Research on Nozzle Flow Field and Performance of Water Ramjet at Large Depth

Jiawen Wang¹, Rui Wang¹, Zhong Yao¹, Yanhu Sun² and Xiaobin Qi¹*,
¹Northwest Institute of Mechanical and Electrical Engineering, 712099, Xianyang, Shaanxi, China
²The Fourth Oil Production Plant of Changqing Oilfield, 718500, Yulin, Shaanxi, China
*Corresponding author: 15829531983@163.com

Abstract. For coprecipitating vehicle, water ramjet is an ideal way to reduce drag. In this paper, the performance of water ramjet at large depth and the flow field structure at nozzle are simulated and analyzed. Through the numerical simulation method of underwater Jet flow, this paper studies the development process of flow field at the initial stage and the parameters variation with time of Jet flow in steady state when the working depth is 200meters. The results show that at the beginning of the jet flow, the water blocking effect seriously prevents the development of the gas bubble, and the gas bubble turns over at the nozzle outlet, and expands in length and width. The unsteady process of ‘necking expansion necking’ occurs repeatedly at the nozzle outlet, with the pressure, velocity and temperature change dramatically near the nozzle outlet. When the flow field is fully developed, the pressure, density, flow rate and thrust of the nozzle outlet fluctuate repeatedly with time, and finally tend to a stable value after many oscillations.

Keywords: Water ramjet, large depth, jet flow, unsteady flow.

1. Introduction
The speed of underwater vehicle is related to its thrust and resistance. In recent years, supercaviting technology has become a research hotspot. Because the resistance generated by water is about 800 times higher than that produced by air, the resistance of underwater vehicle can be greatly reduced after supercaviting envelops the vehicle. As the tail end of underwater vehicle is also in the cavity, the water ramjet engine with jet propulsion is an ideal choice for these supercaviting vehicles. Water ramjet guides the water from outside into the combustion chamber of the engine. The water reacts with the metal carried in the propellant of the water ramjet to produce high temperature and high-pressure gas, which is ejected through the nozzle to generate thrust.

At present, the research on water ramjet at home and abroad mainly focuses on the following aspects, including the research on the flow field in the whole engine [1, 5], water injection mode [7], and water fuel ratio [2], composition and content of combustion metal [4, 6], and the test and calculation of engine thrust [3], mostly focus on small depth, but seldom involve in large depth.
However, at present, the submerge nce depth of the submarine has exceeded 200 meters, and the maximum depth has reached 600 meters. Therefore, it is necessary to carry out the research on the performance of water ramjet engine at large depth. When the working depth is more than 200 meters, the ambient pressure is as high as 2 ~ 3MPa, which will affect the performance of the nozzle and the thrust of the engine as well as the interaction between the gas jet and water. For the water ramjet, the nozzle plays an important role in accelerating the air flow and providing the thrust, so it is necessary to study the flow field around the nozzle of water ramjet at large depth.

2. Numerical method

2.1. VOF method

It is a gas-liquid two-phase flow problem that gas is ejected from nozzle to water. VOF method is a multi-phase flow interface tracking method under the fixed Euler grid. It is based on the physical foundation that two or more phases do not interpenetrate with each other. The governing equations are as follows:

(1) Volume fraction equation:

$$\frac{\partial \alpha_l}{\partial t} + \vec{v} \cdot \nabla \alpha_l = 0$$

VOF can track the interface between gas and liquid by solving the continuous equation of volume fraction of liquid phase. Without considering the phase transition, the liquid phase has no mass source term and the right end of the equation is zero. The subscript $l$ and $g$ represent the liquid and gas phases. As the base phase, the calculation of gas volume fraction is based on the following constraints:

$$\alpha_g + \alpha_l = 1$$

(2) Continuous equation:

$$\frac{\partial \rho}{\partial t} + \frac{\partial}{\partial x_j}(\rho u_j) = 0$$

(3) Momentum equation:

$$\frac{\partial}{\partial t}(\rho u_j) + \frac{\partial}{\partial x_j}(\rho u_j u_i) = \frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j} \left[ \mu \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) \right] + \frac{\partial}{\partial x_j} \left( -\frac{2}{3} \mu \nabla \cdot \vec{V} \right)$$

(4) Energy equation:

$$\frac{\partial}{\partial t}(\rho E) + \frac{\partial}{\partial x_j} \left[ u_j (\rho E + p) \right] = \frac{\partial}{\partial x_j} (k_{eff} \frac{\partial T}{\partial x_j})$$

Among them, the material properties $\phi$ are obtained by weighted average of volume fraction ($\phi$ can be $\rho, \mu$, etc.).

Among them, energy $E$ and temperature $T$ are the average mass variables of each phase, $\varphi = \frac{\alpha_l \rho \varphi_l + \alpha_g \rho \varphi_g}{\alpha_l \rho + \alpha_g \rho}$ ($\varphi$ can be $E, T$, etc), $k_{eff}$ is the effective thermal conductivity, $k_{eff} = (k + k_t)$, here, $k_t$ is turbulent heat conductivity, defined by the turbulence model used.
(5) state equation:

\[ \rho_q = \begin{cases} 
    \frac{p}{RT} & q = g \\
    \text{const} & q = l
\end{cases} \quad (6) \]

The gas phase is compressible and the liquid phase is incompressible. The calculation of the flow field is carried out simultaneously with the volume fraction equation. VOF solves the unified momentum equation and energy equation in the whole region, and the calculated velocity field is shared by each phase.

2.2. Turbulence equation

When the water ramjet works in deep water, the gas ejects from the nozzle outlet at high speed and shears with the water medium. There is an obvious turbulence phenomenon for the gas jet, and the large-scale backflow will appear in the gas bubble. The turbulent motion on the gas-liquid boundary is more severe. Therefore, it is necessary to simulate the turbulent phenomenon appropriately.

The most widely used method in engineering is to solve the Reynolds time averaged equation. The unsteady governing equation is averaged over time. The result is that the Reynolds stress term is produced, which makes the equation unable to be closed. The turbulence model came into being as a supplementary equation to express the Reynolds stress term and make the equation closed again. Based on the standard k-\( \varepsilon \) model and the wall function method, a mixed phase turbulence model was established.

\[
\frac{\partial}{\partial t} (\rho k) + \nabla \cdot (\rho k \mathbf{u}) = \frac{\partial}{\partial x_j} \left[ \left( \mu + \frac{\mu_s}{\sigma_k} \right) \frac{\partial k}{\partial x_j} \right] + G_k + G_b - \rho \varepsilon - Y_M + S_k \quad (7)
\]

\[
\frac{\partial}{\partial t} (\rho \varepsilon) + \nabla \cdot (\rho \varepsilon \mathbf{u}) = \frac{\partial}{\partial x_j} \left[ \left( \mu + \frac{\mu_s}{\sigma_\varepsilon} \right) \frac{\partial \varepsilon}{\partial x_j} \right] + C_\varepsilon \frac{k}{\varepsilon} (G_k + C_{\varepsilon_3}G_b) - C_{\varepsilon_2} \rho \frac{k}{\varepsilon} + S_\varepsilon \quad (8)
\]

\[
\mu_t = \rho C_{\mu} \frac{k^2}{\varepsilon} \quad (9)
\]

Here, \( G_k \) is the turbulent kinetic energy caused by average velocity gradient. \( G_b \) is the source term of turbulent kinetic energy caused by buoyancy. \( Y_M \) is contribution of pulsation expansion in compressible turbulence. \( \sigma_k \) and \( \sigma_\varepsilon \) are turbulent kinetic energy and dissipation rate respectively. \( S_k \) and \( S_\varepsilon \) are source term. \( C_{\mu t}, C_{\varepsilon_2} \) and \( C_{\varepsilon_3} \) are empirical constant.

2.3. Meshing and computing setup

The nozzle adopts typical Laval nozzle structure, and its shape is shown in Fig. 1. The combustion chamber diameter \( Di = 130mm \), throat section diameter \( dt = 60mm \), nozzle outlet section diameter \( do = 90mm \), nozzle total length \( L1 = 200mm \), contraction section \( L2 = 50mm \), expansion section \( L3 = 100mm \).
Due to the axial symmetry of the model, the whole flow field in the inner and outer regions of the nozzle is solved by using the semi model method. The calculation area and boundary conditions used in this paper are shown in Fig. 2.

Using structured grid, the grid size near the nozzle is basically the same, and the external field is divided into gradient grid. The global grid and the details near the nozzle are shown in Fig. 3 and Fig. 4.

The pressure and temperature of the boundary are specified by UDF. The back flow of pressure outlet is specified to be water medium with temperature of 300K. Because this paper focuses on the depth of 200 meters, so the operating pressure is 20atm. PISO and PRESTO method is for Pressure-Velocity Coupling and Pressure discretization respectively. The remaining parameters are default values.

The fluid medium includes incompressible liquid water and compressible gas. The VOF multiphase flow model in fluent software is used to calculate the interaction between gas mixture and water. The pressure and temperature curves varying with time are given at the inlet in Fig. 5 and Fig. 6, and the UDF function is embedded into fluent by compiling UDF function.
3. Study on initial stage of gas jet development process

The high temperature and high-pressure gas produced by the combustor accelerates into water environment through the Laval nozzle, and the medium is composed of water and gas mixture. Due to the high density of water outside the nozzle and the obstruction of the gas flow, a transient gas bubble is formed at the end of the nozzle, which is accompanied by the process of expansion, fracture, contraction and extinction.

3.1. Contours of density about gas bubble formation

Figure 7. Contours of density about gas bubble at different times

The development process of gas bubble is shown in Fig. 7. It can be seen that at the initial time of gas bubble formation, the bubble tends to turn up on both sides and sag in the middle at 1.5ms, which is caused by the great water pressure at large depth. With the increase of nozzle inlet pressure, the bubbles increase gradually, and the upward flow on both sides no longer only develops on the right side of the
nozzle section, it also extends to the left side of the nozzle section. As the gas is continuously ejected, the bubble expands in the length and width direction.

3.2. Establishment of flow in nozzle
The high temperature and high-pressure gas in the combustion chamber reaches the congestion state after the contraction and acceleration of the nozzle, and the shock wave forms in the expansion section of the nozzle and advances towards the outlet direction. When the gas crosses the normal shock wave, the movement of the gas will drop from supersonic to subsonic, and the pressure, temperature and density will change accordingly.

Figure 8. Contour of Mach number
Figure 9. Contour of pressure

The contour of Mach number and pressure in the jet flow field are given in Fig. 8 and Fig. 9. In the nozzle, with the increase of inlet pressure, the nozzle throat reaches the sonic velocity, and then the supersonic flow rapidly advances to the nozzle diffusion section, which shows that two oblique shock waves and one positive shock wave appear in the diffusion section continue to advance towards the nozzle, such as 1ms. Due to the inertia and static pressure of water, the air flow rushing to the water body is blocked, forming a reverse pressure wave. At the same time, the rapidly rising pressure in the combustion chamber will push the air flow out continuously and meet with the back propagation pressure wave, so that the pressure of the whole nozzle increases, and the supersonic flow becomes subsonic flow in the nozzle. The shock wave propagates from the throat to the outlet, which is pressed back into the throat and finally is pushed to the nozzle.

3.3. Development of jet temperature field

The temperature field of gas jet field at different time is given in Fig. 10. It can be seen that the distribution of temperature field is basically consistent with that of jet axis. Because the direction of high-temperature gas injection is basically along the axis, the temperature in the gas bubble is always higher in the axial direction, and the farther along the axis, the lower the temperature.
4. Analysis of steady shock structure and thrust characteristics of jet

4.1. Analysis of shock wave flow field of nozzle steady jet

Fig. 11 shows the gas volume fraction, Mach number, temperature and pressure distribution of the nozzle steady flow field. The flow at the nozzle inlet is subsonic. After accelerating in the contraction section, the velocity of sound is reached at the throat of the nozzle, and the supersonic flow is realized in the expansion section. The results show that the Mach number increases gradually from the nozzle inlet to the outlet, while the temperature and pressure decrease from the inlet to the outlet.

4.2. Performance analysis of nozzle

When the engine is working in deep water, the unsteady process of ‘necking expansion necking’ occurs repeatedly in the initial stage of the jet. With the drastic changes of pressure, velocity and temperature near the nozzle outlet, the jet maintains a dynamic equilibrium, the gas-liquid interface vibrates at the nozzle outlet, and the positive shock wave vibrates in the nozzle expansion section. These factors will affect the engine performance.
Fig. 12 shows the curve of the average pressure at the nozzle outlet under the working environment of 200m water depth with time. It can be seen that in the initial stage of the flow field establishment, the pressure at the nozzle inlet rises rapidly, and the pressure wave is hindered by water medium when it is transmitted to the nozzle outlet. Because the density of water is much higher than that of gas, it presents a solid wall property relative to the gas, and the pressure wave propagates backward. The back propagation pressure wave propagates forward under the action of increasing nozzle inlet pressure, which makes the nozzle outlet pressure present oscillation. With the growth of gas bubble and the establishment of supersonic flow in nozzle throat, the nozzle pressure decreases continuously and is lower than the ambient pressure. The first pressure pulsation peak appears at about 2ms, which corresponds to the first necking phenomenon of gas bubble. During necking, the gas jet is blocked and the pressure rises suddenly. Shock wave is formed in the expansion section of nozzle to adjust. The pressure pulse of nozzle rises and the pressure peak appears. Then the gas bubble expands, the supersonic jet is established rapidly and the pressure drops suddenly. From the diagram, each peak pressure corresponds to the necking phenomenon, and the degree of necking corresponds to the amplitude of pressure fluctuation. The maximum pressure rises to 20MPa, which is 10 MPa higher than the total pressure at the nozzle inlet. Then the amplitude of pressure fluctuation gradually decreased, and the degree of necking began to weaken.

Fig. 13 shows the time varying curve of gas jet velocity at nozzle outlet in 200m water depth. It can be seen that the outlet velocity of the engine nozzle increases linearly at the initial stage of operation. After entering the surge stage, the trend is inversely related to the pressure change. When the pressure pulsation reaches the peak value, the outlet velocity of the nozzle decreases to the lowest. The peak value of pressure fluctuation is larger, the lower the speed value.
Fig. 14 shows the time-varying curve of gas flow at nozzle outlet in 200m water depth. It can be seen that the flow rate will also be affected by these factors due to the tail gas bubble necking expansion cycle during engine operation. With the increase of ambient pressure, the gas flow decreases. The pressure oscillation occurred in the nozzle after several times of flow.

Fig. 15 shows the time-varying curve of engine thrust under the working environment of 200m water depth. It can be seen that at the beginning of the jet flow field, the engine thrust oscillates and rises due to the forward and backward propagation of the pressure wave in the nozzle. As the normal shock wave is pushed out of the nozzle outlet, the high-pressure water medium compresses the combustion bubble resulting in necking. When the gas jet enters the surge stage, and the gas bubble ‘necking expansion necking’ cycle occurs the force also fluctuates. This phenomenon appears repeatedly in the figure, and the final thrust is stable at 40kN.

5. Conclusions
It is a complex process to generate thrust by gas injection of water ramjet. The numerical simulation method of underwater gas jet is established and the initial and steady jet development process are studied in this paper. The working characteristics of ramjet and the flow field around the nozzle were simulated. The conclusions are as follows:

1. At the beginning of the jet, the water blocking effect seriously hindered the development of the gas. The gas bubble in the nozzle outlet develops accompanied with the phenomenon of rolling, and expanded in length and width. The unsteady process of ‘necking-expansion-necking’ occurs repeatedly at the nozzle outlet, and the pressure, velocity and temperature change dramatically near the nozzle outlet.

2. When the flow field is fully developed, the pressure, density and flow rate of the nozzle outlet fluctuate repeatedly with time, and finally tend to a stable value after many oscillations. This accords with the real motion state of the underwater engine.

References
[1] Mingyuan Liu. Numerical Simulation of Flow Field and Performance Research of Hybrid Propulsion System Based on Aluminum-water Combustion [D]. Harbin Engineering University, 2019. (In Chinese).
[2] Ge Dong. Simulation on Transient Ignition of High-metal Magnesium-based Fuel Water Ramjet [D]. National University of Defense Technology. 2016. (In Chinese).
[3] Xiaoxin Wang, Jianjun Dang, Xuelei Zhang et al. A Thrust Test Method for Water Ramjet Engine [J]. Torpedo Technology, 2015, 23 (2): 124 - 133. (In Chinese).
[4] Haitao Huang. A Study on High Energy of Mg-based Hydro-reactive Metal Fuel Propellant [D]. Beijing Institute of Technology. 2015. (In Chinese).
[5] Taijun Yu, Baoshou Wang. Two Phase Flow in Nozzle of Underwater Ramjet [J]. Journal of Sichuan Military Technology. 2012, 33 (3): 4-7. (In Chinese).
[6] Murugesan Ramakrishnan, Satyanarayanan R.Chakravarthy, Jayaraman Kandasamy, Ramanujam Sarathi. Development of Nano - Al Based Highly Metalized Fuel - Rich Propellant for Water Ramjet Propulsion Applications. DOI:10.1002/prep.201900384. Propellants, Explosives, Pyrotechnics. 2020 (45).
[7] Dechao Liu, Shulei Li, Gongnan Xie, Effect of Primary Water Injection Angle on Thermal Propulsion Performance of a Water Ramjet Engine, DOI:10.1016/j.ast.2019.105630, Aerospace Science and Technology, 2020 (97).