Numerical simulation for high speed oblique water entry of different density projectiles

Q Mu, T H Xiong, K J Wang, W J Yi and G Jun
National Key Laboratory of Transient Physics, Nanjing University of Science and Technology, Nanjing 210094, P.R. China
mujqing@163.com

Abstract. To explore the effects of material density on the cavitation flow field of a projectile entering water at an oblique angle at 300m/s, numerical simulations were conducted. The model was based on the finite volume method, VOF (Volume Of Fluid) multiphase model, Schnerr-Sauer cavity model, SST(Shear Stress Transfer) k-ω turbulence model and dynamic mesh method. The evolution laws of cavity shape, motion characteristics and hydrodynamic characteristics of water entry are analyzed. The research results show that the numerical calculation method can effectively simulate the change of cavity shape during the water entry of the projectile. The larger the density is, the smaller the diameter of the cavity formed after entering water is; the longer the cavity is, and the smaller the diameter of the cavity opening is; The peak value of pressure on the surface of projectile with different density is the same at the moment of entering water, but the projectile with larger density experiences a slower rate of the surface pressure decline; Moreover, the larger the density, the smaller the acceleration peak value of the projectile at the moment of entering the water, the slower the velocity decay after entering the water, and the deeper underwater at the same time.

1. Introduction
Any object entering water will experience the impact process with the water surface before entering the water, which is a complex physical process involving the interaction of solid, liquid and gas, which widely exists in the nature and engineering practice. Such as the landing of seaplane, the dropping of lifeboat at sea, the slamming of bow into water, the landing of cosmic devices on water, the water-entry of air-dropped torpedo, etc. The water entry movement of the projectile is the outcome of the inertial force system of the projectile itself and the force system of the surrounding fluid acting on the projectile. The vehicle will interact with the water during the high-speed water entry, and water moves around under the action of vehicle to form a cavity. Meanwhile, the velocity attenuation and trajectory deviation will occur to the vehicle under the reaction of water, such as the phenomenon of whip and bounce [1]. Under the condition of constant water temperature, the whole process of cavitation occurrence, development and collapse is called cavitation phenomenon. With the increasing
demand of marine equipment, it is of great engineering significance to study the cavitation phenomenon of vehicle entering water.

Lu Zhonglei [2] carried out the experimental research on the flow characteristics of the low-speed vertical water entry cavitation of the open cavity cylindrical shell structure based on the high-speed camera method, obtained the shape and dynamic characteristics of the water entry cavitation, summarized the relationship between the flow mode and the water entry speed, and analyzed the formation mechanism of the water entry cavitation wave motion and cloud flow. Chen Cheng [3] studied the law of the load characteristics of the supercavitating vehicle entering the water at an angle of 20° with time through design experiments, studied the influence of the speed of the vehicle entering the water and the area of the cavitating vehicle on the impact load, and analyzed the quantitative relationship between the impact load and the water entering parameters. Truscott [4] studied the small spheres with different surface coatings in 2009. They were coated with different hydrophilic and hydrophobic materials on the surface of the small spheres, and then the small spheres were rotated into the water around the center of the sphere at a certain angular speed. The cavitation formed when the coated small spheres entered the water and its flow field dynamics characteristics were obtained through experiments. The results showed that the surface coating of the small spheres closed the cavitation and its development Features have a great deal of relevance. Yves Marie [5] Based on Wagner theory developed a theoretical method for numerical solution of three-dimensional object into water, which was proved to be accurate and reliable by comparison with the experimental results. Savchenko [6] carried out the supercavitation test in the gravity water tunnel with the initial velocity of 50-150 m/s, and studied the cavitation shape of different shapes of vehicles and the resistance characteristics of cavitators. Jiang Yunhua [7] have studied the generation and development of supercavitation and the characteristics of cavitation flow in the low-speed inclined underwater of the disk cavitating vehicle, and summarized the variation law of the length of the cavitation.

The density of the projectile is an important factor that affects the cavity development and change. In this paper, the water entry process at the speed of 300 m/s with the density of 1 g/cm³, 2.7 g/cm³ and 5 g/cm³ is simulated by using the numerical calculation method. The development law of the cavitation shape, the motion characteristics and hydrodynamic characteristics of water entry are achieved. The research results can provide theoretical reference for engineering practice.

2. Numerical model and calculation method

2.1. Governing equations

The numerical calculation assumes that the fluid medium is incompressible and ignores the heat conduction effect caused by the fluid viscosity, that is, the energy equation is not solved. In addition, the effect of the detachment shock generated by the supersonic motion of the vehicle in the air on the water area and the attitude of the vehicle is not considered. The volume function of fluid (VOF) multiphase flow model regards multiphase fluid as a single fluid medium mixture. According to the solution of water, air and vapor three phase flow problems, the volume fraction of water, air and vapor is represented by $\alpha_1$, $\alpha_2$, and $\alpha_3$ respectively. In any area of the whole flow field, they can be expressed as,
The 2020 Spring International Conference on Defence Technology
Journal of Physics: Conference Series
1507 (2020) 102028
doi:10.1088/1742-6596/1507/10/102028

\[ \alpha_i + \alpha_g + \alpha_v = 1 \] \(^\text{MERGEFORMAT (1)}\)

The continuity equation of the mixture is as follows,

\[ \frac{\partial \alpha_i}{\partial t} + \frac{\partial}{\partial x_i}(\alpha_i u_i) = 0 \] \(^\text{MERGEFORMAT (2)}\)

\[ \frac{\partial}{\partial t}(\rho_m) + \frac{\partial}{\partial x_i}(\rho_m u_i) = 0 \] \(^\text{MERGEFORMAT (3)}\)

\[ \frac{\partial}{\partial t}(\rho_m u_i) + \frac{\partial}{\partial x_j}(\rho_m u_i u_j) = -\frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j}[(\mu_m + \mu_t)(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i})] \] \(^\text{MERGEFORMAT (4)}\)

Where \(u_i\) is the velocity component, \(p\) is the pressure, \(\mu_t\) is the turbulent dynamic viscosity, \(\mu_m\) is the dynamic viscosity of the mixed medium, \(\rho_m\) is the density of the mixed phase,

\[ \rho_m = \alpha_i \rho_i + \alpha_g \rho_g + \alpha_v \rho_v \] \(^\text{MERGEFORMAT (5)}\)

In this paper, the cavitation problem in flow is solved by using the Schnerr and Sauer cavitation model, which describes the transport equation of the volume fraction of vapor as follows,

\[ \frac{\partial}{\partial t}(\alpha_v u_v) + \nabla \cdot (\alpha_v \rho_v v_v) = R_v - \dot{R}_v \] \(^\text{MERGEFORMAT (6)}\)

\[ v_v \] is the vapor phase velocity, \(R_v\) and \(\dot{R}_v\) represent the mass transfer source terms connected to the growth and collapse of the vapor bubbles respectively. When \(p_v \geq p\),

\[ R_v = \frac{\rho_i \rho_v}{\rho_m} \alpha_v (1 - \alpha_v) \frac{3}{R_g} \left( \frac{2}{3} \frac{(\rho_i - p)}{\rho_i} \right) \] \(^\text{MERGEFORMAT (7)}\)

\[ R_g = 1 \times 10^{-6} \text{m} \] is the radius of the gas nucleus in the Rayleigh equation. When \(p_v \leq p\),

\[ R_v = \frac{\rho_i \rho_v}{\rho_m} \alpha_v (1 - \alpha_v) \frac{3}{R_g} \left( \frac{2}{3} \frac{(p - \rho_v)}{\rho_v} \right) \] \(^\text{MERGEFORMAT (8)}\)

In this study, we used the k-\(\omega\) SST turbulence model to solve the fluid control equation. This model is developed by Menter, which combines the advantages of the k-\(\varepsilon\) and k-\(\omega\) turbulence model, and adds the eddy viscosity limit equation to appropriately describe the transmission of turbulent shear stress. It has an remarkable advantage in predicting the near-wall flow and swirl[8].

2.2. Computational model mesh generation method
The physical model is a projectile with a cylindrical body and a disk head, the full-length \(L\) is 65 mm, the front cone length \(l\) is 15 mm, the largest diameter \(D\) is 10 mm, and the head disk diameter \(d\) is 3 mm. The symmetry surface shape of the projectile is shown as figure 1.
Using dynamic grid technology to solve unsteady problems often requires a lot of computing resources because of the computational analysis of the large volumes of data. Therefore, the computational domain is divided into the motion domain on the motion path of the projectile and the static domain which is less affected by the motion of the projectile for improving the calculation speed and precision. In order to improve the accuracy of the calculation result of air region, vapor-water interface and cavitation region, a more dense mesh is used in the motion domain and the mesh around the revolution body is further refined. The boundary layer mesh is defined near the wall of the revolution body. The mesh in the motion domain and the static domain slip through a set of mesh interfaces, and in fluent, which are matched through the mesh interface tool. The mesh partition of computational domain is shown in figure 2.

2.3. The computational domain and boundary conditions
The computational domain is a cylinder whose symmetry plane is shown in figure 3. According to the conclusion of numerical simulation in reference [9], the radial size of the basin should be 46 times larger than the maximum diameter of the projectile to avoid the wall effect. The computational domain diameter is 1000mm, and the height is 2300mm. The gas-water junction is located at 25mm below the origin of coordinates, where the air domain height is 500mm and the water domain height is 2000mm. In initial state, the angle between the axis of the revolution body and the X axis is $\alpha$, and the origin of the coordinate is placed at the middle point of the head. The pressure exit boundary condition (pressure outlet) is used in the outer basin boundary, meanwhile the user defined function (UDF) is used for defining pressure on the boundary. The computational environment pressure $P_0$ is set to 101
325 Pa.

Figure 3. The schematic diagram of the computational domain.

2.5. Numerical method
In this paper, the finite volume approach based on the VOF multiphase flow model is used to displace the fluid governing equations. The pressure velocity coupling is solved using the Pressure Implicit with Split Operators (PISO) algorithm. The pressure field and velocity field were discretized using PRESTO! and the second order upwind schemes, respectively, and the volume fractions of phases are discretized by using CICSAM. The user defined function (UDF) written in C is developed to define the mass of body, moment of inertia, and boundary pressure of computation domain, and finally to achieve the water entry motion of projectile.

3. Calculation results and analysis
3.1. The analysis of cavity shape and flow field
Figure 4 shows the liquid volume fraction contours of different density projectile during entering water, which can directly show the development process of cavitation. It can be seen from figure 4 that the development process of cavitation formed after entering water is almost similar. There is no significant difference of the cavitation shape generated by different density projectile within 1 ms after entering water, and the difference of cavitation size and shape appears gradually after 1 ms of water entry.
It can be seen from figure 4 that there is almost no difference in the cavitation shape formed by the water entry of projectile with different density at 0.5 ms of water entry, the water entry depth of projectiles are almost the same, and the uplift of liquid level are also extremely similar, indicating that the energy transmitted by the projectiles to the surrounding fluid are almost the same. The navigation resistance of the projectile with different density are almost the same at the same water entry depth. The larger the density is, the smaller the diameter of the cavity is, the smaller the opening diameter is at the free liquid level, the greater the degree of cavity closure is. There is little difference in the height of liquid level uplift, which indicates that there is little difference in the kinetic energy lost by the projectile. There are obvious differences in the cavity produced by different density projectile in the development stage of the cavity. The lifting part outside the free surface of the projectile with density of 1.0 g/cm$^3$ is in the vertical shape, and the cavity is still in the expansion stage, while the lifting part outside the free surface of the projectile with density of 2.7 g/cm$^3$ and 5.0 g/cm$^3$ collapses into the cavity, and the cavity are in the closure stage.

3.2. Influence of projectile density on motion characteristics

Figure 5 shows the displacement curve of a projectile with different density within 2 ms after water entry. In the figure, $l_x$ and $l_y$ are the displacements of the projectile in the horizontal and vertical directions respectively, and D is the maximum diameter of the projectile. It can be seen from figure 5 that the displacement of each projectile is the same from launch to contact with the water surface due to the small air resistance. After contact with the water surface, the water entry movement of the projectile with the density of 1 g/cm$^3$ is significantly shallower than that of the other two projectiles with different density at the same time point. From 0.5 ms on, the slope of the projectileome displacement curve with the density of 5 g/cm$^3$ is gradually greater than that of the projectile with the density of 2.7 g/cm$^3$. On the whole, the larger the density of the projectile is, the deeper the projectile will navigate in the same time, and the longer the generated cavity will be.
Figure 5. Displacement variation curves.

Figure 6 shows the velocity curves of different density projectiles in the process of water entry. It can be seen from figure 6 that before entering the free water surface, the impact of air resistance on the projectile is quite small so that the impact on the speed can be ignored, and the projectile moves at a uniform speed. After impacting the water surface, the speed of the projectile starts to decrease under the action of the great reverse impact force, and the reduction amplitude decreases continuously. The projectile with higher density, the smaller the velocity decay is, and the little the velocity decay is. That is because the volume and shape structure of the projectiles with different density are the same, the wetted area of the projectile is the plane at the front end of the frustum at the moment of entering the water, and the impact force on the surface is the same at the same water entry speed. The smaller the density is, the smaller the mass of the projectile is, the greater the reverse acceleration is, and the faster the speed decays after entering the water.

Figure 6. Velocity attenuation curves.
3.2. Influence of projectile density on hydrodynamic characteristics

Figure 7 shows the acceleration curves of different density projectile after entering water. It can be seen from figure 7 that the acceleration is 0 before the projectile enters the water, the acceleration reaches the peak value at the moment of reaching the water surface, and the smaller the density is, the greater the acceleration peak value is. Then the acceleration begins to decay rapidly, and gradually tends to be stable, and the faster the acceleration decay is for the projectile with higher density. The acceleration curves of different density projectile gradually coincide after 1 ms of the launch, then become stable at 2 ms. The smaller the density is, the larger the acceleration stability value is, but the difference is very small, which can even be ignored.

![Acceleration variation curves.](image)

Figure 8 shows the curves of the pressure on the projectiles with different density when they enter the water. It can be seen from figure 8 that in the early stage of water entry, the surface pressure of the projectile is 0 because of the small air resistance. The surface pressure of the projectiles with different density reach the peak value at the moment of contact with the water surface. The peak value is exactly the same, which indicates that the density has no effect on the surface impact pressure at the moment of water entry. The surface pressure of the projectile decreases rapidly within 0.2 ms after the projectile enters the water surface. In the following 1.8 ms, the attenuation velocity of pressure on the surface of projectiles with different density decreases gradually, and the pressure on the surface decreases continuously and then tends to be stable. After reaching the pressure peak, the slower the attenuation velocity of pressure on the surface of projectile with higher density, and the corresponding pressure value is bigger when the pressure tends to be stable. On the whole, the impact pressure of different density projectile is quite great. The experimental model in this paper will be impacted by more than 300 times of atmospheric pressure when entering the water, and the impact pressure of projectile maintains at a high level in the following voyage. Therefore, it is necessary to design the surface structure of projectile reasonably so that it can bear the super impact pressure.
4. Conclusion

In this paper, a numerical simulation study is carried out on the oblique entry of the panhead projectile with a density of 1 g/cm³, 2.7 g/cm³ and 5 g/cm³ at an initial speed of 300 m/s, and the cavitation shape, hydrodynamic characteristics and motion characteristics of the projectile are analyzed. The main conclusions are as follows:

1. The larger the density of the projectile, the smaller the diameter of the cavity profile, the longer the cavity profile, and the smaller the diameter of the cavity opening.

2. The greater the density of the projectile is, the greater the water depth of the projectile is in the same time, the longer the cavity is. The smaller the velocity attenuation of the projectile with the greater density is, the more gentle the velocity attenuation is.

3. In the initial stage of water entry, the projectile is affected by great impact load, which can reach hundreds times of atmospheric pressure. The peak value of the surface impact pressure of the projectiles with different density are the same during the oblique water entry. But the higher the density is, the smaller the acceleration peak value of the projectile is at the moment of entering the water, the slower the pressure decline is.

Acknowledgments

This work was supported by the Postgraduate Research & Practice Innovation Program of Jiangsu Province (Grant No. KYCX19 0338), the China Postdoctoral Science Foundation (Grant No. 2019M651838) and National Key Laboratory of Transition Physics Foundation (Grant No. 61426040402).

References

[1] YI W J, XIONG T H 2009 Research on drag characteristics of natural supercavitation profile for high speed bodies Ship Science and Technology 232 39-43

[2] LU Z L, WEI Y J, WANG C 2017 Experimental and numerical investigation on the flow structure and instability of water-entry cavity by a semi-closed cylinder Acta Physica Sinica 66 150-163
[3] CHEN C, YUAN X L, DANG J J 2018 Experimental Investigation into Impact Load during Oblique Water-entry of a Supercavitating Vehicle at 20° Acta Armamentarii 39 1159-1164
[4] TRUSCOTT T T, TECHET A H 2009 Water entry of spinning spheres Journal of Fluid Mechanics 625 135-165
[5] SCOLAN Y M 2015 Oblique water entry of a three dimensional body International Journal of Naval Architecture and Ocean Engineering 6 1197-1208.
[6] SAVCHENKO Y N 2001 Control of supercavitation Flow and stability of supercavitating motion of bodies RTO AVT Lecture Series on "Supercavitating Flows" (Belgium: RTO EN-010)
[7] JIANG Y H, XU S L, ZHOU J 2017 Water entry experiment of a cylindrical vehicle with disc cavitator Engineering Mechanics 34 246-251
[8] Menter F R, Kuntz M, Langtry R 2003 Ten years of industrial experience with the SST turbulence model Turbulence Heat & Mass Transfer 4 625 – 632
[9] HUANG C, LUO K, DANG J J 2015 Influence of flow field's radial dimension on natural supercavity Journal of Northwestern Polytechnical University 33 936-941