which eliminates an important factor for our subsequent research.

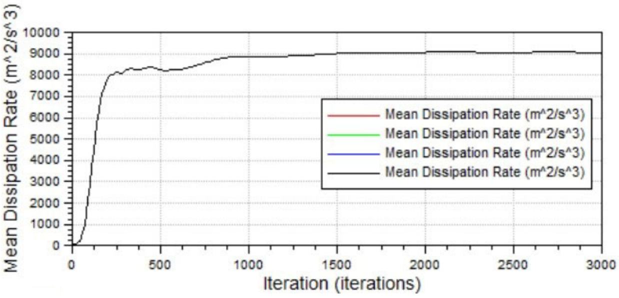


Figure 7. Average dissipation rate curve at different temperatures

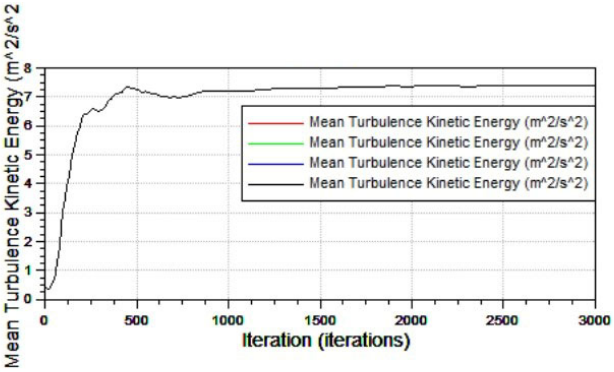


Figure 8. Average turbulent kinetic energy curve at different temperatures

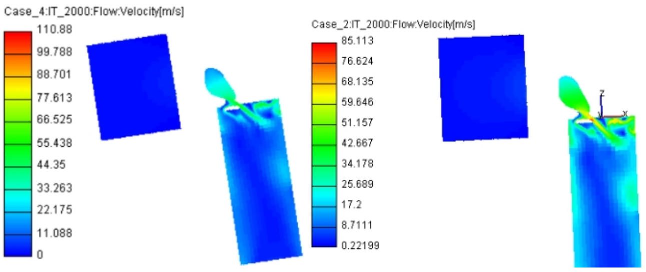
3.2. The influence of pressure on cylinder air pressure and airflow velocity

Due to the fact that changes in pressure will inevitably lead to changes in cylinder airflow and pressure, we have set multiple controls to study the impact of changing pressure on cylinder airflow. We have simulated each group, observed changes in images and curves, and analyzed cylinder airflow. Pressure Setting Parameters for the Experimental and Control Groups is shown in Table 4.

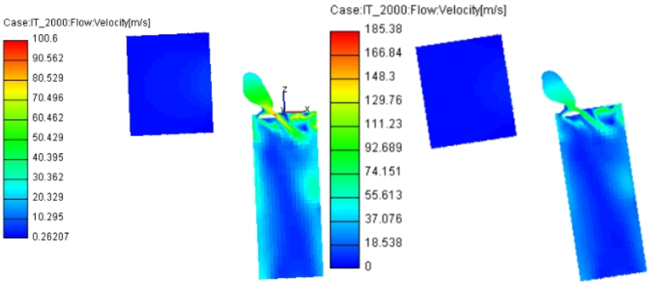
Table 4. Pressure Setting Parameters for the Experimental and Control Groups

Name	Pressure(Pa)
Test(experimental group)	100000
Test2(control group)	99000
Test3(control group)	101000
Test4 (control group)	102000
Test5(control group)	103000

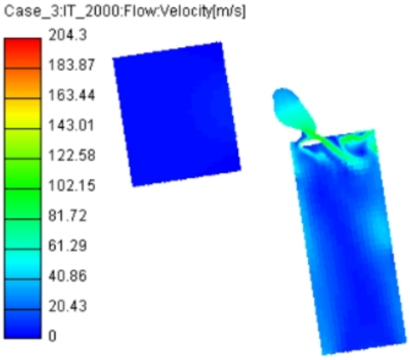
As shown in the Figure 9, every change of 1000Pa has a significant impact on the flow velocity in the cylinder. The maximum flow velocity appears at the valve port, and the experimental results show that the peak value at 100000Pa is 85.113m/s, achieving the goal of minimizing intake resistance as much as possible. Increasing or decreasing the pressure based on this pressure will increase the peak value.



(a) 99000Pa flow rate chart (b) 100000Pa flow rate chart



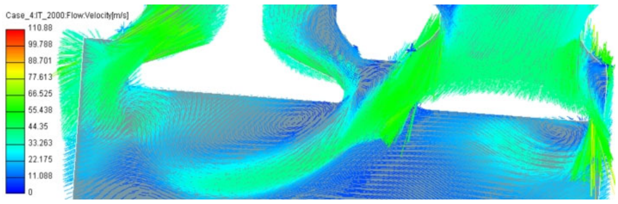
(c) 101000Pa flow rate chart (d) 102000Pa flow rate chart



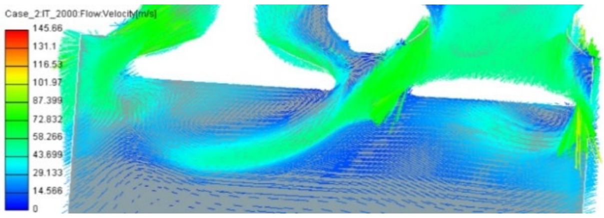
(e) 103000Pa flow rate chart

Figure 9. Flow velocity diagram under different pressures

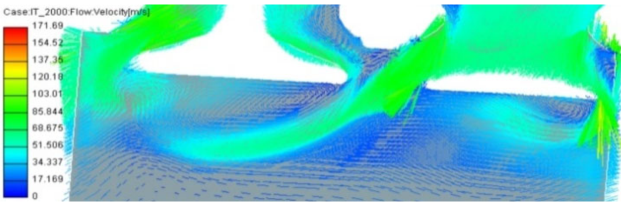
Next, it is necessary to observe the forms of turbulence and the magnitude of turbulence velocity under different pressures. Find the optimal pressure to maximize power and reduce airflow resistance. Figure 10 shows a zoomed in view of the local flow velocity under different pressures. According to the five control charts, it can be seen that under higher boundary pressure conditions, the right vortex is more pronounced, and the faster the speed, the greater the resistance, which affects the intake effect. The maximum speed in the chart appears near the right vortex.



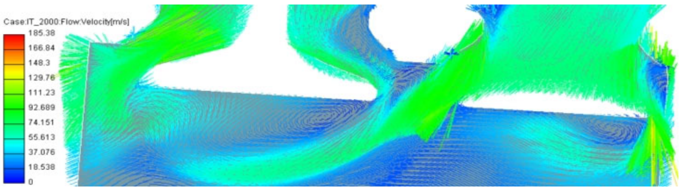
(a) Enlarged image of 99000Pa downstream flow rate



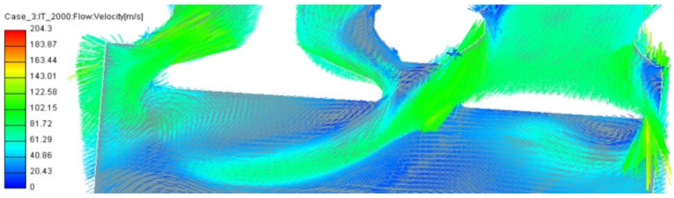
(b) Enlarged image of 100000Pa downstream flow rate



(c) Enlarged image of 101000Pa downstream flow rate



(d) Enlarged image of 102000Pa downstream flow rate



(e) Enlarged image of 103000Pa downstream flow rate

Figure 10. Enlarged diagram of flow velocity under different pressures

Through Figure 11 and Figure12, the average dissipation rate and average turbulent kinetic energy curves under different pressures are analyzed. Under the same external conditions, the higher the pressure, the greater the average dissipation rate and average turbulent kinetic energy.

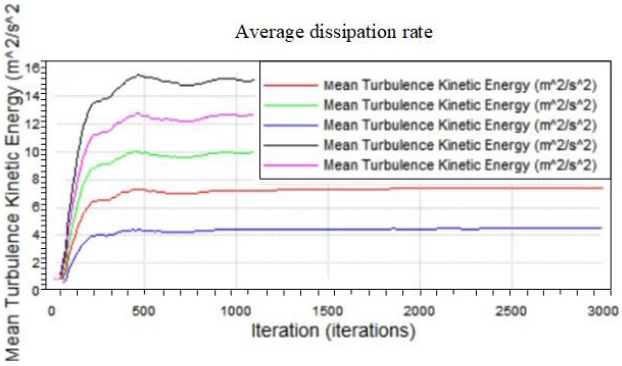


Figure 11. Average dissipation rate curve under different pressures

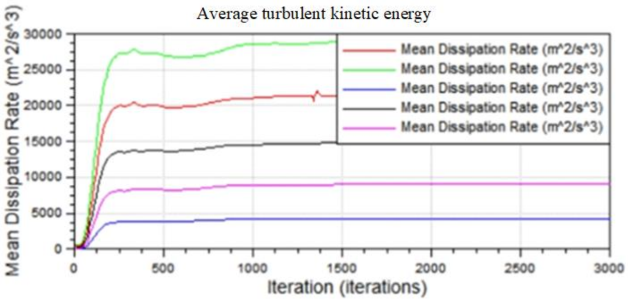


Figure 12. Average turbulent kinetic energy curve under different pressures

4. Conclusion

The main content of this article is to study the influence of external conditions on the airflow flow inside the intake duct, as well as to optimize and design the intake stroke and intake duct structure. Construct a 3D model using CATIA, and perform mesh partitioning and simulation experiments using AVL software. Compare the simulation results to determine the impact of external factors on the cylinder airflow. The experimental preparation work and final results are as follows:

1. Use CATIA to create a 3D model, then import the model into AVL fire workflow manager for subsequent mesh processing. Finally, set various parameters to calculate and simulate the flow of airflow in the intake pipe and cylinder, and obtain velocity and pressure maps. Draw various parameter curve graphs in the impression chart for analysis.

2. While keeping other external conditions constant, change the temperature and pressure in sequence. Study and observe their impact on cylinder flow. The following are the experimental results:

(1) Temperature has little effect on cylinder flow to a certain extent and is not being studied as an important factor.

(2) The pressure group experiment has a significant effect, and changing the pressure has a significant impact on the flow in the cylinder. Analyzing the experimental results, it can be concluded that Sthe gas resistance in the cylinder reaches the optimal value at a pressure of 100000Pa.

References

[1] Zhao Y L, Hong W, Li H, et al. Optimization of stratified EGR intake port in gasoline direct injection engine, Automotive engines, 2015, (02), 61-65.

[2] Taylor III W, Leylek J H, Tran L T, et al. Advanced computational methods for predicting flow losses in intake regions of diesel engines, SAE transactions, 1997, pp, 964-980.

[3] Kim J N, Kim H Y, Yoon S S, et al. Effect of intake valve swirl on fuel-gas mixing and subsequent combustion in a CAI engine, International journal of automotive technology, 2008, 9, 649-657.

[4] Huh K Y, Choi C R, Kim J G. Flow analysis of the helical intake port and cylinder of a direct injection diesel engine, SAE transactions, 1995, 104, 1993-2006.

[5] Yi J, Wooldridge s, Coulson G, et al. Development and Optimization of the Ford3.5LV6 EcoBoost Combustion System, Mycoses, 2009, 2(1), 1388-1407.

[6] Godrie P, Zellat M, Zellat M. Simulation of flow field generated by intake port-valve-cylinder configurations-comparison with measurements and applications, SAE transactions, 1994, pp, 803-823.

[7] Schurov S M, Collings N. A numerical simulation of intake port phenomena in a spark ignition engine under cold starting conditions, SAE transactions, 1994, pp, 853-868.

[8] O Bailly C, Buchou A, Floch L, et al. Simulation of the Intake and Compression Strokes of a Motored 4-Valve Si Engine with a Finite Element Code, Oil & Gas Science and Technology, 1999, 54(2).

[9] Luo M J, Huang Z, Chen G H, Three-dimensional transient numerical simulation of engine intake flow, Acta Aerodynamica Sinica, 2005, (01), 74-78+83.

[10] Pei P C, Fan Y J, Liu S L. An Investigation on the Tumbling Intake Ports Configuration for Multi-Valve S．I．Engines, Transactions of CSICE, 1998, (03), 79-85.

[11] WANG Q, M C C, S X, Utilizing3D CFD to Calculate ICE Inlet Valve Flow Coefficient, Vehicle Engine，2002, (06), 18-20.

[12] L Y, P P C, C P X, Design And Research On New Gasoline Direct Injection Engine Tumble Conbustion Chamber, Journal Of Xian JiaoTong University, 2012, 46.

[13] Y X F, L J L, J D M, Three-Dimensional Simulation Calculation andExperimental Study of Flow in the Port of IC Engines, Transactions Of Csice, 1990, (02), 111-117.\

[14] Kong X H, Optimization Design a 1.5L Gasoline EngineIntake Port Based on AVL-Fire, Equipment Manufacturing Technology, 2021, (02). 67-70.