Simulation Study on the Internal Flow in the Injector Nozzle of a Low Speed Diesel Engine

Yuqin Xu, Xusheng Zhang and Zhengwei Jiang

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

Abstract

Fuel injection and evaporation have significant effects on the subsequent combustion and emissions in diesel engine. Asymmetric spray characteristics of low-speed diesel engine is highly related to the cavitation formation in the injector nozzle. Therefore, it is necessary to investigate the nozzle cavitation for understanding the spray development in the low-speed diesel engine. Based on the classic low-pressure experiments of internal flow in the nozzles with different sizes, validation of cavitation models and simulation method was carried out according to the flow rate, flow speed distribution and cavitation structure. Finally, the simulation method was used to study the internal flow in a slide-valve nozzle, simulation results had good agreement with the experimental results. It is proved that this simulation method could be used to predict the two-phase flow in the slide-valve injector.

Keywords

Low-Speed Diesel Engine; Slide Valve Nozzle; Cavitation Simulation; Internal Flow.

1. Introduction

With the development of global shipping industry, the total amount of pollutants exhausted by ships is significantly increased. The harmful effect of the exhaust has been paid more and more attention by the international community and the Chinese government, among which gas emissions from large vessle account for the primary air pollutants [1]. Therefore,it is urgent to control the emission of large two-stroke marine diesel engines in large ships.

Generally speaking, the emission control of conventional diesel engine can be divided into two ways: inside and outside the engine. For large marine diesel engines, the fuel spray and combustion process must be optimized with the in-cylinder control method chosen. Fuel injection and evaporation processes largely determine the formation of subsequent combustion processes and emissions, so the first step is to understand the spray characteristics of large Marine diesel engine fuel in order to control in-cylinder combustion. In recent years, it has been found that there is a significant asymmetry structure in the diesel spray, and this feature is closely related to the cavitation inside the nozzle [2, 3]. In addition, the variation of the structure of the slide valve and the nozzle sac leads to the change of the internal flow, which leads to the difference of the injection velocity [4]. The study of double-hole nozzles found that the flow distribution in the two holes was different, and the decrease of flow velocity led to the asymmetry of spray [5]. The spacial structure, internal flow and nozzle cavitation of low-speed engine have a significant influence on the development process of spray. Therefore, investigation on the internal flow and nozzle cavitation is of significant importance to reveal the cavitation mechanism of spray and optimize the nozzle structure.

In this paper, the internal cavitation characteristics of the slide valve nozzle are taken as the research object. Based on the classical cavitation experiment of simple nozzle, the simulation method is confirmed according to the nozzle flow rate, orifice velocity and cavitation morphology. Then the internal flow of the nozzle of a low-speed engine is studied.

2. Verification of Cavity Model of Nozzle Flow Rate and Velocity

Cavitation in the nozzle will significantly affect the nozzle flow rate and velocity distribution in the nozzle hole, so the model is studied by comparing these two parameters. The experimental result is

Winklhofer's classic low-pressure internal flow experiment [6]. The structural dimensions of the nozzle are: hole length 1mm, hole thickness 0.3mm, fuel inlet 0.301mm, fuel outlet 0.284mm (aperture close to 0.3mm), and inlet fillet radius 0.02mm. Two-dimensional symmetric model is used for analysis, and local refine treatment is carried out on the wall and inside the nozzle hole, as showed in Fig. 1. After grid independent analysis, it is determined that the number of grids is 32390.

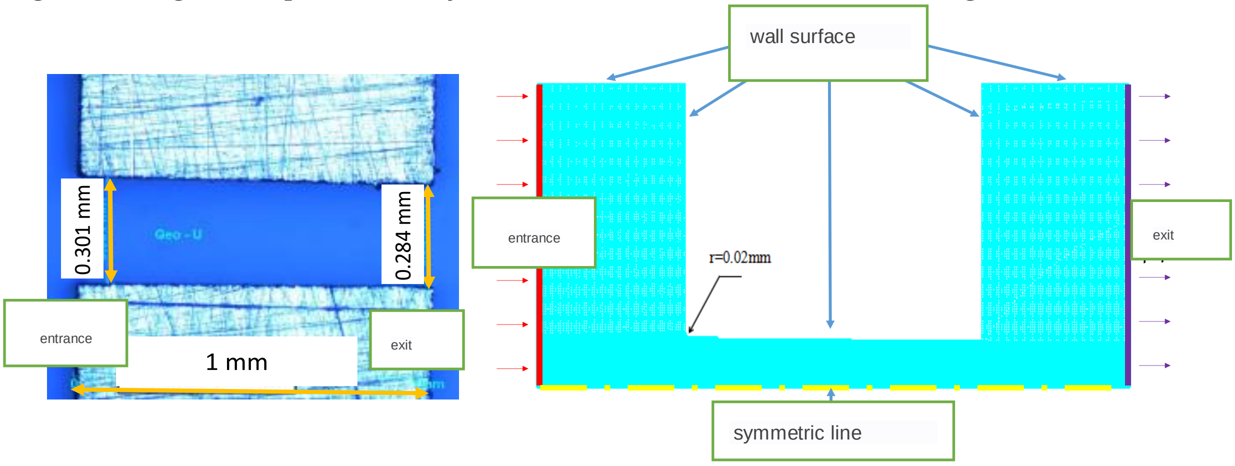

Fig. 1 Simulation geometric model diagram

The injection fuel was light diesel. Schnerr-sauer model, ZGB model and Singhal full cavitation model are provided in the Ansys Fluent software. Based on the experimental results of the nozzle supercavitation condition in the experiment, the simulation results using three cavitation models are shown in Fig. 2. It can be seen that the total cavitation model of Singhal has good agreement with the experimental results, which are consistent with other research conclusions [7]. The specific model settings in the simulation calculation are shown in Table 1.

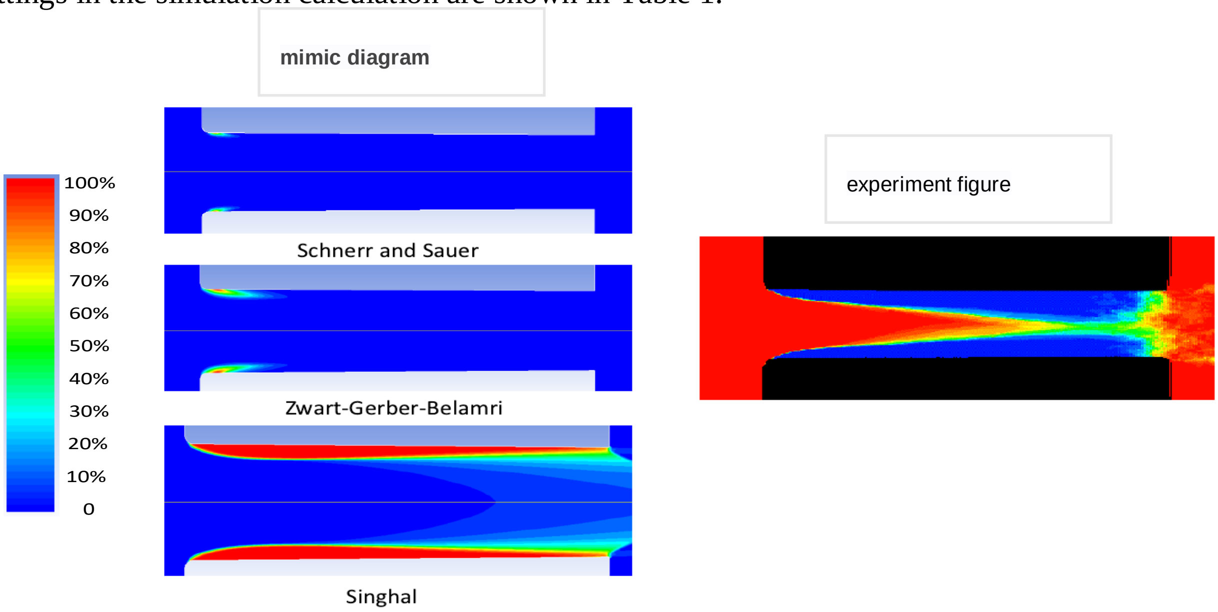

Fig. 2 Comparison of calculation results of different hole models

Table 1 Selection of simulation model

Set the item	model
Two-phase flow model	Mixture model
The turbulence model	Realizable k - epsilon model
The cavitation model	Singhal full cavitation model
Entry/exit boundary	Pressure boundary
Pressure correction equation	The SIMPLEC/PRESTO algorithm

The pressure setting in the simulation is consistent with the experiment. The inlet pressure is kept at 10MPa and the outlet pressure is changed between 1 and 8 MPa.As can be seen from Fig. 3, the deviation of the flow rates between experiment and simulation is less than 5%, and the simulation is in good agreement with the experimental gas-liquid two-phase distribution. When the pressure difference is over 7MPa, the supercavitaiton is completely formed, and the flow rate remains almost unchanged.

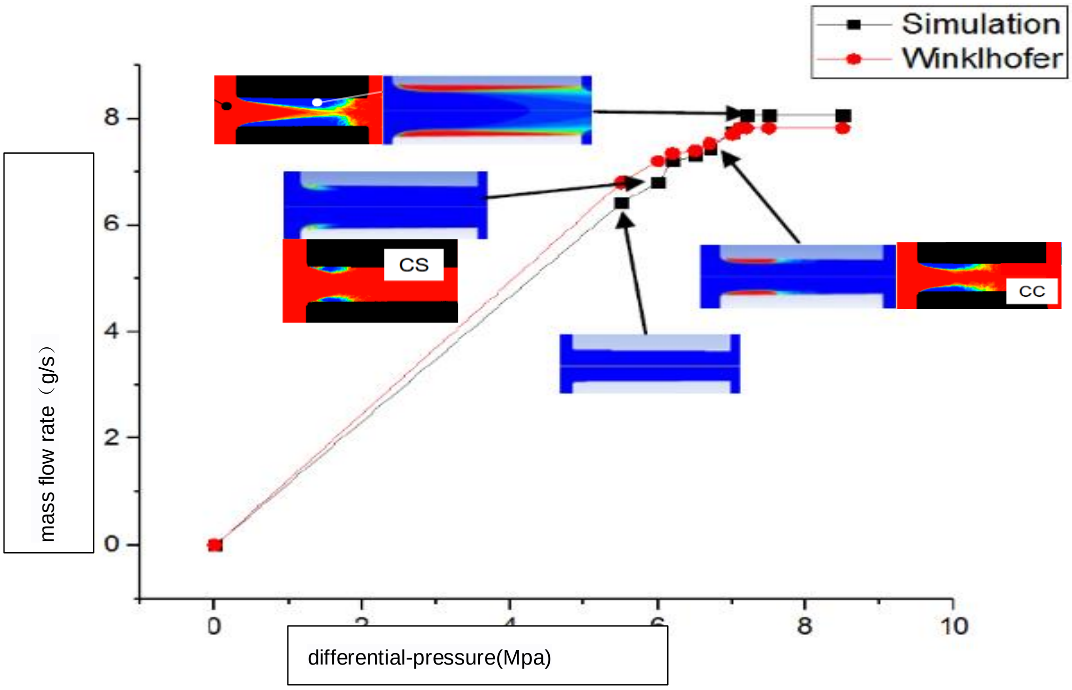

Fig. 3 Comparison of flow rate between simulation and experiment

Furthermore, the simulated velocity distribution in the hole was compared with the measurement at 53 mm away from the nozzle entrance. The comparison results are shown in Fig. 4. At two different pressures, there is slight deviation between the simulation results and the experimental results, but the overall trend is basically consistent.

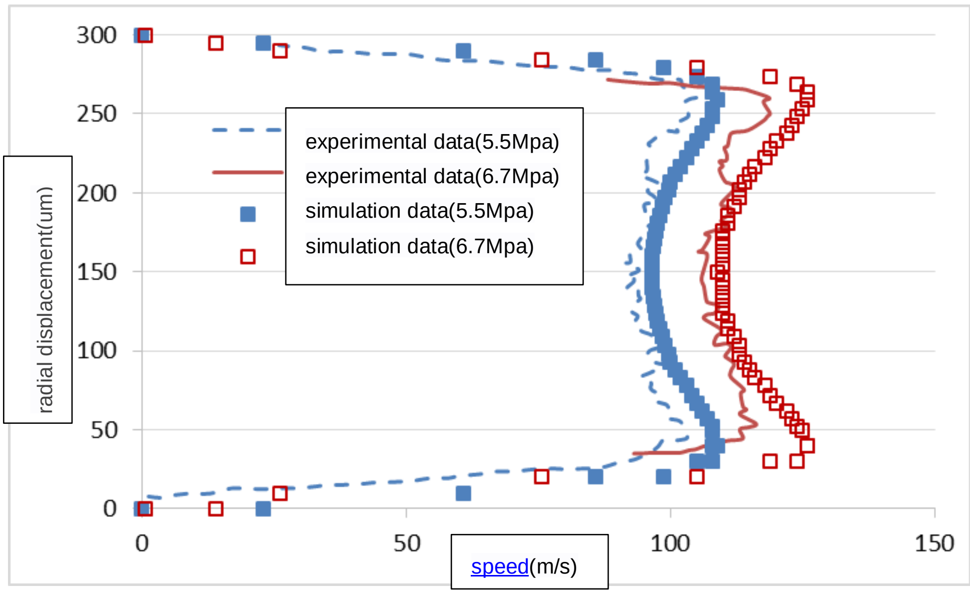

Fig. 4 Comparison of simulation and experimental results of nozzle internal velocity

In addition, it is clear that the cavitation area in the experiment is larger than that in the simulation, which may be related to the difference of fuel parameters between simulation and experiment and the selected test method is an average method. Aiming at the difference between simulation and experiment, another set of typical internal flow experiments was selected for further verification.

3. Validation of cavitation model for hole structure

In order to evaluate the influence of the selected calculation model on the characteristic scale of nozzle internal cavitation, the experimental results of Sou et al. were selected. A high-speed camera was used to capture the liquid flow in a two-dimensional transparent nozzle at low pressure. The width, length and upstream area width of the rectangular nozzle are 4.21mm, 16mm and 32mm respectively.

The model is calculated by a symmetrical model, water at 308K is selected for calculation in the simulation. The nozzle geometry model and grid are shown in Fig 5. After grid independent judgment, the number of selecting computational grids is 31000. Simulation model is the similar to Table 1.

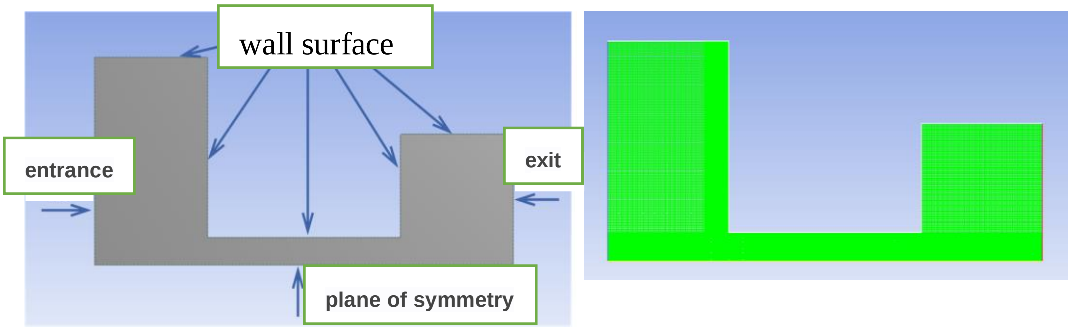

Fig. 5 2D simulation model of sou's experiment

2D numerical simulation was carried out to analyze the gas phase volume fraction cloud map including cavitation initiation, developed cavitation and supercavitation, as shown in Fig. 4 (The Reynolds numbers corresponding to conditions A, B, C and D were 50000, 58000, 64000 and 70000, respectively).

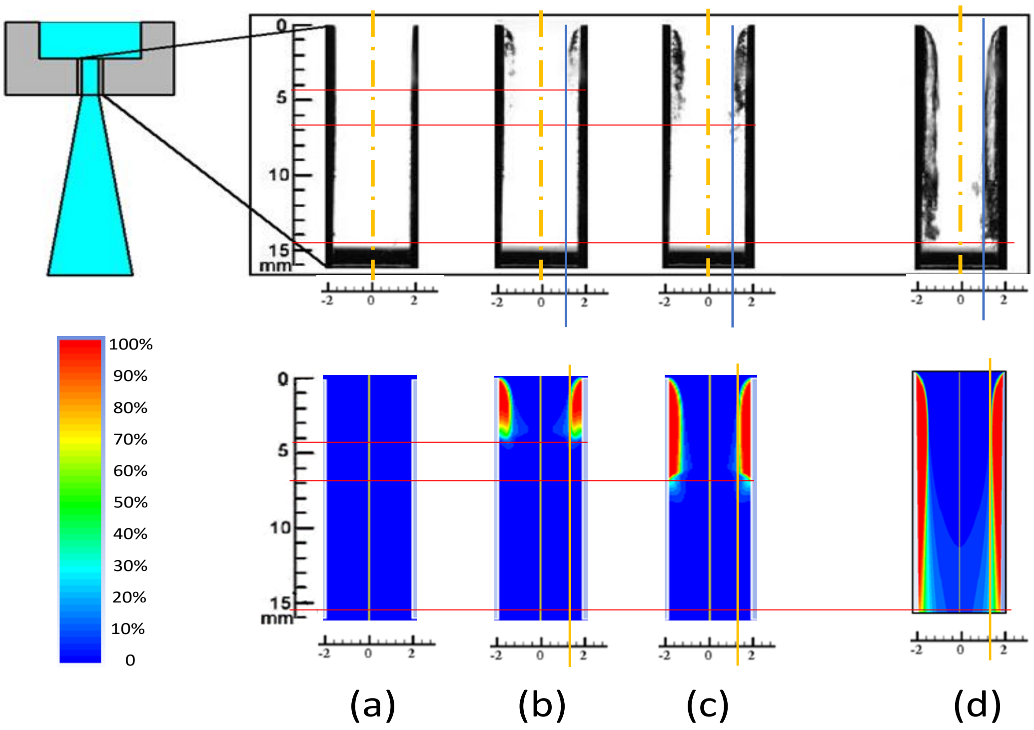

Fig. 6 Cavitation structure in the hole of experiments and simulation

In order to compare the length and width of the cavitation layer more accurately, the layer parameters are measured and compared. It can be seen that under the four experimental conditions, the simulated cavitation layer are in good agreement with the experimental results. With the increase of Reynolds number, the length of the cavitation layer increases significantly, but the maximum width of the layer remains basically unchanged. Furthermore, little changes was found for the inlet chocking area caused by the cavitation.

The method of calculating the internal flow of the nozzle is verified by parameters such as nozzle flow rate, flow velocity and cavitation structure. The results show that the simulation method can predict the occurrence and development of cavity inside the nozzle.

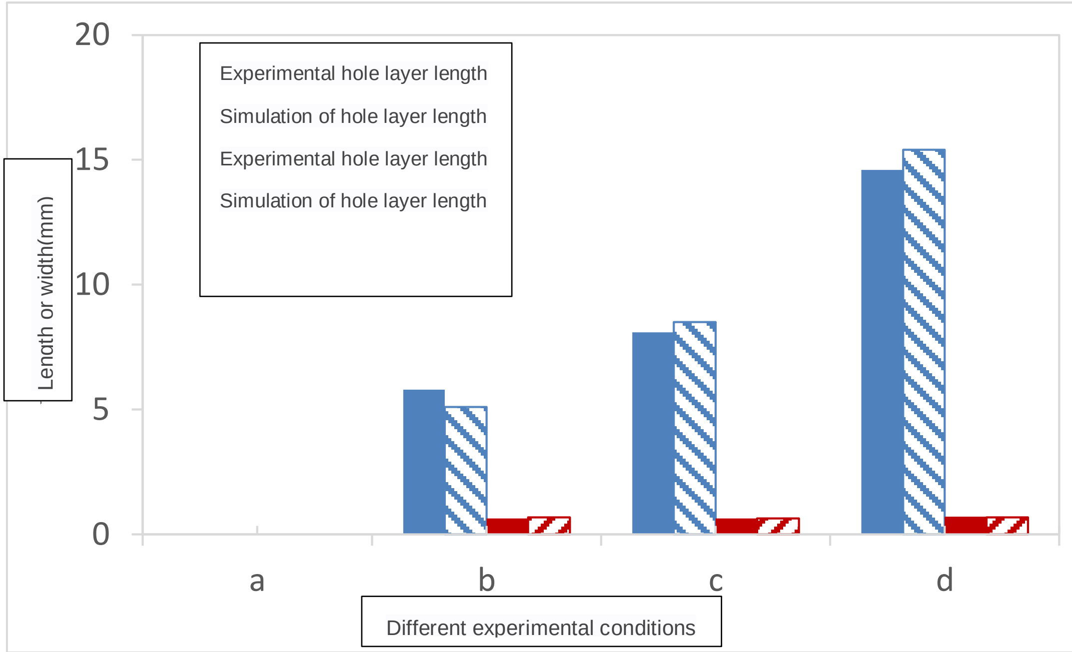

Fig. 7 Simulation and experimental results of the cavity layer under different experimental conditions

4. Simulation of Internal Flow in Nozzle of Large Low-Speed Engine

Modern large-scale low-speed engine mostly use slide valve injector, which can effectively reduce the exhaust emission of ship engines with the reduction of sac volume. Hult et al. used the nozzle geometry of the slide valve injector, and carried out an experimental study to visualize the internal flow using a transparent real-size nozzle [8]. Based on the experimental results, previous verified internal flow simulation method was used to calculate the internal flow of the this nozzle, and compare the calculated results with the experiment. The experimental diagram, computational 2D model and grid are shown in Fig. 8. After the grid independence is judged, the number of grids is 57745.

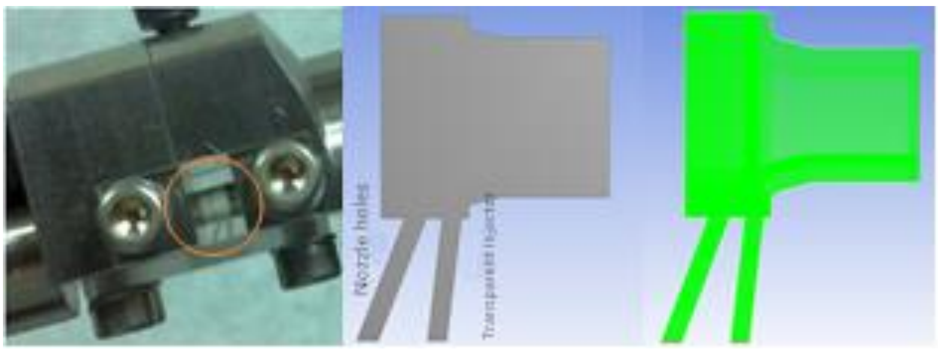

Fig. 8 Sliding valve nozzle model and grid diagram

4.1 Comparison of Cavitation Images with Different Cavitation Numbers

According to the pressure setting conditions in the experiment, light oil is selected as the simulation fluid (consistent with the experiment). Three different cavitation numbers (1.5, 2 and 4) are selected, as shown in Fig. 9 (a, b, and c correspond to cavitation numbers 1.5, 2 and 4 respectively). Numerical simulation results show that cavitation occurs in both nozzle holes, and the cavitation in the top and bottom nozzle holes is also generated on the left side of the nozzle hole entrance, and the cavitation strength of the bottom hole is significantly higher than that of the top hole. Compared with the experimental images, it is found that the cavitation is mainly located on the left side of the nozzle hole, and appears on both sides of the top hole entrance with greater intensity on the left side. The simulation results are in good agreement with the experimental results. When the number of cavities is 2, the cavitation intensity of the nozzle on the right side in the experiment is higher than the simulation result. This is mainly because the simulation only calculates the two-dimensional interface, and the actual nozzle inlet structure is three-dimensional. There may be deviations in the scale of the local structure, and the sac structure will significantly affect the formation of cavitation.

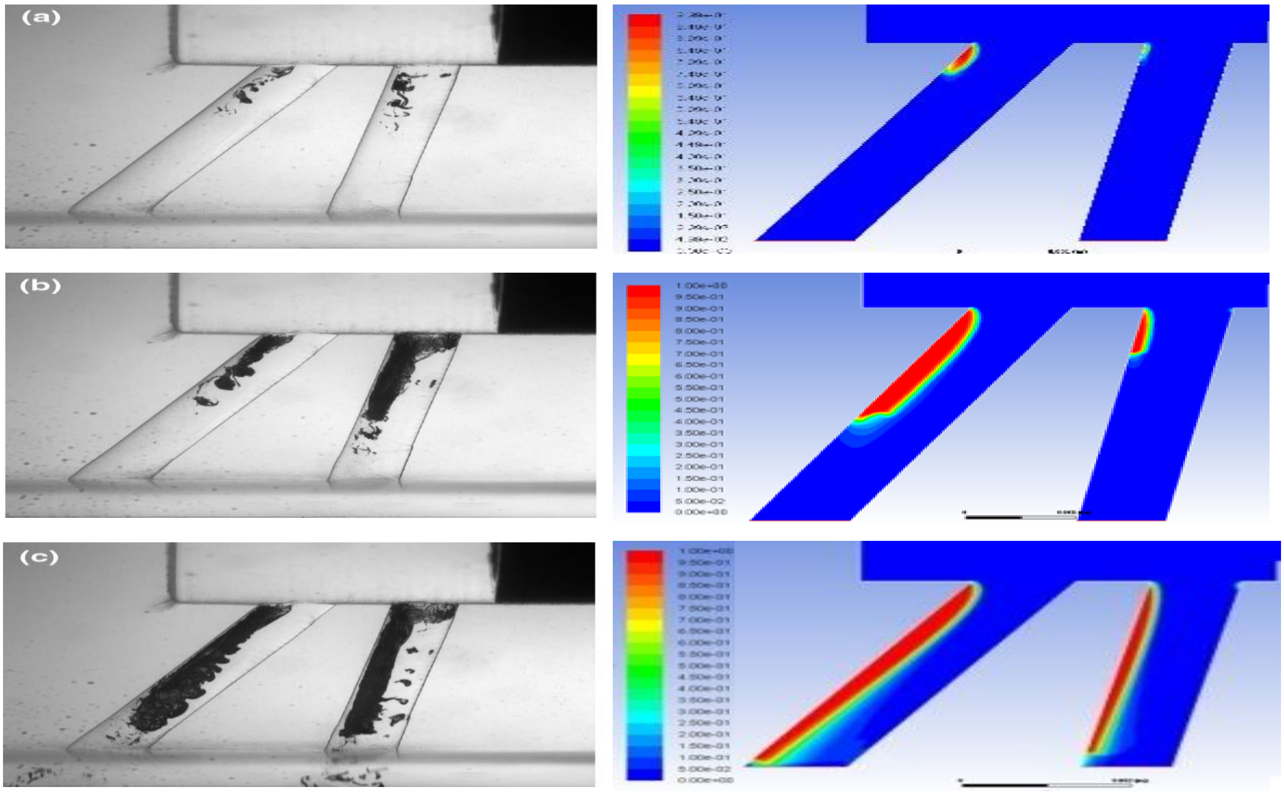

Fig. 9 Comparison of internal flow simulation and experimental results of dual-hole nozzle

4.2 Comparison of Simulation and Experimental Results of Internal Velocity

Hult's experiment measured the flow velocity in two nozzle holes at a distance of 0.1mm (L1) and 1.5mm (L2) from the nozzle hole (as shown by the green dotted line). The numerical simulation also compares the flow velocity distribution at the nozzle hole position 0.1mm and 1.5mm away from the nozzle entrance with the experimental data, as shown in Fig. 10.

At the L1 section, the simulation results and the experimental data are in good agreement at the nozzle entrance, and the velocity to the left wall of the top hole is slightly higher than the experimental value. It may be because the experimental PIV measurement does not have sufficient resolution to resolve the large velocity gradient at the hole wall. The prediction on the L2 section is also closer to the experimental results, and the deviation of velocity between the maximum and the left wall is also small.

In the downstream of 1.5 mm from the nozzle inlet, the maximum velocity in the numerical simulation is also consistent with the experimental results, especially the velocity change trend in the low velocity area on the left side. In summary, the simulation accurately predicts the two-phase flow characteristics inside the nozzle hole.

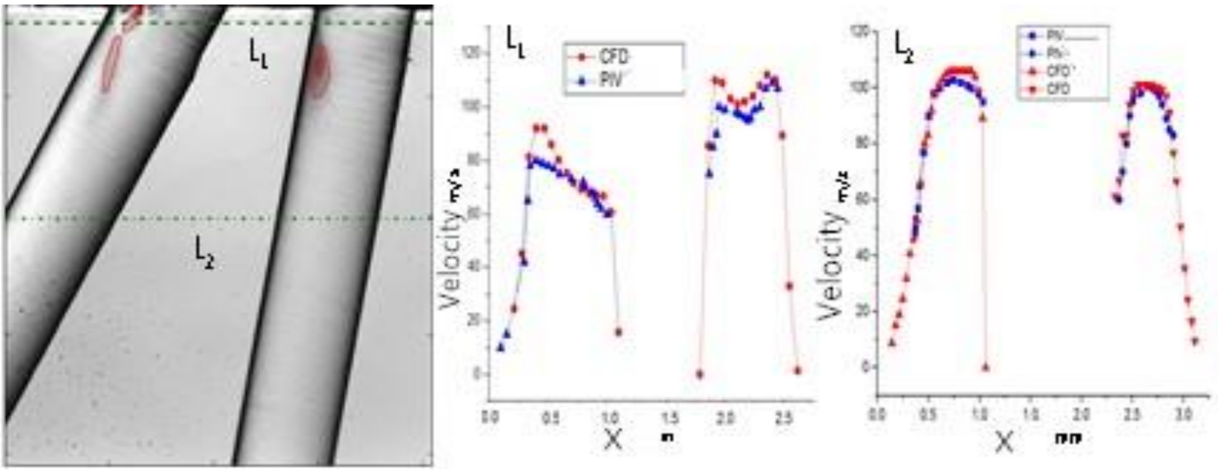

Fig. 10 Comparison of PIV Measurement and Simulation with the Cavitation Number of 1.3

5. Conclusion

Based on the classic nozzle internal cavitation experiment, the cavitation model and simulation method of the nozzle internal flow simulation was verified using parameters such as nozzle flow rate, nozzle velocity distribution and cavitation quantitative characteristics. On the basis of the verification model, the simulation and experimental comparison in the low-speed engine slide valve nozzle is further carried out. The main conclusions are as follows:

(1) For Winklhofer's experiment, the simulation results of SS, ZGB and Singhal cavitation models were compared, and it was found that the Singhal cavitation model had the highest agreement with the experimental results and could be used for numerical simulation.

(2) The simulation results of the nozzle flow rate, nozzle hole velocity, and cavitation characteristic size of the nozzles with different sizes are in good agreement with the experimental data, indicating that the simulation method effectively predicts nozzle cavitation.

(3) The simulation results of the internal flow of the slide valve two-hole nozzle show that with the increase of the cavitation number, the cavitation intensity continues to increase, and the thickness of the cavity layer gradually increases. The simulation results are consistent with the experimental results. The cavitation causes the asymmetric flow in the nozzle hole and the velocity on the cavitation side is reduced. The calculated velocity in the nozzle hole agrees well with the PIV measurement results. Therefore, this simulation method can be used to predict the internal flow of the slide valve nozzle of the marine low-speed engine.

References

[1] Fu Qingyan, Shen Yin, ZHANG Jian. Study on the Emission Inventory of Air Pollutants from Ships at Shanghai Port, Journal of Safety and Environment, vol. 12 (2012), 57-64.

[2] SCHMID A, HABCHI C, BOHBOT J, et al. Influence of in-nozzle flow on spray morphology. ILASS-Europe 2014. Bremen, Germany. 2014.

[3] HABCHI C, GILLET N, VELGHE A, et al. On the role of Cavitation in Marine Large Diesel injector: Numerical study of nozzle orifices eccentricity proceedings of ILASS - Europe 2014, 26 th Annual Conference on Liquid Atomization and the Spray Systems, Bremen, Germany F, 2014. 2014.

[4] HULT J, MATLOK S, MAYER S. Optical Diagnostics of Fuel Injection and Ignition in a Marine Two-Stroke Diesel Engine, SAE Int J Engines, vol. 7 (2014), 1-12.

[5] HULT J, SIMMANK P, MATLOK S, et al. Interior flow and near-nozzle spray development in a marine-engine diesel fuel injector, Exp Fluids, vol. 57 (2016), 1-19.

[6] WINKLHOFER E, KULL E, KELZ E, et al. Comprehensive hydraulic and flow field documentation in model throttle experiments under cavitation conditions [M]. ILASS-Europe 2001 Zurich 2001.

[7] Ma Bin, GAO Ying, LIU Yu et al. Influence of non-condensable gas and turbulence on cavitation in nozzle Hole, Journal of Agricultural Machinery, vol. 48 (2017), 342-348.

[8] SOU A, TOMIYAMA A, HOSOKAWA S, et al. Cavitation in a Two-Dimensional Nozzle and Liquid Jet Atomization, JSME International Journal Series B, vol. 49 (2007), 1253-1259.