ISSN: 2688-8645

Structure Design of Pressure Swirl Atomizing Nozzle

Shuxin Li, Chaoyang Chen, Yixiang Sun

Department of Power Engineering, North China Electric Power University, Baoding 071003, China

Abstract

Based on Solidworks and Fluent, the inner flow field of swirl nozzle is modeled and simulated. The simulation results of the internal flow field of the nozzle show that the change of the internal structural parameters of the nozzle has an importa nt impact on the flow characteristics of the working medium at the outlet of the nozzle. The parameters such as the axial velocity, tangential velocity and atomization angle at the outlet of the nozzle under different structural parameters are comprehensively compared, and finally the nozzle structure with the optimal atomization effect is determined.

Keywords

Swirl Nozzle; Simulation; Flow Field; Structure Parameter; Atomization Parameter.

1. Introduction

1.1. Research Background and Significance

Nozzle is a device that can break liquid into small droplets, which is widely used in energy, environmental construction[1], aerospace[2], chemical[3], industry, electronics[4], agriculture and forestry, automobile[5] and other fields, and has made great contributions to industrial production. It is of great significance to study the atomization characteristics of the pressure swirl nozzle in depth to improve the safety and economy of the process.

The nozzle can spray the atomized fluid into the external environment more evenly, and the atomized fluid is broken into liquid film under the action of surface tension and shear force generated between the environment and then atomized into droplets, which makes the contact area between the fluid and the environment rapidly increase, and is conducive to the follow-up heat exchange or mass exchange process. For the field of pesticide spraying, the spray area and spray height of the nozzle outlet play a decisive role in the spray effect; For the field of metal processing and cutting, because water cutting needs the nozzle to spray out high pressure jet, so the nozzle outlet flow rate and outlet pressure are the focus of attention of scholars; In the field of spray cooling, flow and heat transfer are closely related, and the flow field inside and outside the nozzle has become the direction of in-depth research.

The nozzle atomization effect is closely related to the nozzle structure and working conditions, therefore, it is necessary to explore the influence of different structural parameters and working conditions on the nozzle atomization effect.

1.2. Research Status at Home and Abroad

Research on atomizing nozzles has been active in the forefront of academic circles in various fields, and currently focuses on atomizing characteristics of nozzles

The research is mainly divided into two kinds: one is experimental research, the use of pump and other pressurizing equipment, the construction of atomization pipeline and atomization environment, through optical equipment, the flow process of the atomization outflow field is captured, and the use of image processing technology to carry out a comprehensive study of the nozzle flow process; The other is the use of modeling and simulation software to model the nozzle outflow field and numerical calculation,[6] so as to realize the nozzle internal visualization research, the nozzle internal flow and the nozzle external atomization law are analyzed and summarized.

In recent years, many scholars have used simulation software to simulate the internal and external field of the nozzle. For example, Rizk[7] et al. have summarized the laws of fuel atomization, gas nozzle combustion and other reactions through comprehensive experiments and simulation, and have also summarized the empirical formulas under specific conditions.

SOM and DATTA[8] focused on studying the influence of nozzle structure size on nozzle outlet flow characteristics and atomization characteristics, simulated the pressure swirl nozzle through the VOF model, and predicted the flow coefficient and spray cone Angle of the swirl nozzle.

Taylor[9] only considered the non-viscous flow and studied the flow field of the centrifugal nozzle, and obtained the calculation formula between the air cone radius, the atomization cone Angle and the flow coefficient. However, the nozzle is small and the internal flow is complex, so the assumption of invisco-free flow leads to a large error between the calculated results and the experimental results.

Chinn[10] deduces the calculation formula of invisco-free flow in the case of viscous flow, and obtains the formula that can be widely applied to nozzle flow.

Wang Guohui[11] et al. simulated the gas-liquid flow in a swirl nozzle based on Fluent, and found that there was an internal correlation between the pressure at the nozzle exit and the mass flow. The results show that the structural parameters of each part of the swirl nozzle will affect the flow rate of the working fluid at the nozzle outlet, but the atomization Angle only depends on the shape and number of holes.

Based on the SIMPLEC algorithm in Fluent, Zhou Zhanggen[12] et al. took the incompressible Reynold equation as the momentum equation to study the flow field characteristics of a shrinking nozzle with a given initial pressure and outlet diameter. The results show that the flow velocity increases rapidly at the nozzle contraction section and tends to be stable at the exit, forming a nuclear flow zone with equal velocity, and the pressure changes are similar to the flow velocity changes.

1.3. Research Content of this Paper

The research object of numerical simulation in this paper is the atomization effect of the pressure swirl nozzle. The atomization characteristics of the pressure swirl nozzle are obtained by changing the pressure difference between the inlet and outlet of the nozzle, the number of swirl holes of the pressure swirl nozzle, the inclination Angle of the swirl hole and the structural parameters of the nozzle exit aperture. The specific work is as follows:

(1) Analyze the flow law of the working fluid in the nozzle

(2) Conduct three-dimensional modeling and mesh division of the target pressure swirl atomizing nozzle, establish the nozzle atomization model, and build the relevant model and set the numerical solution in Fluent;

(3) Carried out grid independence verification to verify the reliability of the model

(4) By analyzing the changes in the velocity distribution of nozzle atomization flow field, atomization cone Angle, droplet diameter and distribution, the influence of the changes in the swirl chamber cone Angle, swirl hole Angle, swirl hole number and nozzle diameter structural parameters on the atomization flow field, atomization cone Angle, droplet average particle size and particle size distribution under different pressures was explored. And the optimal nozzle structure parameters under different working pressures were summarized.

(5) Comprehensively analyze the flow law of the internal and external fields, and conclude the relationship between the flow characteristics and atomization characteristics of the swirl nozzle.

2. The Main Content of this Paper

2.1. Numerical Simulation and Analysis of Pressure Atomization Swirl Nozzle Flow Field

2.1.1. Simulation Model

Solidworks was used to model the flow calculation domain in the nozzle, as shown in Figure 1.

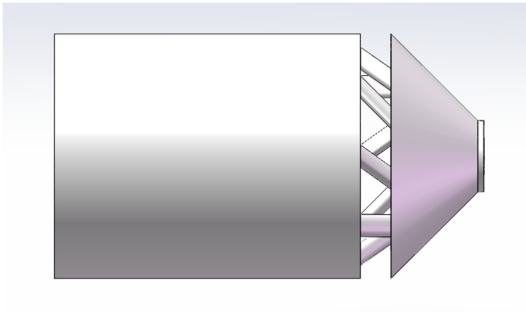

Figure 1. Nozzle flow calculation domain model

2.1.2. Mesh Division

In this paper, ANSYS ICEM software is used to divide the grid, which provides a variety of grid division methods. Because of the complex structure of the calculation domain model of the flow field in the nozzle, unstructured mesh (tetrahedral mesh) is used to divide the swirl hole and swirl chamber, and the mesh is encrypted. For the main drainage basin, due to the single structure and simple flow, structured grid (hexahedral grid) is used, and the grid division is shown in the figure2.

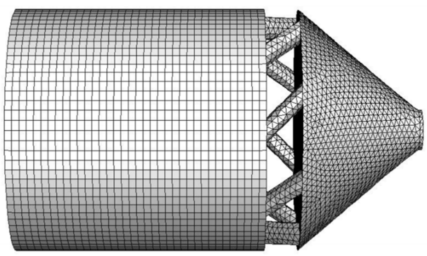

Figure 2. Flow calculation domain grid division diagram in the nozzle

2.1.3. Internal Condition Setting of Fluent

After importing Fluent, the unit is set to mm and the Type is Pressure-based. Gravity is ignored for the flow in the nozzle, so the gravity is set to 0. The turbulence model is set to Realizable k-ε model and the working medium is set to water-liquid. The inlet boundary condition is set as pressure-inlet, and the inlet pressure range is 0-1MPa; The outlet boundary condition is set as pressure-outlet, and the outlet pressure is atmospheric pressure, that is, the gauge pressure is 0; The wall is set as a non-slip wall.

2.1.4. Model Reliability Verification

In this paper, in order to verify the grid independence, the number of grids from 30326 to 300886 were drawn, and the inlet pressure was set to 0.5MPa, and the flow rate was used as the criterion for the grid independence verification. The final results are plotted in Figure 3. As can be seen from the figure, when the number of grids is greater than 110,000, the flow rate basically does not change with the increase of the number of grids, so the grid independence verification is established. The number of grids finally selected by the nozzle model in this paper is 110918.

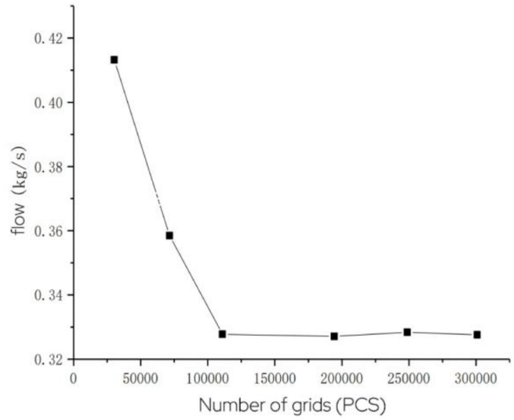

Figure 3. Grid independence verification

2.2. Influence of Different Structure Sizes on Nozzle Flow Characteristics

2.2.1. Influence of Inlet and Outlet Pressure Difference on Nozzle Flow Characteristics

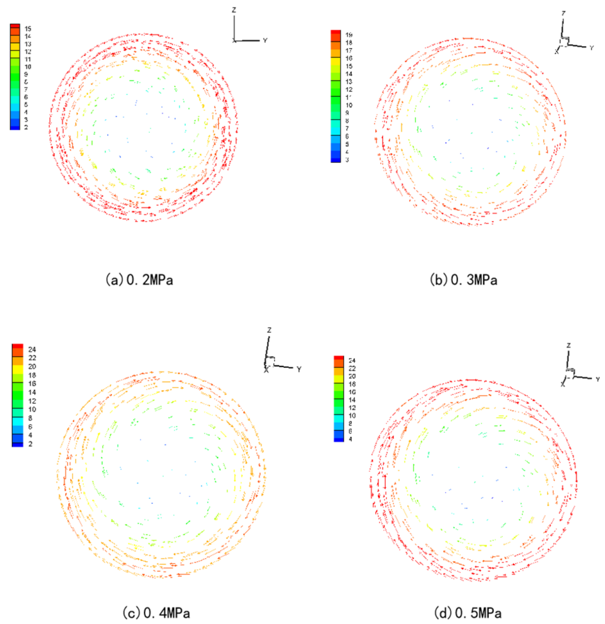

Figure 4. Velocity vector diagram of nozzle internal section under different inlet and outlet pressure differences (x=1mm)

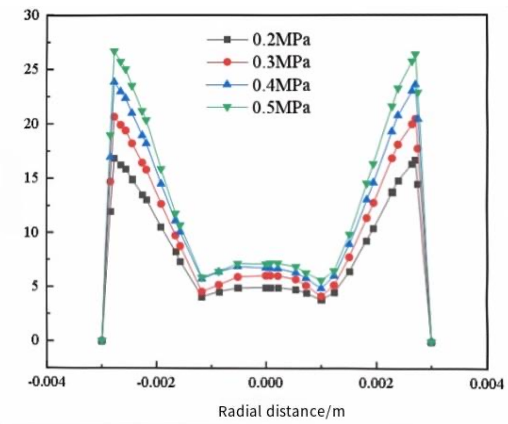

Figure 5. Radial relationship of outlet velocity under different inlet and outlet pressure differences

As shown in the figure5, the nozzle outlet still has two local peak points, one at the nozzle outlet liquid film, the other at the nozzle outlet center. As the pressure difference between inlet and outlet increases, the velocity at the nozzle outlet also increases.

2.2.2. Influence of Nozzle Swirl Hole Tilt Angle on Nozzle Atomization Characteristics

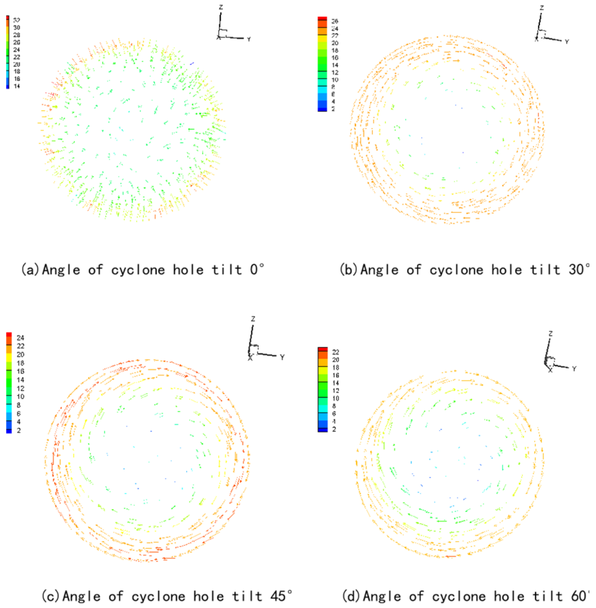

Figure 6. Velocity vector diagram of the inner section of the nozzle at different tilting angles(x=1mm)

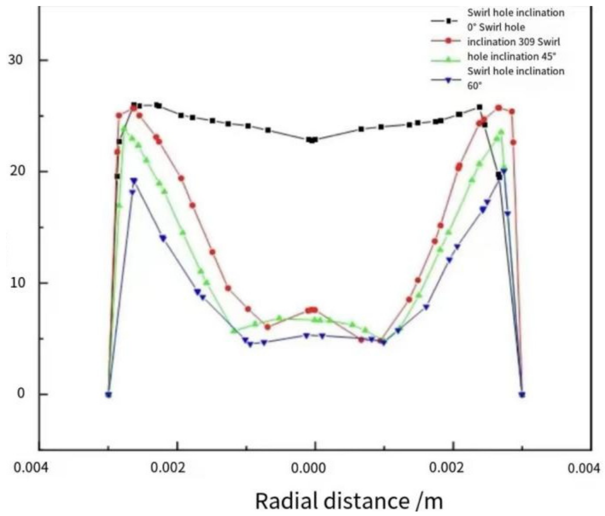

Figure 7. Relation of outlet velocity along radial direction at different swirling hole inclination angles

As can be seen from the figure7, when the swirl chamber Angle increases from 30° to 60°, the water jet velocity decreases from 25.7m/s to 19.3m/s, because the axial velocity in the flow space velocity is more transformed into the tangential velocity, which accounts for a larger proportion in the flow space velocity with the increase of the swirl hole Angle. In addition, the radial velocity distribution curve of the swirl hole with an inclination of 0° is different from the other three velocity distribution curves. This is because the water flow tilts at a certain Angle to maintain the tangential velocity without the swirl hole, so the degree of swirl flow in the nozzle swirl chamber is small.

2.2.3. Influence of the Number of Swirl Holes on Nozzle Atomization Characteristics

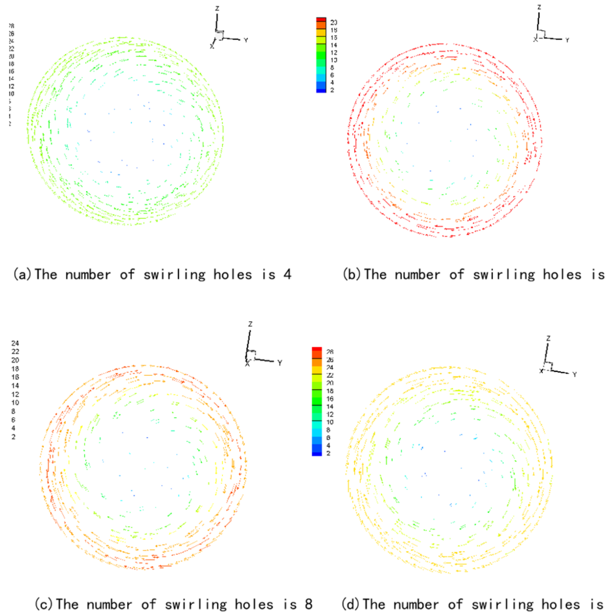

Figure 8. Velocity vector diagram of nozzle internal section under different number of swirl holes(x=1mm)

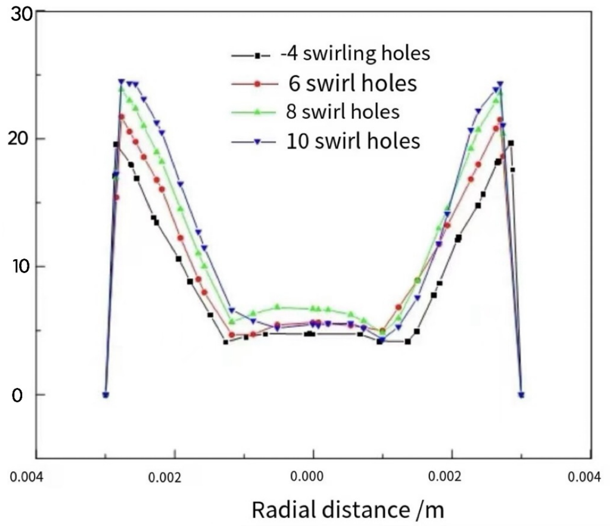

Figure 9. Relation of outlet velocity along radial direction with different number of swirl holes

As can be seen from the figure9, the more swirl holes, the faster the water jet velocity, which increases from 19.6m/s to 24.3m/s. When the number of swirl holes is 6 and 8, the water jet velocity is not much different, especially the velocity at the center of the nozzle outlet is approximately equal, indicating that the intensity of air being sucked into the nozzle is basically the same.

2.2.4. Influence of Nozzle Outlet Aperture on Nozzle Atomization Characteristics

As can be seen from the figure11, the outlet velocity distribution is generally consistent. When the inlet and outlet pressure difference is unchanged, the outlet water jet velocity of the nozzle with different outlet aperture changes little, mainly in the range of 22.9m/ S-24.2m /s. When the relative pressure difference in the nozzle swirl room is larger, the air sucked into the nozzle is stronger. For example, when the outlet aperture is 4mm, the center velocity of the nozzle outlet is larger.

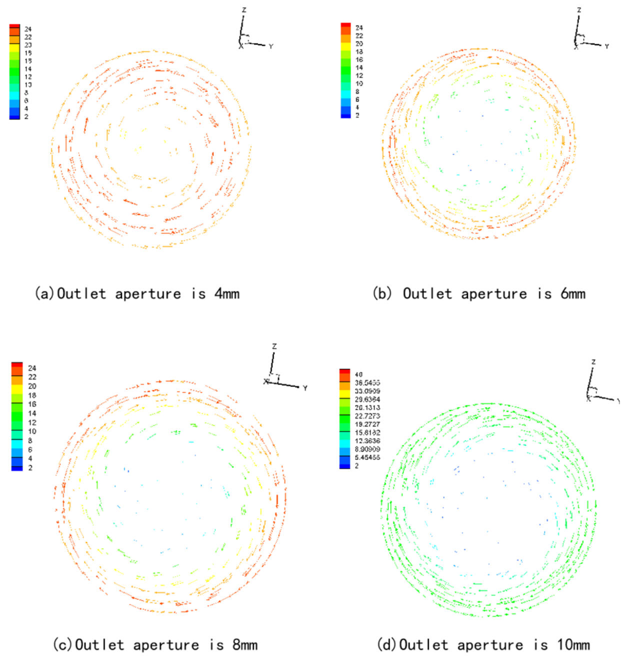

Figure 10. Velocity vector diagram of nozzle internal section with different nozzle exit apertures (x=1mm)

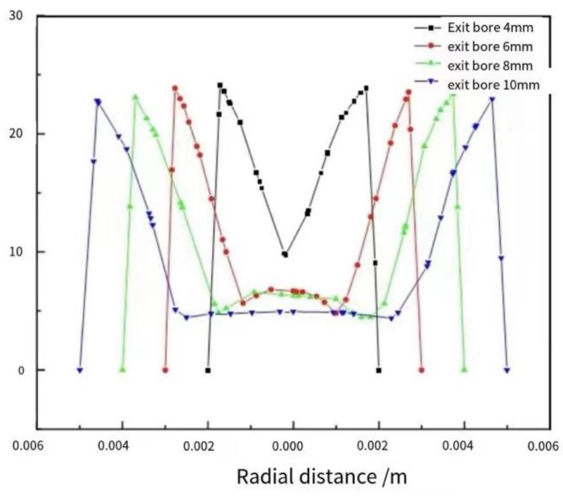

Figure 11. The radial relationship of outlet velocity with different nozzle outlet aperture

3. Conclusion

Based on the actual nozzle model, this paper uses Solidworks to build the nozzle flow field model. Then import it into ICEM to expand the mesh division and output it to Fluent to expand the numerical simulation. This paper focuses on analyzing the relationship between the internal structure size of the nozzle and the flow characteristics of the nozzle. The conclusions are as follows:

(1)The inlet and outlet pressure difference of the nozzle has a certain influence on the nozzle atomization effect. When the inlet and outlet pressure difference increases, the average atomization flow rate, water jet velocity and swirl strength at the outlet also increase, but the atomization cone Angle increases first and then tends to be stable at about 89.5°. After comprehensive consideration of each atomization characteristic parameter, it can be concluded that the pressure swirl nozzle of the reference structure has the best atomization effect under the inlet and outlet pressure difference of 0.4MPa.

(2)When the Angle of the swirl hole increases, the atomization cone Angle increases first and then decreases. When the Angle of the swirl hole is 45°, the atomization cone Angle is the largest, that is, the widest atomization range. The increase of the swirling hole Angle will lead to the increase of atomized particle size and particle size distribution range, and the increase will decrease with the increase of pressure. When the pressure is 0.3MPa and 0.4MPa, the swirling hole Angle is 45°, the atomized particle size is small and the particle size distribution is more concentrated.

(3)When the number of cyclone holes increases, the atomization cone Angle increases first and then decreases. When the number of cyclone holes is 8, the atomization cone Angle is the largest. The increase of the number of swirl holes has little effect on the nozzle exit velocity. The atomized particle size and the particle size distribution range gradually decrease with the increase of the number of swirl holes, and the reduction range gradually decreases with the increase of pressure.

(4)The change of nozzle outlet aperture has great influence on nozzle atomization effect. The average nozzle atomization flow rate, atomization cone Angle and swirl strength increase with the increase of nozzle outlet aperture, but the water jet velocity at the outlet is decreasing, which is due to the joint action of liquid film thickness, air core and atomization average flow rate increase, resulting in a decrease in the axial velocity of the outlet, and swirl strength and atomization cone Angle are jointly determined by the axial velocity and tangential velocity of the outlet.

(5)As described above, the structure of the swirl hole number 8, swirl hole tilt 45°, swirl chamber cone Angle 90°, nozzle diameter of 6mm nozzle in the inlet and outlet pressure difference of 0.4MPa atomization effect is good. The nozzle structure is the best.

References

[1] Xu Yanqiu. Discussion on technology of Water spray system for generator in high-rise Building [J]. Fujian Architecture,2009(10):155-156.

[2] Yang Lijun, Fu Qingfei, Tong Mingxi, Du Minglong, Wang Chen, Zhang Wei, Zhao Jianbo. Mechanical Problems in Propellant atomization process of liquid rocket Engine [A]. Fluid Mechanics Professional Committee of Chinese Mechanics Society. Abstract collection of the 7th National Fluid Mechanics Academic Conference [C]. Fluid Mechanics Committee of the Chinese Mechanics Society: Chinese Mechanics Society,2012:1.

[3] Lin Yuyang, Wu Hong, Wang Juanmei, Zhou Ning. Development of an Alloy Nozzle for Slurry Output in Coal Chemical Industry [J]. Mechanical Research and Application,2019,32(01):169-170.

[4] Ji Ailin, ZHONG Jianfeng, ShuAI Liguo.Heat Dissipation Method of Electronic Equipment with High Heat Flux [J]. Journal of Electronic Mechanical Engineering,2013,29(06):30-35.

[5] Sui Yongxu. Three-dimensional Flow Analysis in Cross-hole Nozzle of Diesel Engine [D]. Dalian University of Technology,2011.

[6] Ma X, Wang S, Li F, et al. Effect of air flow rate and temperature on the atomization characteristics of biodiesel in internal and external flow fields of the pressure swirl nozzle[J]. Energy, 2022, 253: 124112.

[7] N.K.Rizk, J.S.Chin. Comprehensive Fuel Nozzle Model[M]. AIAA, 94-3278.

[8] DATTA A, SOM S K.N umerical prediction of air core diameter, coefficient of discharge and spray cone angle of a swirl pressure nozzle [J]. International Journal of Heat and Fluid Flow, 2000,21 (4) : 412-419.

ISSN: 2688-8645

[9] Taylor G I. Seventh International Congress for Applied Mechanics, 1948[J]. Nature, 1950, 165(4190): 258-260.

[10] Yan K, Ning Z, Lu M, et al. Study on droplet size and velocity distributions of a pressure swirl atomizer based on the Maximum Entropy Formalism[J]. Entropy, 2015, 17(2): 580-593.

[11] Wang Guohui, CAI Timin, HE Guoqiang et al. Experimental and Numerical Study of a Swirl Nozzle [J]. Chinese Journal of Propulsion, 2003,24(1):28-32.

[12] Zhou Zhanggen, Ma Deyi. Numerical Simulation of High Pressure Nozzle Jet Flow Based on Fluent [J]. Machine Building & Automation, 2010(01):61-62.