Ballistic Characteristics of High-Speed Projectiles Entering Water Vertically

B. Hao1,2, Y. G. Lu1† and H. Dai1

1 School of Mechanical Engineering and Automation, Northeastern University, Shenyang110819, China
2 School of Control Engineering, Northeastern University at Qinhuangdao, Shenyang Qinhuangdao 066004, China

†Corresponding Author Email: luyangen97@163.com

ABSTRACT

The formation of supercavitation after a high-speed projectile enters water has a decisive impact on the underwater ballistic and penetration of the projectile. In this study, Ansysfluent19.0 simulation software is used to study water entry of a chosen projectile at velocities of 300, 400, 500, and 600 m/s. The underwater cavitation characteristics, trajectories, and flow-field characteristics are analyzed for a 5.8-mm caliber conical flat head projectile, as well as for two other projectiles of the same caliber and different head shapes — conical cone head and elliptical flat head — entering water vertically at the same velocities. The attenuation rate of water entry velocity increases with the increase of velocity. Within first 3ms, the velocity attenuation rate of the conical flat-head projectile with a water entry velocity of 600m/s is 55.6 %, while the velocity attenuation rate of the projectile with a water entry velocity of 300m/s is only 16.3 % within 3ms. Among the head shapes considered, the conical flat head projectile is the most stable for vertical water entry. The stability of an elliptical flat head projectile is worse, and the trajectory stability of a conical cone head projectile is still worse.

1. INTRODUCTION

The cavitation phenomenon of an object entering water is an interesting topic worthy of study. In the 20th century, researchers worked on related issues. Miloh (1981, 1991a, 1991b) experimentally studied the load and flow distribution of inclined and vertical water entry of a sphere, and proposed a number order expression of the drag coefficient and the slamming load of at the moment the ball enters the water. Lee et al. (1997) studied the water entry characteristics of a sphere at different speeds, ignoring the viscosity of the fluid. When the velocity is small, the cavity closure will occur after the surface closure. In recent years, cavitation phenomenon and ballistic stability of a projectile entering water have attracted the interest of many researchers and have been explored through numerical simulation. Based on the Navier-Stokes (N-S) equation and the fluid volume multiphase flow (VOF) model, Wang (2019) carried out a numerical simulation for a spherical projectile, and analyzed the influence of surface roughness of the projectile on its hydrodynamic characteristics. Xiao et al. (2019) used six-degrees-of-freedom (6DOF) and overlapping networks to find the properties of projectiles entering water at small angles and analyzed the angle-dependence of the ballistic characteristics of such projectiles. Their results show that in the early stage of a projectile contacts with the water, the trajectory is approximately a straight line. But after a period of time, the trajectory of the projectile is strongly deflected by the flow force generated by the contact between the lower surface of the projectile and the water. Huang (2018) used dynamic network technology combined with the 6DOF control equation to carry out numerical simulation studies of a projectile of 12.7-mm caliber. Smirnov et al. (2020, 2022) presented a new algorithm and studied the problem of hypervelocity impact of projectiles with different configurations and different materials. These results showed that the stability of the underwater projectile is related to its angle of entry, initial velocity, and rotational speed. Therefore, it is important to learn about the rules governing the underwater behavior of high-speed projectiles at different initial speeds to further improve understanding of their underwater ballistic characteristics.

Some progress has been made in the study of projectile water entry characteristics based on factors such as velocity and head shape. Guo et al. (2012) analyzed the resistance coefficient of projectiles with different velocities and shapes, and found that projectiles
of different velocities had basically the same cavity size before a deep pinch phenomenon occurred. Guo et al. (2020) studied the cavitation characteristics and velocity attenuation law of a series of projectiles with different velocities in a closed container, and found a basic law relating velocity attenuation and container size. Using Fluent fluid calculation software as a platform, Meng et al. (2019) found the relevant characteristics of projectiles entering the water vertically, and studied the rules governing cavitation closure of a projectile which velocity is subsonic velocity. They found that when the velocity of the projectile approached the speed of sound in the water, a bow shock wave was generated when the projectile head entered the water. Chen et al. (2021), through experiments on projectiles of different shapes entering the water at different speeds, found that increasing the velocity can accelerate the development of cavitation and improve the stability of projectiles motion. Fan (2022) numerically studied the resistance characteristics and flow characteristics of projectiles with free and constant velocity entering water. It was found that the resistance characteristics are mainly related to the head shape and the velocity of the projectile. However, these studies were not comprehensive enough to reveal the ballistic characteristics of high-speed projectiles entering water vertically. In order to further study the influence of water-entry velocity on the underwater stability of ballistics, this study comprehensively analyzes the cavitation shape, and resistance coefficient of projectiles with different water-entry velocities through numerical simulation. The influence pattern of projectile head shape and velocity on projectile cavitation and ballistic stability is found, and provides a reference for studying the stable behavior of projectiles with higher penetration ability.

At present, research on supercavitating weapons is active, and many researchers have studied the problem of water entry. However, study of the cavitation and ballistic characteristics of underwater projectiles is not yet thorough enough. In order to better understand the effects of velocity and underwater pressure on projectiles, this study simulates vertical water entry with initial velocities of 300, 400, 500, and 600 m/s. In this study, VOF and turbulence models are adopted on the Reynolds mean-time N-S equation. The Schnerr-Saer cavity model combines the 6DOF dynamic network technology and analyzes the
effect of inlet velocity on cavity, trajectory, and flow field characteristics. The results of this study can promote understanding of the way that cavitation influences the steadiness of the underwater trajectory of the projectile. This analysis shows that the best shape of the projectile is a conical flat head. Finally, the ballistic stability of different projectiles with conical flat head, conical cone head, and elliptical flat head is studied.

2. NUMERICAL CALCULATION MODEL

2.1 Basic Control Equation

2.1.1 VOF Model

In the numerical simulation carried out in this paper, the VOF model is used to describe the multiphase flow formed by air, water, and water vapor generated by friction between the projectile and water when a projectile enters the water. Volume fractions are introduced to describe the proportion of different phases in the fluid medium — \( \beta_w \), \( \beta_v \), and \( \beta_a \) are the volume fractions of water, water vapor, and air, respectively, and satisfy Eq. (1) in all computational domains of the flow field:

\[
\beta_w + \beta_v + \beta_a = 1  \tag{1}
\]

Eq (2) is the continuity equation of the fluid mixture:

\[
\frac{\partial}{\partial t} (\rho_m u_i) + \frac{\partial}{\partial x_j} (\rho_m u_j u_i) = 0
\]

In this equation, \( i = 1, 2, 3; u_i \) is the component of the fluid velocity in the \( i \) direction; and \( x_j \) is the corresponding coordinate.

The momentum conservation equation of fluid mixing is Eq. (3):

\[
\frac{\partial}{\partial t} (\rho_m u_i) + \frac{\partial}{\partial x_j} (\rho_m u_j u_i) = -\frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j} \left[(\mu_m + \mu_t) \left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i}\right)\right]
\]

where \( \mu_m \) is the dynamic viscosity, given by \( \mu_m = \beta_w \mu_w + \beta_v \mu_v + \beta_a \mu_a \); \( \mu_t \) is the turbulent viscosity coefficient; \( x \) is the distance in the \( X \) direction; and \( \rho_m \) is the density of the fluid.

2.1.2 SST k-\( \omega \) Turbulent Flow Model

The SST k-\( \omega \) turbulent flow model can solve the flow separation problem and achieve good accuracy of the cavitation characteristics when a high-speed projectile enters water. Its mathematical expression may be written

\[
\frac{\partial (\rho k)}{\partial t} + \frac{\partial (\rho u_i k)}{\partial x_i} = \frac{\partial}{\partial x_j} \left[(\mu + \sigma_m \mu_t) \frac{\partial k}{\partial x_j}\right] - \beta \rho ok + P_{ab}
\]

where \( k \) is the turbulent intensity of the turbulent fluid, \( \omega \) is turbulence frequency , \( P_{ab} \) and \( P_{ab} \) are the turbulent kinetic energy terms due to buoyancy, \( P \) is the kinetic energy term due to the velocity gradient, \( \mu \) is the dynamic viscosity coefficient, and \( \beta \), \( \sigma_v \), \( \sigma_k \) are constants.

The specific expression for the mixing function \( F_b \) is

\[
F_b = \tanh(\arg^4)
\]  \tag{6}

\[
\arg_i = \min[\max(\frac{k_{SST}}{\beta \alpha y}, 500\nu), 4\rho \sigma_{\alpha_1} k_{\SST}] \tag{7}
\]

where \( \beta_{\alpha_1} \) is the kinetic energy of the turbulent fluid, \( \omega \) is turbulence frequency , \( P_{ab} \) and \( P_{ab} \) are the turbulent kinetic energy terms due to buoyancy, \( P \) is the kinetic energy term due to the velocity gradient, and \( \sigma_v \), \( \sigma_k \) are constants.

In Eq. (7), \( v \) is the viscosity coefficient of the fluid, \( v = \frac{\mu}{\rho} \); \( CD_{\alpha_1} \) is a cross-diffusion term; and \( Y \) is the minimum distance between the node and the wall. An expression for \( \mu \) is

\[
\mu = \frac{\rho_{\alpha_1} k_{\SST}}{\max(a_0, SF_2)} \tag{9}
\]

where \( S \) is a fixed measure of shear strain rate, \( S = \sqrt{2W_y} \), and \( F_2 = \tanh(\arg^2) \) and where

\[
W_y = \frac{1}{2} \left(\frac{\partial u_i}{\partial x_j} - \frac{\partial u_j}{\partial x_i}\right) \tag{10}
\]

\[
\arg_i = \max(2\rho \sigma_{\alpha_1} k_{\SST} \frac{1}{\omega}, 500\nu) \tag{11}
\]

2.1.3 Cavitation Model

In this study, the Schnerr-Sauer cavitation model (Schnerr 2001) is used to analyze the cavitation problem. The evaporation rate \( R_e \) and the condensation rate \( R_c \) of the model are expressed as

\[
R_e = \frac{\rho_e \rho_s}{\rho_m} \alpha(1-\alpha) \frac{2}{3} \frac{p_c - p}{\rho_v} \tag{12}
\]

\[
R_c = \frac{\rho_e \rho_s}{\rho_m} \alpha(1-\alpha) \frac{2}{3} \frac{p - p_c}{\rho_v} \tag{13}
\]

where \( R_b \), the bubble radius, is given by

\[
R_b = \left(\frac{\alpha}{1-\alpha} \frac{3}{4\pi N}\right)^{1/3} \tag{14}
\]

and where \( \alpha \) is the volume fraction of steam and \( N \) is the number of cavitations per unit volume. The governing
equation is
\[
\frac{\partial \varphi}{\partial t} + \frac{\partial}{\partial x}(\varphi u_x) = F_{\text{vis}} \frac{2 \alpha_{\text{vis}}(1 - \varphi)}{R_v} \left( \frac{2 P_v - P}{\rho_w} \sqrt{\frac{2}{3}} \right)
\]  \hspace{1cm} \text{(15)}

In this equation, \( F_{\text{vis}} = 50 \) and \( F_{\text{con}} = 0.001 \) are empirical constants taken from published work (Meng et al. 2019); \( P_v \) is the pressure of the water vapor; \( \rho_v, \rho_w \) are, respectively, the densities of water vapor and water; \( \alpha_{\text{vis}} = 5 \times 10^{-4} \) is the volume fraction of non-coagulate gas; and \( R_v = 1 \times 10^{-6} \text{ m} \) is the gas nucleus radial reference from the Rayleigh formula.

2.1.4 Six-Degrees-of-Freedom Model and UDF

UDF (user-defined functions) is a secondary development program based on C language that can be dynamically loaded into Fluent19.0. There are two loading methods for UDF: interpreted and compiled. Interpretation does not need to compile. It can be directly loaded and is very simple to use, although its calculations are slow. It needs to be compiled using the compiled software before it can be used. Its calculation speed is then fast, making the running time short. Considering the number of grids and the complexity of programming, this study uses compiled loading for UDF. At the same time, the fluid force and moment of the underwater projectile need to be output, which adds to the analytic efficiency and greatly improves the calculated results. The 6DOF solver of FLUENT software is used in all cases to solve for the motion of a underwater projectile. According to the force balance of the projectile, the displacement, velocity, and other variables of the projectile are calculated, to determine the position of the projectile’s center of mass and the direction of its motion. The center of mass of the projectile and the direction of motion obtained in calculation step \( n \) can be used to calculate these quantities in calculation step \( n + 1 \). The equations are
\[
x_{c,g}^{n+1} = x_{c,g}^{n} + v_{c,g}^{n} [t]
\]
\[
\theta_{c,g}^{n+1} = \theta_{c,g}^{n} + \omega_{c,g}^{n} [t]
\]
where \( v_{c,g} \) (given below in Eq. (18)) is the linear velocity of the projectile, \( \omega_{c,g} \) is its angular velocity, \( \theta_{c,g}^{n} \) is calculation step \( n \) in the projectile’s direction of motion, \( x_{c,g}^{n} \) is the position of the projectile’s center of mass at the \( n \)th calculation step, and \( G \) is the transform matrix.
\[
v_{c,g} = \frac{v_0}{1 + \alpha_v v_g t}
\]  \hspace{1cm} \text{(18)}

In the formulation, \( v_0 \) is the initial velocity; \( \alpha_v \) is the coefficient of velocity attenuation, and \( \alpha_v = \rho_v A_c C_v / 2 m_p \), where \( m_p \) is the mass of the projectile.

2.2 Computation model and Domain

In this study, a flat headed conical projectile made of tungsten alloy with a diameter of 5.8 mm was used as a model. In order to study the characteristics of the projectile entering water at different velocities, the study got better accuracy by calculating less. In this study, a three-dimensional model is used to numerically calculate four identical projectiles entering vertically, with different entry velocities (Hao, 2022).

The head of the main underwater projectile used in this study is a 16.5-mm truncated cone, the projectile head diameter is 2.6 mm, the head shape is conical flat-head. Its overall length is 30 mm. The conical flat-head projectile enters the water vertically at 300, 400, 500 and 600 m/s respectively. The cavity development, ballistic shape, and hydrodynamics of the projectile at different speeds were studied. In this study, SolidWorks was used for geometric modeling, followed by meshing of the model using ICEM CFD 19.0. The resulting grid was then imported into Fluent 19.0 for calculation, and the calculation results were analyzed using CFD-post. Finally, the analysis data was imported into Origin for plotting.

Figure 1 is the global sketch of the grid and the local sketch of the projectile surface. Figure 2 shows three different numbers of grids 1.3 \times 10^6, 2 \times 10^6 and 3 \times 10^6. The velocity of the projectile is 400m/s. It can be seen that the velocity of the 1.3 \times 10^6 grid attenuates faster, and the attenuation speed of 2 \times 10^6 and 3 \times 10^6 is not much different. This paper chooses the number of 2 \times 10^6 grids for research. The y+ values is approximately 1.(see,Fig3)
Fig. 3 Y plus diagram of projectile

Fig. 4 Computational domain

Fig. 5 Time steps independency

Fig. 6 Speed vs. time to validate the calculational method

Fig. 4 shows the computational domain of the numerical simulation method, in which the \( z \) value is set equal to zero; the cylindrical computational domain is a square with side length of 3400mm; the vertical range of the air part is 925 mm and the liquid depth is 2475 mm. The coordinate origin of the projectile is shown in the figure. The initial distance between the front end of the projectile and the water surface is 25 mm. Gravity acts in the \( x \) direction. The surface of the projectile is the wall condition, and the edge of the computational domain is the computational boundary. A hexahedron structure network is used for grid setting and refinement to ensure the accuracy of the model.

2.3 Numerical Methods and Verification

In this study, using the finite volume method, discrete equations are established for following the fluid motion in time and space. The PRESTO scheme is used for the spatial dispersion of the pressure field, and the gradient solution of the equation is determined by the least-squares method. The dissipation, turbulence, and momentum equations in the model equations are all second-order upwind forms. Dynamic mesh technology is used to update the mesh, and the process of high-speed projectiles entering water vertically is simulated numerically. The time step \( \Delta t \) is \( 1 \times 10^{-5} \) s, and the number of max iterations is 20 (Wang 2016)

\[
\Delta t = \frac{\text{Courant} \times \Delta x}{v_{x,y}}
\]  \hspace{1cm} (19)

Where \( \Delta x \) is overall length of projectile, the value of \( \text{Courant} \) is an empirical constant that typically falls within the range of 0.2 to 0.01 in order to maintain the accuracy of the calculation. The velocity attenuation remains consistent when the time step is either \( 1 \times 10^{-5} \) or \( 1 \times 10^{-6} \). The velocity attenuation is more rapid with a time step of \( 2 \times 10^{-5} \) s., as illustrated in Fig 5.

In order to check the correctness of the research method, the vertical water entry of a flat-nosed projectile model in the work of Guo (Guo 2012) is simulated numerically. The overall length of the projectile used in that study was 38.1 mm and its diameter was 12.65 mm.
Its initial velocity was 498.1 m/s. The calculated results show that the velocity attenuation and water entry depth of the projectile are consistent with Guo’s experimental results, as shown in Fig. 6. These experiments support the correctness of the numerical calculation method in this paper.

3. Numerical Calculations and Results

In this study, identical projectiles with initial velocities of 300, 400, 500, and 600 m/s were numerically simulated. By considering the mixing of water, water vapor, and air, the influence of velocity on the cavitation characteristics of projectiles entering water vertically was studied. By comprehensively analyzing the differences of underwater cavitation of high-speed projectiles at different speeds, and with different ballistic and fluid-mechanics characteristics, the underwater stability of projectiles at different speeds after entering the water was analyzed and judged.

The three graphs in Fig. 7 show the instantaneous water-phase volume fraction produced by projectiles with four water-entry velocities at three different times after water entry. The cavitation behavior is similar for
Figure 8 Cavitation shapes for different water entry speeds

projectiles with different velocities after water entry, including the trends of radial expansion, axial stretching, and surface closure of the cavitation. Yet, at the same time, the cavity development patterns of water entry at different speeds are distinctly different. In the first 0.2 ms, the top of the cavitation bubble is open to the air. The greater the velocity, the greater the length and diameter of the cavitation bubble (see Fig. 7(a)). When the time increases to 0.5 ms, the top of the cavitation bubble tends to move closer to the axis (see Fig. 7(b)). When the time reaches 0.8 ms, and the velocity is reduced, the degree of surface closure is greater, indicating, as one would expect, that lower speed leads preferentially to surface closure (see Fig. 7(c)).

With the passage of time, the larger the velocity of the projectile, the larger the cavity radius, which is consistent with the cavity state of the projectile with different velocities in different containers in Fan Chunyong’s study (Fan 2022). In addition, for the same time, with the water entry velocity of the projectile increases, the fluctuation range of the water surface around the head of the projectile will be greater, because the drag on the projectile increases with an increase of the velocity, which also makes the water gain greater kinetic energy.

Figure 8 shows cavitation shapes for projectiles with different velocities at different depths. The depths are 1H and 2H, where H is the length of the projectile. The lines in the figure outline the cavity. The comparative results show that for the same penetration depth the projectile with larger initial velocity has a larger diameter of projectile tail cavity, but the shape of the cavity at the head does not change as the velocity changes. From Figs.7 and 8, it can be seen that with the same water entry time, the cavitation length of the projectile increases with the increase of the velocity. When the water entry depth is the same, the cavity shape is basically the same, and the increase of the velocity has little effect on the cavitation diameter. This is consistent with the findings of Zitao Guo. That work showed that projectiles of the same shape have similar cavity sizes at different initial velocities (Guo 2020).

3.1 Analysis of Underwater Characteristics of Projectile

Figure 9 shows the dependence of velocity on time for various initial velocities of the projectile. Table 1 shows the underwater velocity attenuation rate of projectiles with different initial velocities in different containers in Fan Chunyong’s study (Fan 2022). In addition, for the same time, with the water entry velocity of the projectile increases, the fluctuation range of the water surface around the head of the projectile will be greater, because the drag on the projectile increases with an increase of the velocity, which also makes the water gain greater kinetic energy.

Figure 11 shows the drag coefficients of the projectiles as functions of time for various entry velocities for the first 3 ms. Figure 11(b) is an amplified section of the water entry impact stage, and Fig. 11(c) covers the time from 1.0 to 1.2 ms, a stable stage of cavitation flow. This study finds that the projectile drag coefficient

<table>
<thead>
<tr>
<th>Water-entry velocity/ m/s</th>
<th>Final velocity/m/s</th>
<th>Time/ms</th>
<th>Decay rate/% per ms</th>
</tr>
</thead>
<tbody>
<tr>
<td>300</td>
<td>251.0</td>
<td>3</td>
<td>16.3</td>
</tr>
<tr>
<td>400</td>
<td>318.2</td>
<td>3</td>
<td>27.3</td>
</tr>
<tr>
<td>500</td>
<td>378.4</td>
<td>3</td>
<td>40.5</td>
</tr>
<tr>
<td>600</td>
<td>433.1</td>
<td>3</td>
<td>55.6</td>
</tr>
</tbody>
</table>

Fig. 9 Time dependence of projectile velocities

Fig. 10 Time dependence of projectile displacements

Fig. 11 Drag coefficient vs. time after water entry for several initial speeds.: (a) 0-3 ms. (b) 0-0.1 ms. (c) 1-1.2 ms
changes only slightly as the velocity changes (see Fig. 11(a)). A very large drag coefficient peak is generated at the moment of water entry; it then decreases rapidly. The drag coefficient gradually decreases after the cavity flow becomes stable. It is clear from Fig. 11(b) that the peaks of the drag coefficients encountered by projectiles with different initial velocities entering water are similar to each other when they travel under water, but the greater the initial velocity, the earlier the drag coefficient reaches its peak after water entry. The higher the initial velocity, the smaller the drag coefficient, but with only small differences in its value (see Fig. 11(c)). It can be concluded that the initial velocity of water entry has very little effect on the drag coefficient of the conical flat head projectile, but the time for the projectile to reach the peak value decreases as the water entry velocity increases.

3.2 Analysis of underwater cavity flow fields of projectiles

Figure 12 shows the instantaneous bubble wall velocity vectors at a water entry depth 2H for projectiles with different initial velocities. After a projectile enters the water vertically, the cavity walls around it are expanded downward at a certain angle, and the closer to the position of the projectile head, the greater the inclination of the vacuole wall velocity vector and the greater the velocity. The expansion direction of the bubble wall is oblique at the top of the bubble. It can also be seen that the expansion velocity of the cavitation wall is very different for different initial velocities when the projectile is in the same position. The faster the water entry velocity for conical flat head projectiles, the faster the expansion of the cavitation wall.

3.3 Underwater trajectory analysis of projectiles with different head shapes

To study the ballistic stability of projectiles with three head shapes entering water vertically, conical cone head and elliptical flat head projectiles with the same size and material as the conical flat head projectiles are studied.
Before the projectile enters the water, its centroid trajectory is almost a straight line. After entering the water, the projectile trajectory begins to deviate. As the contact area between the projectile and the water increases, the projectile centroid begins to shift significantly.

From Figs 14(a) and (b), the velocity of the projectile is 0 except for the y direction before entering the water, although the projectile extends in these directions. When the head of the projectile is contact with the water, and the lateral force is generated by small disturbance of the water flow. With an increase of the water area, the x and z directions offset increase. When the projectile enters the water, due to differences in the shapes of the heads, the hydrodynamic forces on the projectile are different. As time increases, the conical flat head projectile continues vertically. The elliptical flat head and conical cone head projectiles deviate in the x and z directions. The elliptical flat head projectile is deflected in the x+ direction and z-direction, the conical cone head projectile is deflected in the x+ direction and z+ direction, and the conical flat head projectile is almost not deflected. Therefore, the underwater stability of the conical flat head projectile is very better than that of the elliptical flat head projectile, and the stability of the conical cone head projectile is poor.

Figure 15 shows the velocity changes in the coordinate directions during the process of projectiles vertical water entry with different shapes. The projectile velocity remains almost unchanged before contact with free liquid surface. From Figs. 15(a) and (c), the velocity components in the x and z directions are basically zero from 0-0.7 ms. After 0.7 ms, the projectile head contacts the water, resulting in fluid dynamics in the x and z directions. It can be seen from Fig. 15(b) that at the same time before the projectile enters the water, the speed in the y direction of the conical flat head is the smallest, and the velocity in the y direction of the conical cone head is the largest. This is because the weight of the projectile is nearly the same, the resistance of the conical flat head is larger, and the velocity decays faster. After 0.9 ms, the conical cone head shows poor stability. The centroid is deflected, and the contact area between the head of conical cone head projectile and the water in y-axis direction becomes larger, resulting in a larger fluid resistance in the y-axis direction and a very rapid speed attenuation.

5. CONCLUSION

In this study, through numerical simulation, the influence of velocity and head shape on the vertical water entry process of projectile is studied., and the ballistic characteristics, the flow fields, and the cavitation behaviors are revealed. The underwater stability of the projectiles is also considered. Here are the principal conclusions:

(1) With the water-entry velocity of the projectile is creasing, the more rapid the decay of the velocity and the greater the underwater displacement in a given time, also the greater the velocity attenuation per unit time and the greater the change of the centroid position of the projectile. However, there is no direct effect of the velocity of the projectile on the drag coefficient in the water.

(2) For greater water entry velocity, the attenuation rate of velocity is more.

(3) The pressure at the projectile’s head peaks when it touches the water surface, then drops sharply, and finally levels off. In the stable stage of the flow, the greater the velocity, the greater the pressure on the projectile’s head.

(4) After the flow is stable, the pressure increases with the increase of velocity.

(5) The stability of the projectile with conical flat head is the best, and the stability of the conical cone head projectile is the worst.

Based on the above results, the research team will further test the influence of the two tuning parameters of the cavitation model. The effects of different turbulence models and mesh shapes on numerical calculations will be studied, and the ballistic stability and cavitation characteristics of projectiles with different shapes and different angles of incidence will also be further studied.
CONFLICT OF INTEREST

The authors declare that they have no known competing financial interests or personal relationships that could have appeared to influence the work reported in this paper.

AUTHORS CONTRIBUTION

B. Hao guides and carries out the numerical simulation of this study; Y. G. Lu designs the simulation scheme and the writing of the paper; H. Dai prepares all the charts of this paper.

REFERENCES


characteristics of high-speed water-entry cavity. *Journal of Applied Fluid Mechanics*, 15(6), 1901-1912. https://doi.org/10.47176/jafm.15.06.1310


