Flow Field Analysis and Parameter Optimization of Marine Diesel Engine Diluter

Qin Wang, Xusheng Zhang

Merchant Marine College, Shanghai Maritime University, Shanghai, 201306, China.

Abstract

It is necessary that the measurement of diesel particulate matter PM requires a dilution system in the marien engine emission regulation. Understanding the internal flow in an ejector dilutor is important to improve the dilutor design. In this paper, experimental and simulation study on the performance of the ejector dilutor were carried out. The real dilution ratio was measured in the experiment based on the designed dilutor. A three-dimensional (3D) model of the dilutor was created and the internal flow field was extracted to proceed the CFD simulation using Ansys Fluent. Suitable simulation method is obtained based on a good agreement between the simulation results and the experimental results. In addtion, the effects of throat inlet angle and the throat width on the internal flow and the dilution ratio were studied.

Keywords

Diesel Engine; Exhaust Dilutor; CFD Simulation; Turbulence Model.

1. Introduction

With the rapid development of the shipping industry, diesel engines have become an important source of enviromental pollution. Particulate matter (PM) emissions in diesel engine exhaust can have a significant impact on human health. There are obvious differences between large marine diesel engines and automotive four-stroke diesel engines: the types of diesel engines are different (two-stroke, slow technology update), different fuel characteristics (heavy marine diesel), and less Aftertreatment System. These differences result in high particle specific mass emissions of large marine diesel engines, and the PM composition is more complex [1-3]. In the PM measurement, due to the large displacement of marine diesel engines [4], partial flow dilution sampling is generally selected and partial flow ejector diluter has been widely used in exhaust particle dilution due to its small size, stable and reliable performance, and has been tried and applied in real ship experiments[5].

The dilution ratio of a typical ejector diluter is mainly constant, which limits its wider use,especially for the high dilution ratio in the measurement of black carbon emissions of ships (multi-stage series connection). As there is the ejection of high-speed airflow inside the dilutorand the formation of high-speed airflow is closely related to the structure of the diluter. Although one-dimensional simulations have confirmed that the gas flow velocity inside the diluter is close to the speed of sound [6], there are few studies on the relationship between the flow and structure of the diluter due to the high-speed air flow, and the simulation is limited to incompressible flow [7].

In this paper, an exhaust diluter is a protype based on the ejection principle, and the three-dimensional (3D) internal flow field model of the diluter is extracted. Under given conditions, AnsysFluent is used to simulate the internal flow field of the diluter, and the simulation methods are compared to the experimental results. Based on the suitable simulation methods, the effects of different designs on the internal air flow and dilution ratio are discussed.

2. Model of the diluter

2.1 The establishment of the geometric model of the diluter

The diluter is designed based on the principle of high-speed jet ejection, and the structure of the gap is designed to achieve mixing and dilution of air and gas. The Fig. 1 shows the 3D structure of dilutor, which mainly includes an exhaust air inlet, a compressed air inlet, a mixing chamber and a sampling port. Compressed air enters from the dilution air inlet and passes through the gap to generate a negative pressure zone to eject the exhaust gas from the exhaust inlet into the diluter, and both gases enter the dilution chamber to mix.

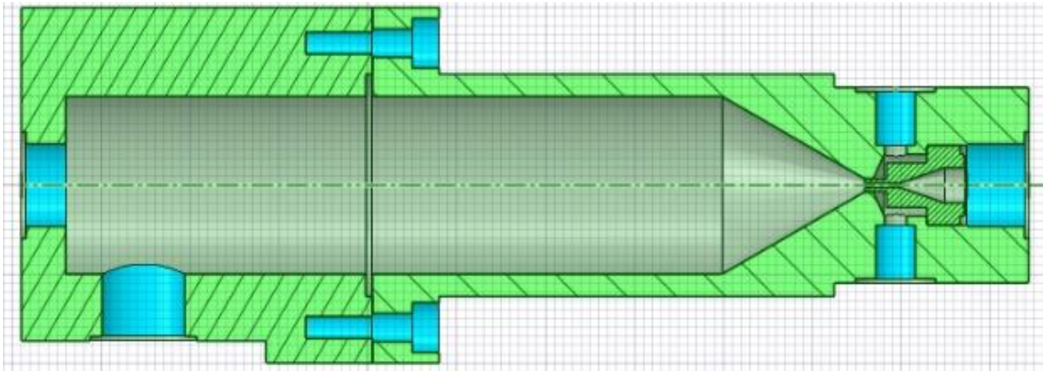
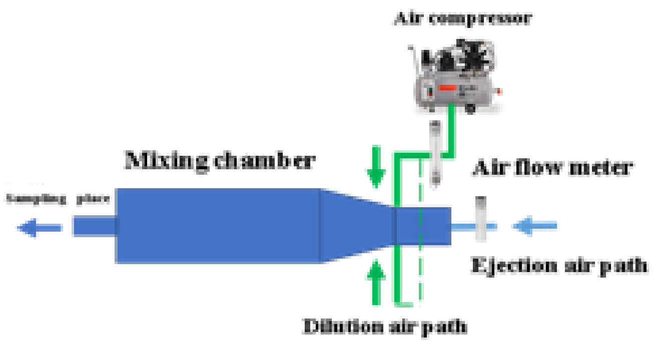
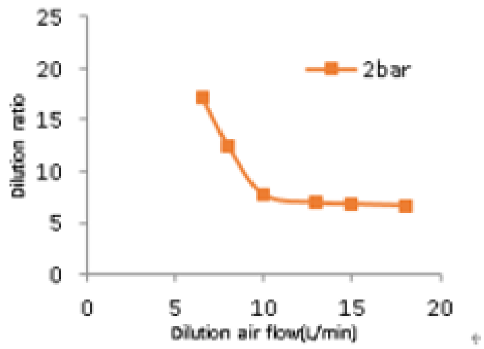


Fig. 1 Geometric model of the diluter

2.2 Test instrument, device and method

The schematic diagram of the test device and test results are shown in Fig. 2 An air compressor was used to provide filtered dilution air at a certain pressure. The flow rate of dilution air was meaured by a flow meter. The ejected gas was also introduced into the diluter after passing through a flow meter.





(a) (b)

Fig. 2 Schematic diagram of the experiment system and the experimental results

The test principle is based on setting the pressure of the dilution air inlet, changing the dilution air flow through the regulating valve of the flow meter inlet, studing the variaration of the ejection gas inlet flow, and calculating the dilution ratio of the dilutor. The formula for calculating the dilution ratio N is:

$$N = \frac{Q_{air} + Q_{sample}}{Q_{sample}}$$

In the formula, Qair means the flow rate of dilution air; Qsample means the flow rate of ejected gas.

The test was carried out at a temperature of 25°C, and the pressure conditions was standard atmospheric pressure except for the adjustable dilution air inlet. The dilution air was set to the pressure of 2.0 bar, and six flow rates were tested. In present experiments, air was selected as the ejected gas for the simple operation. The relationship between the dilution air inlet flow rate and the dilution ratio is shown in Fig. 2 It can be seen that the dilution ratio decreases with the increase of the dilution air inlet flow rate. When the dilution air flow reaches a certain value, the dilution ratio is around 7.

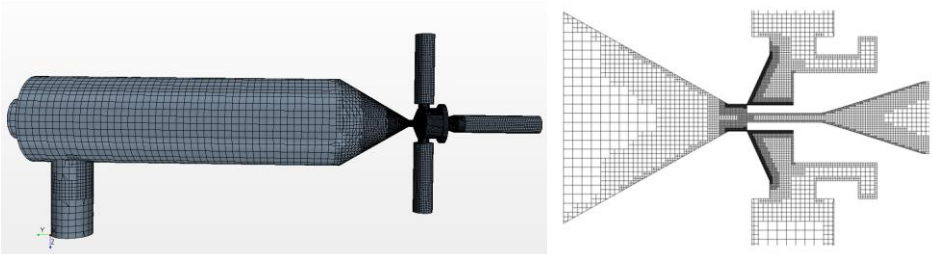
3. Verification of the diluter simulation model

3.1 Calculation conditions of internal flow field

The internal flow of the diluter is turbulent flow. In addition, the flow velocity at the throat is close to the speed of sound, so compressible fluid is used for calculation.In Ansys FLUENT, compressible fluid is used as the flowing medium of the diluter flow field and the RNG k-e turbulence model is combined with the Couple solution algorithm to simulate and calculate the internal flow field of the diluter.

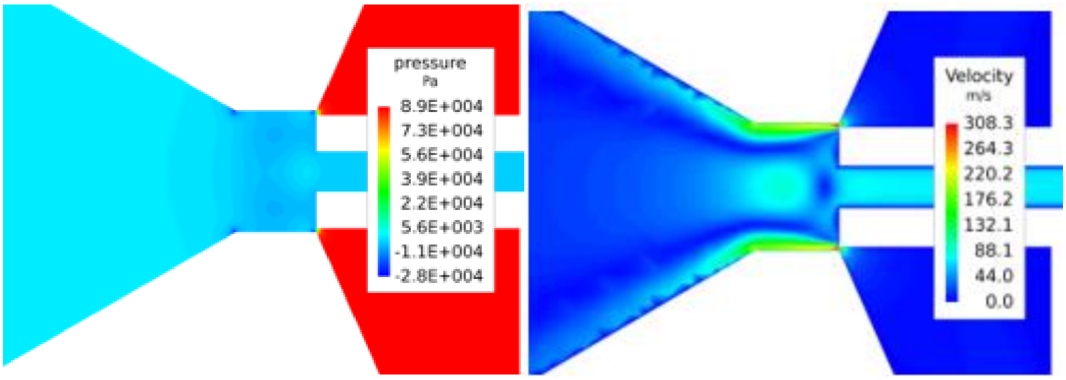
3.2 Analysis of the internal flow field of the diluter

STAR-CCM+ software was used to create a hexahedral mesh for the diluter , with 1.27 million grids, as shown in Fig. 3.

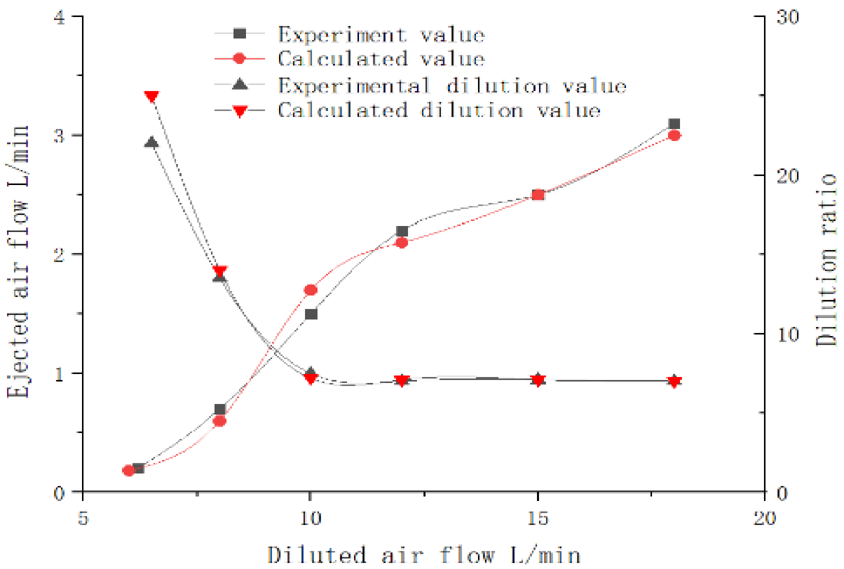


(a) (b)

Fig. 3 Result of the diluter grid division



(a) (b)



(c)

Fig. 4 Simulation and calculation results

The simulation condition of the dilution air inlet velocity of 20.6m/s was selected to analyze the internal flow field of the diluter. It can be seen from Fig. 4a and Fig. 4b that the outlet pressure and velocity distribution are reasonable. The calculated dilution air flow rate is 17.5 l/min, the ejected air flow rate is 3.06 l/min, and the dilution ratio is 6.72. The error is around 2%.

Based on the establishment of CFD calculation method, other operating conditions of the experiment are calculated, and the results are finally summarized, as shown in Fig. 4c The simulation results have a good agreement with the experimental results. Therefore, the calculation model and method can reproduce the actual operation process of the diluter, and can be used for simulation and prediction the internal flow.

4. Research on optimization of diluter parameters

Based on the establishment of a simulation calculation model, the effects of the cone angle in front of the throat of the diluter and the throat gap on the internal flow are calculated.

4.1 The influence of the front cone angle of the throat

The current cone angle of the diluter is 24°. Other two cone angle (30° and 18°) were selected to calculated without changing other parameters The Fig. 5 shows the 3D modles and simulation results.

It can be seen that the trend of the pressure cloud diagram is completely consistent, the cross-section has good vertical symmetry. There is a high pressure area at the nozzle outlet, and a low area (spiral movement center) above and below the nozzle outlet. Although the cone angles are different, the distribution of the velocity contour images is basically same. The difference between the maximum and minimum speed values is only 1.5%, and the throat flow corresponding to the two cone angles can be considered the same.

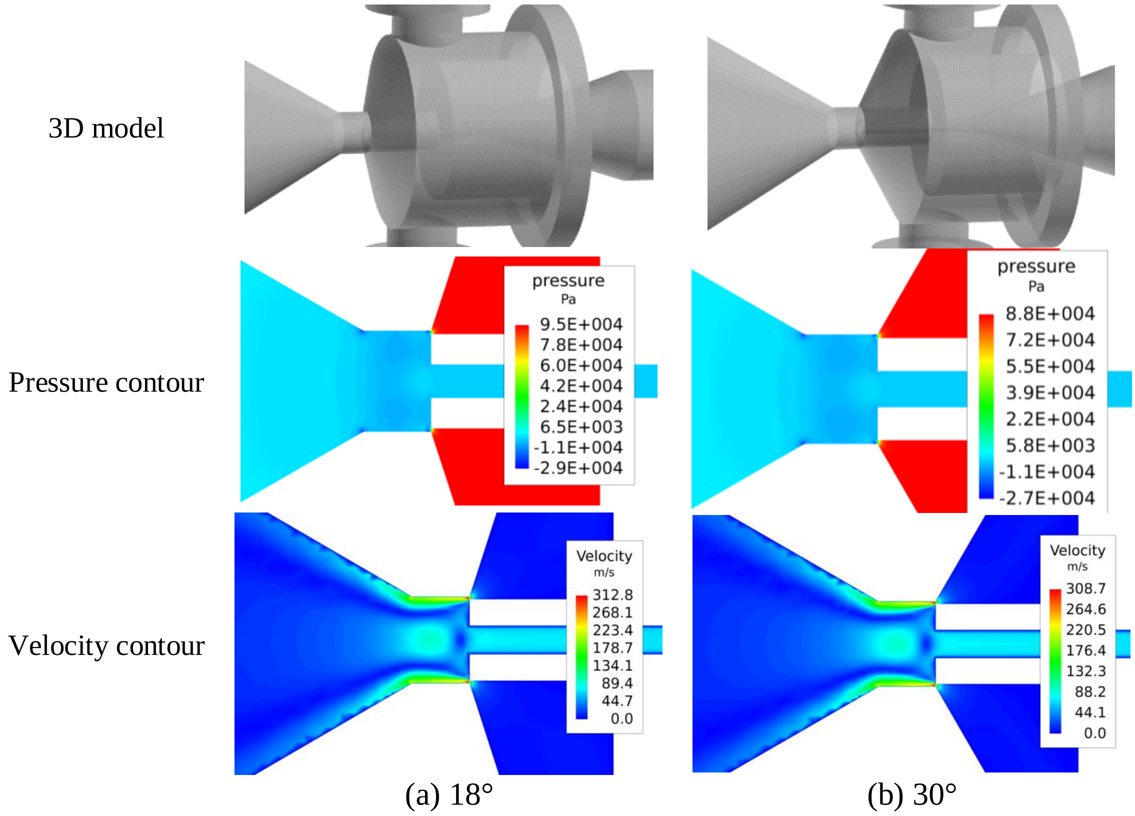


Fig. 5 3D model and calculation results

The Table 1 lists the statistic data of exhaust gas flow and dilution ratio The dilution ratios under the three schemes are basically the same, so it can be concluded that the cone angle has no effect on the performance.

Table 1. Statistics of exhaust gas flow and dilution ratio

Numble	18°	24°	30°
Exhaust gas flow L/min	2.972	2.966	2.965
Dilution ratio	6.89	6.90	6.90

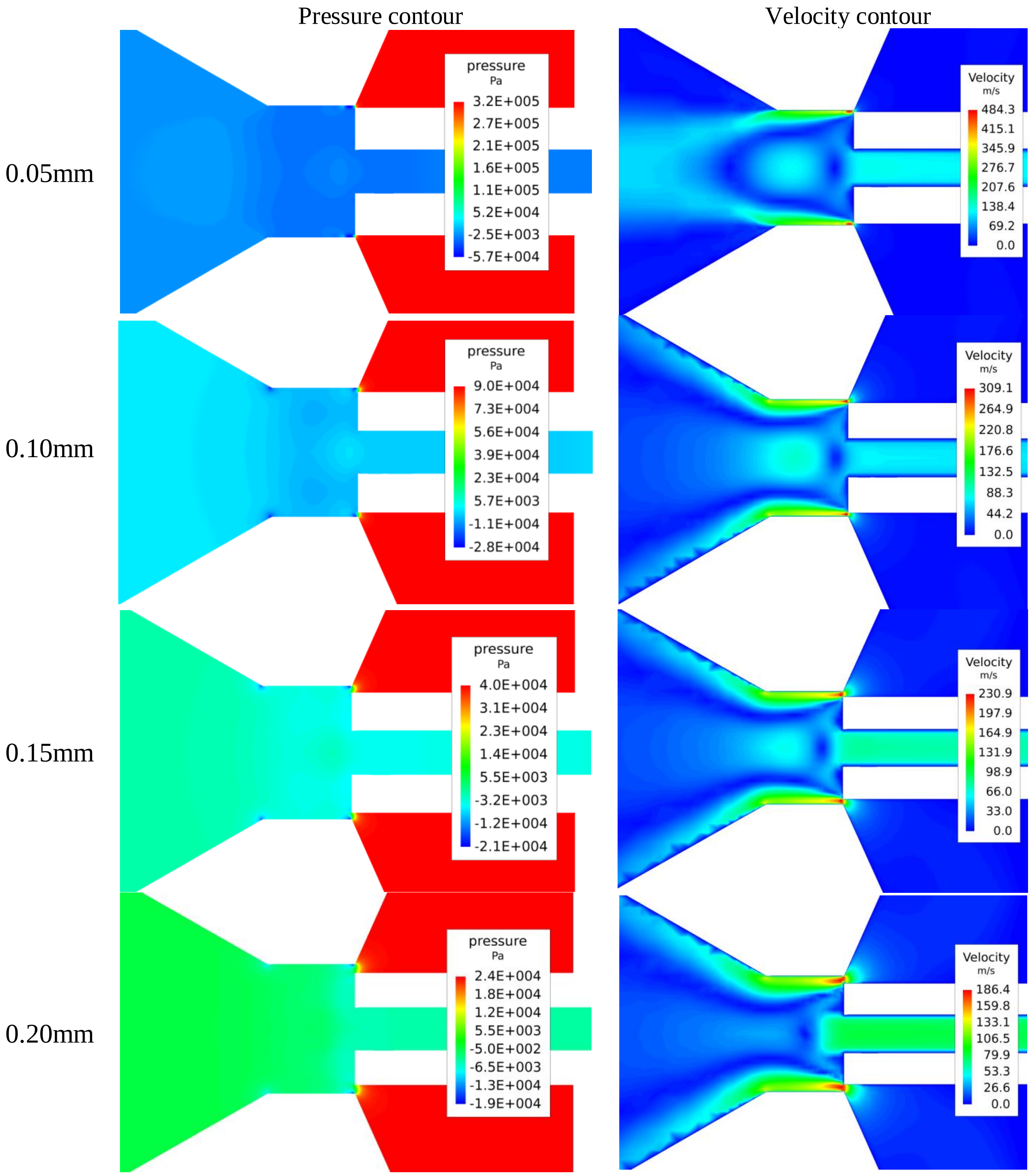


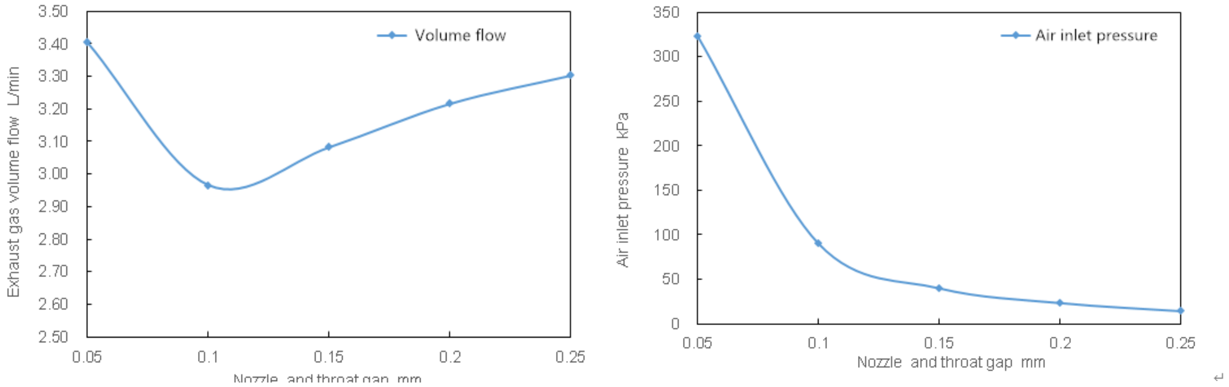
Fig. 6 Pressure and velocity contours under different gap width

4.2 The effect of the throat gap

The current gap between the nozzle and the throat of the diluter is 0.1mm. Three gaps(0.05mm, 0.15mm and 0.20mm)are selected for simulation. the simulation results are shown in Fig. 6.

It can be seen from Fig. 6 that as the gap width decreases, the pressure at the inlet continues to increase, and the negative pressure at the throat also decreases. Under 0.1mm, the pressure contour at the throat changes more obviously. When the gap increases, there are smaller pressure values at the gap outlet and the corner of the expansion tube.

For the four different throat widths, the velocity contour have good symmetry, and the general trend is basically the same. There are high-speed airflows around the throat, and there is a high-speed gas flow area in the middle of the throat. Under the 0.05mm gap, there is a clear high-speed flow area in the middle area behind the throat. As the gap decreases, the maximum flow rate keeps increasing because the total gas flow does not change. In order to compare the flow under different gaps, the calculation results of the relationship between the exhaust gas flow and air inlet pressure with the size of the gap are shown in Fig. 7.



(a) (b)

Fig. 7 The relationship between exhaust gas flow / compressed air inlet pressure and clearance

It can be seen from Fig. 7 that if the gap of the diluter throat is reduced, the ejection effect is better, but too high dilution air pressure is required; if the gap is enlarged, the ejection effect is slightly better, but the dilution air pressure is significantly reduced. Taking into account the two factors of exhaust gas flow and inlet pressure, a gap in the range of 0.1-0.15mm can be selected.

5. Conclusion

Experiment and simulation of the flow field in the diluter were carried out base on the designed dilutor. The distribution diagram of the internal flow field of the diluter is obtained, and the following conclusions are obtained:

(1) The experiment of ejector diluter shows that when the dilution air flow reaches a certain level, the dilution ratio keeps stable.

(2) Using compressible fluid as the flowing medium of the diluter flow field and the RNG k-e turbulence model with the Couple solution algorithm, the simulation results are in good agreement with the experimental results.

(3) In the present study, the change of the diluter cone angle has no effect on the performance.

(4) If the gap between the throat of the diluter is reduced, the ejection effect is better, but too high dilution air pressure is required; if the gap is enlarged, the drainage effect is slightly better, but the dilution air pressure is significantly reduced.Taking into account the two factors of exhaust gas flow and inlet pressure, a gap in the range of 0.1-0.15mm can be selected.

References

[1] Cooper, D.A.. Exhaust emissions from ships at berth[J]. Atmospheric Environment, 2003.37: 3817~3830.

[2] Kasper, A., et al.. Particulate Emissions from a Low-Speed Marine Diesel Engine[J]. aerosol Science and Technology. 2007. 41(1):24~32.

[3] Agrawal, H., et al.. In-use gaseous and particulate matter emissions from a modern ocean going container vessel[J]. Atmospheric Environment. 2008.42:5504~5510.

[4] GB 15097-2016, Limits and measurement methods for emissions of pollutants from marine engines (China's first and second phases) [S].

[5] Lou Diming, Bao Songjie, Hu Zhiyuan, Tan Piqiang. Tug speed optimization based on actual ship fuel consumption and emissions[J]. Journal of Traffic and Transportation Engineering. 2017, 17(01), 93-100.

[6] Barouch Giechaskiel, Leonidas Ntziachristos and Zissis Samaras, Calibration and modelling of ejector dilutors for automotive exhaust sampling[J]. Meas. Sci. Technol. 2004, 15: 2199–2206

[7] Nie Xuejun, Yue Senfeng, Gong Yanxia. Flow field analysis and parameter optimization of the diluter in the exhaust gas detection system[J]. Environmental Science and Technology. 2013, 36(12): 324~327.