Numerical Investigations of Cavitation Nose Structure of a High-Speed Projectile Impact on Water-Entry Characteristics

Qiang Li and Lin Lu *

School of Mechatronics Engineering, North University of China, Taiyuan 030051, China; liqiang17@nuc.edu.cn
* Correspondence: lulin2016@nuc.edu.cn

Received: 11 March 2020; Accepted: 8 April 2020; Published: 9 April 2020

Abstract: In this study, a detailed analysis of the influences of cavitation nose structure of a high-speed projectile on the trajectory stability during the water-entry process was investigated numerically. The Zwart-Gerber-Belamri (Z-G-B) cavitation model and the Shear Stress Transport (SST) k-ω turbulence model based on the Reynolds Averaged Navier–Stokes (RANS) method were employed. The numerical methodology was validated by comparing the numerical simulation results with the experimental photograph of cavitation shape and the experimental underwater velocity. Based on the numerical methodology, the disk and the conical cavitation noses were selected to investigate the water-entry characteristics. The influences of cavitation nose angle and cavitation nose diameter of the projectile on the trajectory stability and flow characteristics were carried out in detail. The variation features of projectile trajectory, velocity attenuation and drag were conducted, respectively. In addition, the cavitation characteristics of water-entry is presented and analyzed. Results show that the trajectory stability can be improved by increasing the cavitation nose angle, but the drag reduction performance will be reduced simultaneously. Additionally, due to the weakening of drag reduction performance, the lower velocity of the projectile will cause the damage of the cavitation shape and the trajectory instability. Furthermore, the conical cavitation nose has preferable trajectory stability and drag reduction performance than the disk cavitation nose.

Keywords: high-speed projectile; water-entry; trajectory stability; cavitation nose structure; numerical investigations

1. Introduction

Water-entry refers to the process in which the projectile passes through the free surface into the water with an initial velocity. Especially for the high-speed projectile, the water-entry process will be accompanied by the appearance of turbulences, phase changes and cavitation phenomenon, which will have a significant effect on the trajectory stability and flow characteristics of the projectile. Besides, the cavitation nose structure of the projectile is the initial part to open the free surface and generate the cavity. Therefore, the research on the influences of the cavitation nose structure of a high-speed projectile on the water-entry characteristics has become more and more important and necessary.

The early research on the water-entry problem mainly focused on spheres [1–3]. With the development of computational and testing technology, a lot of experimental and numerical research of water-entry of other shape bodies have been carried out and published. For the numerical simulations, Reference [4,5] analyzed the hydrodynamic problem of a two-dimensional wedge entering water through free fall motion based on the velocity potential theory. Reference [6]
described the fully nonlinear free-surface deformations of initially calm water caused by the water entry and water exit of a horizontal circular cylinder under the two-dimensional flow conditions. Reference [7] carried out the experimental study to investigate the multiphase flow during water entry of spheres with different surface wettability for low Froude numbers. Reference [8] simulated the water entry process of hydrophobic objects numerically. The water crown, the cavity, and the flow pattern were analyzed. Reference [9] conducted several experimental and numerical investigations to investigate the mechanism of cavity dynamics and vortex evolution through various water entry angles of different objects. Reference [10] discussed the problem of the oblique water entry of an elliptic paraboloid. Pressure, force and dynamics of the wetted surface expansion were assessed. Reference [11] simulated the flow problem of hydrodynamic impact during water entry of solid objects of various shapes and configurations by a two-fluid free surface code based on the solution of the Navier–Stokes equations (NSE). Reference [12] used mathematical modeling, absent simplifying assumptions and coupled with numerical simulation to determine the motions and forces experienced by a sphere penetrating a water surface from an air space above the surface. Reference [13] investigated the water entry problem of a spherical-nose projectile numerically and experimentally. The simulation results such as air cavity shape and the projectile trajectory were compared with the presented experimental data. Reference [14] investigated the complicated hydrodynamics of the horizontal circular cylinder entering water numerically for low Froude numbers. Reference [15] analyzed the compressibility effects of multiphase cavitating flow during the water-entry process. For experimental studies, a lot of useful and reliable results have been obtained, which are of benefit to advance research of the water-entry features of a high-speed projectile. Reference [16] conducted laboratory experiments on conical objects freely moving through water and fixed within imposed flows to determine the dependence of orientational stability on shape. Nevertheless, the water-entry problem with a lower projectile velocity has been conducted primarily. For the experimental investigations, some studies about particle image velocimetry (PIV) test [17–19], trajectory features [20–22], slamming loads [23–26] have been presented. Nevertheless, due to the limitation of experimental conditions and test methods, the present literature review suggests that the experimental studies of water-entry is few and far between, especially the investigation of a high-speed water-entry problem is even less.

In summary, the high-speed water-entry issue is challenging and a topic of great research value. Consequently, in this study, a detailed analysis of water-entry problem was performed by three dimensional numerical simulations of high-speed projectiles with a different cavitation nose structure. The numerical method was verified. The influences of cavitation nose angle and cavitation nose diameter of the projectile on the trajectory stability are presented and have been studied in detail.

2. Numerical simulation method

Governing Equations.

The Volume of Fluid (VOF) multiphase model has been applied to solve the simulation and capture of the characteristics of the gas–liquid–vapor three-phase flow. The governing equations for the turbulent incompressible flow encountered in this research are the three-dimensional RANS equations for the conservation of mass and momentum, given as:

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

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

where \(i=1, 2, 3, j=1, 2, 3\), \(x_i\) and \(x_j\) denote the Cartesian coordinate components. \(P\) represents the pressure. \(u_i\) and \(u_j\) depict the absolute velocity, respectively. \(\mu_t\) is the turbulent viscosity. \(\rho_m\) and \(\mu_w\) are the density and the dynamic viscosity of the mixture, separately.
\[
\rho_{\alpha} = \alpha_v \rho_v + \alpha_i \rho_l + \rho_g (1 - \alpha_v - \alpha_i) \\
\mu_{\alpha} = \alpha_v \mu_v + \alpha_i \mu_l + \mu_g (1 - \alpha_v - \alpha_i)
\]

where \(\alpha_v\) and \(\rho_v\) denote the vapor volume fraction and density, respectively. \(\alpha_i\) and \(\rho_l\) represent the liquid volume fraction and density, respectively. \(\rho_g\) is the gas density. \(\mu_v\), \(\mu_l\) and \(\mu_g\) depict the three-phase dynamic viscosity, respectively.

Turbulence model.

According to the existing study [27], the Shear Stress Transport (SST) \(k-\omega\) turbulence model combines the advantages of stability of the near-wall \(k-\omega\) turbulence model and independent of the external boundary \(k-\epsilon\) turbulence model. The SST \(k-\omega\) turbulence can adapt to a variety of physical phenomenon caused by the pressure gradient changes, and it can utilize the inner viscous layer combined with the wall function to accurately simulate the phenomenon of the boundary layer without the use of easier distortion viscous-attenuation function. In the low Reynolds number regions, the \(k-\omega\) turbulence model was applied. While in the high Reynolds number regions, the \(k-\epsilon\) turbulence model was adopted. Consequently, the SST \(k-\omega\) turbulence model has better practicability and reliability in dealing with boundary problems of different Reynolds numbers. Thus, the SST \(k-\omega\) turbulence model was adopted for closing the numerical prediction in this study.

Cavitation model

According to the available research data [28–30], the Singhal cavitation model, also known as the full cavitation model, is most widely applied for the numerical prediction of the cavitating flows. However, the Singhal cavitation model has its own limitation if applied to evaporation. As the vapor volume fraction increases, the nucleation site density needs to decrease correspondingly. To modify this situation, the Zwart-Gerber-Belamri cavitation model [31] was adopted in this study and employed as the governing equation of phase transition as follows:

\[
\frac{\partial (\rho \alpha_v)}{\partial t} + \frac{\partial (\rho \alpha_v \mu_v)}{\partial x_i} = R_v - R_c \\
R_v = F_{vap} \frac{3 \alpha_v (1 - \alpha_v) \rho_v}{R_b} \sqrt{\frac{2}{3}} \frac{(p_v - p)}{\rho_l} \\
R_c = F_{cond} \frac{3 \alpha_v \rho_v}{R_b} \sqrt{\frac{2}{3}} \frac{(p_v - p)}{\rho_l}
\]

where \(R_v\) and \(R_c\) represent the vaporization and condensation rates per unit volume, respectively. The bubble radius \(R_b = 1\mu m\); \(\alpha_{nuc}\) denotes nucleation site volume fraction and equals to \(5 \times 10^{-4}\); The evaporation coefficient \(F_{vap} = 50\) and the condensation coefficient is \(F_{cond} = 0.001\); \(p_v\) represents the saturation vapor pressure.

3. Validation of numerical method

In order to verify the accuracy of the numerical simulation method in this study, comparisons of numerical predictions with experimental data and the photograph in Reference [32] were applied. A circular cylinder \((r = 12.7mm, h = 25.4mm)\) was selected as the experimental and numerical model. The initial velocity of the projectile \((v)\) was set to 603m/s. The projectile enters the water along the vertical direction. A 3D numerical simulation was carried out. The computational domain and the grid are shown in Figures 1 and 2. The number of the entire computational grid was about \(0.7 \times 10^5\). The Dynamic Mesh technology was carried out to deal with the grid moving of the circular cylinder.
The numerical results of the underwater velocity of a steel sphere was compared with the experimental data and shown in Figure 3. As illustrated in Figure 3, the underwater velocity of the circular cylinder decreased with the increasing of underwater time. The numerical simulation results were in good agreement with the experimental results, and the errors were less than 7.5%. Moreover, the photograph in Figure 4 was taken by a high-speed camera, which shows the shape of water-entry cavity at different times. As shown in Figure 4, the steel sphere was able to penetrate the free surface to enter the water, and the water-entry cavitation appeared and expanded. After a comparison between the experimental and the numerical prediction results, it was found that the cavity shape of the numerical prediction agreed with experimental photograph with reasonable accuracy. Consequently, it was presumed that the numerical simulation method would be applicable and reliable for the trajectory characteristics and stability of a high-speed water-entry projectile.
Figure 4. Experimental graph and numerical results of cavity shape of a circular cylinder.

4. Projectile structure impact on water-entry characteristics

4.1. Computational model and boundary conditions

The computational model of the projectile is illustrated in Figure 5, which is the modified model of the original projectile in Reference [33]. The parameter of the modified projectile is shown in Table 1. The projectile has a conical nose followed by a small annular groove, as shown in Figure 5.

| Projective diameter (D) | 12.7mm |
|------------------------|--------|
| Conical nose angle     | 100°   |
| Mass                   | 90g    |
| Length (L)             | 99mm   |

Figure 5. Computational model of water-entry projectile.

Figure 6. shows that the computational domain is a cuboid (800 mm (8L) × 800 mm (8L) × 2000 mm (20L)) surrounding the projectile. As can be seen from Figure 6, the longitudinal axis of the projectile and +Z is the penetration direction. The computational domain was divided into two parts by the free surface, including the air domain and the water domain. The length of the air
domain and the water domain was 700 mm (7L) and 1300 mm (13L), respectively. The settings of boundary conditions are illustrated in Figure 6(b). The outlet boundary was set to the pressures outlet with User Define Function (UDF). The no-slip wall boundary condition was applied on the projectile. On the cuboid surface the free slip wall boundary was defined. The time shaft of the process of water-entry and water-exit is shown in Figure 6(a). As shown in Figure 6(a), the time was zero when the nose of the projectile touched the free surface, and \( t > 0 \) after water-entry.

![Computational domain and boundary conditions](image)

**Figure 6.** Computational domain and boundary conditions: (a) Computational domain; (b) Boundary conditions.

4.2. Grid generation and independent inspection

4.2.1. Grid generation

The quality of the computational grid directly determines the accuracy and reliability of the numerical prediction results. In this study, an unstructured grid generated by the commercial code Gambit was adopted for all the computational domains, as shown in Figure 7. The first layer of grids around the projectile was 0.5 mm. Then the grids expand around the projectile body surface. The grid size close to the projectile surface was 6 mm and the grid size of other computational regions was 12 mm. The number of entire computational domain grids was approximately \( 0.8 \times 10^6 \). In addition, the local grid around the projectile was refined in order to ensure a better prediction of trajectory characteristics. A zoomed view of the projectile grids is illustrated in Figure 7. In order to reflect the actual situation of the movement process of the projectile, the dynamic grid technology (Fluent, 2012) was used to deal with the grid deformation. Meanwhile, the spatial discretization of pressure field adopted the PRESTO! format. The PISO algorithm was used to solve the coupling of pressure and velocity flow fields.
4.2.2. Grid independent inspection and grid convergence index

The grid independent inspection was applied in order to ensure the accuracy and precision of the numerical simulation. Three different sizes of grids were applied. The computational grids were regenerated by reducing and increasing the size of the first layer of grids. The first layers of regenerated grids around the projectile were 0.33 mm and 0.75 mm, respectively. The thin grid number was approximately $6 \times 10^6$. The refined grid number was about $6 \times 10^7$. Additionally, the grid convergence index (GCI) was applied. The grid refinement ratio $r=1.414$. The velocities of three different grids are $v_1=845.4696\text{m/s}$, $v_2=845.4055\text{m/s}$, $v_3=845.3425\text{m/s}$ at $t=0.6\text{ms}$, respectively $p=0.005947$, $\text{GCl}_{12}=0.022941$, $\text{GCl}_{23}=0.022549$, $\text{GCl}_{12}/\text{GCl}_{23}=1.013198$, which was approximately one and indicated that the grids were well within the asymptotic range of convergence. Figures 8 and 9 show the pressure distribution and the underwater velocity attenuation with different grid numbers, respectively. As can be seen from Figures 8 and 9, there was no significant difference between the three kinds of grids. The errors of different water-entry characteristics were less than 1% after the grids were regenerated. Consequently, based on the computing power of the current devices, the undefined grid ($0.8 \times 10^8$) was selected for numerical simulation and analysis.
Figure 8. Pressure distribution of different grid numbers: (a) Thin; (b) Unrefined; (c) Refined.

Figure 9. Velocity attenuation of refined and unrefined grids.

4.3. The compressibility of the liquid

Based on the results of some high-speed water-entry problem research papers [15,34–39], as the velocity is under 1000 m/s (890 m/s in this paper), the influence of the compressibility of the liquid on the stability of the projectile is not obvious. Figures 10 and 11 show the underwater velocity attenuation and the trajectory along the Z direction, respectively. As shown in Figures 10 and 11, there is typically almost no visible difference between the compressible and incompressible liquid. Consequently, the compressibility of the liquid was not applied.
4.4. Results and discussions

4.4.1. Effect of cavitation nose angle of the projectile

During the process of water-entry, the projectile nose was the initial part to come into contact with the free surface. Therefore, the structure of cavitation nose of the projectile had a direct effect on the trajectory stability and flow characteristics of water-entry. In this section, maintaining the initial velocity of the projectile equaled 890 m/s, the water-entry results with three conical different cavitation nose angles (\( \alpha=60^\circ,90^\circ,120^\circ \)) were investigated. Figures 12 and 13 show the computational models and the local cavitation noses with different \( \alpha \), respectively.
Figure 12. Computational model of three different cavitation nose angles.

Figure 13. Local cavitation nose angles.

As can be seen from the 3D trajectory in Figure 14 (a), the cavitation nose angle ($\alpha$) had a great influence on the trajectory stability of water-entry. The trajectory was relatively stable at $\alpha=60^\circ,90^\circ$, while the deflection of trajectory was the most obvious when $\alpha$ was $120^\circ$. As illustrated in Figure 14 (b) and (c), the values on the X-axis and Y-axis represented the projection of the trajectory deflection on the X-axis and Y-axis relative to the predetermined trajectory. The trajectory did not have much deflection during the initial stage of water-entry, and it was basically consistent with the predetermined trajectory. With the increase of the water depth, the deflection happened on all the projectiles. By comparing Figure 14 (b) and (c), for $\alpha=60^\circ,90^\circ$, deflections were found along the Y direction, which were more obvious than those along the X direction. In addition, the coincidence of the X-axis and Y-axis values for different angles came into sight. The different nose angle caused the difference in the asymmetry of the shock wave, incompleteness of cavity coverage and the unevenness of the pressure distribution, which affected the features of the trajectory deflection directly. Besides, the randomness of the deflection leads to the coincidence of the X-axis and Y-axis values for different angles. However, because the 3D trajectory of the projectile was composed of the deflection of the X-axis and Y-axis simultaneously, the coincidence of the X-axis and Y-axis values for different angles was possible and reasonable, and the trajectories for different angles have their diversity. Moreover, the projectile of $\alpha=90^\circ$ had the best stabilization along the X direction, and the deflection of $\alpha=120^\circ$ was most prominent along the X direction. There was not much difference in deflections of all the projectiles along the Y direction.
Figure 14. Centroid trajectory of projectile with different $\alpha$: (a) 3D centroid trajectory; (b) Centroid deflection along X direction; (c) Centroid deflection along Y direction.

Figure 15 shows the velocity attenuation with different $\alpha$. In Figure 15, the projectile velocity basically remained constant during the initial stage of water-entry. However, the velocity attenuation came into sight with the immersion of the projectile. The attenuation trends of all the projectiles were generally the same. The velocity of $\alpha=120^\circ$ decayed most, while the minimum attenuation of velocity happened on the projectile of $\alpha=60^\circ$, which indicates that the projectile of $\alpha=60^\circ$ had better drag reduction performance. From Figure 16 we can see, due to the trajectory stability of $\alpha=60^\circ,90^\circ$, the total drag of these two angles varied little, which was consistent with the results as shown in Figure 14 (a). However, for $\alpha=120^\circ$, after a period of water-entry, the total drag of the projectile increased rapidly. It was mainly because the cavitation shape of the projectile had been damaged and had collapsed, which resulted in the rapid attenuation of the velocity, as shown in Figure 15.
Figure 15. Velocity attenuation with different $\alpha$.

Figure 16. Drag with different $\alpha$ along the Z direction.

Figure 17 shows the contours of the cavity of water-entry with $\alpha=60^\circ, 90^\circ, 120^\circ$. The blue part and the red part represent the air and water, respectively. For three different $\alpha$, all the projectiles were able to smoothly penetrate the free surface to enter the water and generate the water-entry cavitation. When the projectile entered the water completely, the cavitation closure of the free surface gradually appeared. Then, the projectile continued to dive surrounded by the cavitation. Besides, for all three projectiles, the shapes of cavitation were semi-symmetrical and stable. The obvious collapse and deformation of cavitation had not occurred. Furthermore, the cavitation of free surface of $\alpha=60^\circ$ closed first, and the deep cavitation closure did not appear. The projectile of $\alpha=60^\circ$ penetrated with a circumstance of a higher density that was detrimental to the stability of the trajectory. However, the deep cavitation closure happened to the projectiles of $\alpha=90^\circ, 120^\circ$. Especially, for the development of the deep cavitation, closure was more sufficient. The cavity was filled completely with vapor with a smaller density that was generated by the cavitation after the deep cavitation closed, which caused the trajectory of the projectile of $\alpha=90^\circ$, to have a smaller deflection and was more stable, as shown in Figure 14(a). Thus, it is indicated that $\alpha=90^\circ$ was more conductive to the formation of the deep cavitation and the stability of the trajectory than $\alpha=60^\circ, 120^\circ$, and the cavitation nose angle had a direct influence on the shape, development and stability of water-entry cavitation.
Through the above analysis, it can be concluded that the overall processes of water-entry of $\alpha=60^\circ, 90^\circ$ were better than that of $\alpha=120^\circ$. The projectile of $\alpha=60^\circ$ had better drag reduction performance, and the projectile of $\alpha=90^\circ$ had better trajectory stability. In the scope of $\alpha=60^\circ$–$120^\circ$, the trajectory stability can be improved by increasing the cavitation nose angle, but the drag reduction performance will be reduced at the same time. Due to the weakening of drag reduction performance, the underwater velocity of the projectile will be at a very low level, which causes the cavitation shape to be destroyed and the trajectory stability will also be affected directly.

4.4.2. Effect of cavitation nose diameter of the projectile

In the previous section, the influence of conical cavitation nose on the process of water-entry was carried out and analyzed. In this section, the disk cavitation noses with three different cavitation nose diameters ($r=0.5$ mm, 1.0 mm, 1.5 mm) were selected to investigate the water-entry results. Figures 18 and 19 show the computational models and the local cavitation noses with different $r$, respectively.
Figure 19. Local cavitation nose diameters.

Figure 20 shows the 3D trajectory stability and the deflection of the projectile centroid with different \( r \), respectively. As illustrated in Figure 20 (a), the trajectories of all the projectiles kept good stabilization during the initial stage of the water-entry, and then the deflection of the trajectory rose with the increase of time. The trajectory of \( r=1 \) mm was more stable when it entered the water, while the trajectories of the other two diameters had more obvious deflection characteristics. It demonstrated that the cavitation nose diameter had a strong impact on the trajectory stability of the projectile water-entry. As can be seen from Figure 20(b) and (c), the apparent deflection along X and Y directions had not happened on the projectile of \( r=1 \) mm, which was in good agreement with Figure 20(a). For other projectiles, the deflections along the Y direction were basically the same, while the projectile of \( r=1.5 \) mm had a greater trajectory deflection than that of \( r=0.5 \) mm.

Figure 20. Centroid trajectory of projectile with different \( r \): (a) 3D centroid trajectory; (b) Centroid deflection along the X direction; (c) Centroid deflection along the Y direction.
Centroid deflection along X direction; (c) Centroid deflection along Y direction.

Figure 21 shows the velocity attenuation with different $r$. As shown in Figure 21, during the beginning stage of water-entry, the attenuation trends of all the projectiles were generally the same and approximately linear. However, with the time increase of water-entry, the trend of $r=1$ mm remained linear, and there was a trend of accelerated attenuation for $r=0.5$ mm, 1.5 mm. Especially, the velocity attenuation of $r=1.5$ mm was more significant. It can be concluded that the projectile of $r=1$ mm had a better drag reduction performance. Furthermore, from the variation of the total drag in Figure 22 we can know, when the projectile just touches the free surface, the drag of all three projectiles increases rapidly and is kept steady for a period of time, which accords with the features of velocity attenuation of the projectile in Figure 21. Thereafter, the drag of all three projectiles increases to different degrees. It can be implied that the cavitation shape of the projectile has been damaged to some extent, and the drag reduction performance is constantly losing, which leads to the rapid velocity attenuation performance along the Z direction and the increase of the deflection along X and Y directions, as illustrated in Figure 20(b) and (c). In addition, the drag of $r=1.5$ mm increases most dramatically and decreases slightly in the end. The main reason for the decrease of the drag of $r=1.5$ mm is that there is a destabilization of the trajectory that causes the decrease of the Z direction component of the drag.

Figure 21. Velocity attenuation with different $a$.

Figure 22. Drag with different $a$ along the Z direction.

Figure 23 shows the contours of the cavity of vertical water-entry with different $r$. All three projectiles can successfully complete the process of water-entry, and then the free surface is slowly
closed. The obvious deflection of trajectory did not come into appearance. The shape of cavitation was semi-symmetrical. The development of the cavity of \( r=0.5 \) mm, 1 mm was more sufficient than that of \( r=1.5 \) mm. The projectile of \( r=1 \) mm displayed the most stable cavity shape, which was identical with the previous results of trajectory stability. In particular, for \( r=1.5 \) mm, the structure of cavitation was damaged with the submersion of the projectile. Some water penetrated the interior of the cavity and part of the projectile was wet with water, which caused the trajectory to deviate from the \( Z \) direction as shown in Figure 21(a), and the sharp attenuation of the velocity and the increase of total drag of the projectile, as shown in Figures 21 and 22.

Based on the above results, it can be found that the overall process of \( r=1 \) mm water-entry is better than those of \( r=0.5 \) mm, 1.5 mm. Among all three projectiles, the projectile of \( r=1 \) mm has better trajectory stability and drag reduction performance. However, the drag and the trajectory deflection of the disk cavitation nose is larger and more intense than those of the conical cavitation nose. Thus, for the water-entry process, the structure of the conical cavitation nose is better for the formation and development of the cavitation, and has the preferable trajectory stability and drag reduction performance than the structure of the disk cavitation nose.
**Figure 23.** Contours of the cavity of vertical water-entry with different r: (a) r=0.5 mm; (b) r=1.0 mm; (c) r=1.5 mm.

5. Conclusions

In this study, a detailed analysis of the influences of the cavitation nose structure of a high-speed projectile on the water-entry problems was investigated numerically. The Z-G-B cavitation model and the SST k-ω turbulence model based on the RANS method were employed. The numerical method was verified. The disk and the conical cavitation noses were selected to investigate the water-entry features. The influences of cavitation nose angle and cavitation nose diameter of the projectile on the trajectory stability and cavitation characteristics have been presented and studied in detail. The primary findings and conclusions are as follows:

1. In the scope of \( \alpha = 60^\circ - 120^\circ \), the overall processes of water-entry of \( \alpha = 60^\circ, 90^\circ \) are better than that of \( \alpha = 120^\circ \). The projectile of \( \alpha = 60^\circ \) has better drag reduction performance, and the projectile of \( \alpha = 90^\circ \) has better trajectory stability.

2. The trajectory stability can be improved by increasing the cavitation nose angle, but the drag reduction performance will be reduced at the same time. Due to the weakening of drag reduction performance, the lower velocity of the projectile will cause the damage of the cavitation shape and the trajectory instability.

3. Among all the projectiles with three different \( r \), the projectile of \( r=1 \) mm had a better trajectory stability and drag reduction performance. For the water-entry process, the conical cavitation nose has the preferable trajectory stability and drag reduction performance than the disk cavitation nose.

**Author Contributions:** The manuscript was written by Q.L. and L.L.; all authors discussed the original idea; conceptualization, Q.L. and L.L.; methodology, Q.L. and L.L.; software, L.L.; validation, Q.L., L.L.; formal analysis, L.L.; investigation, L.L.; resources, Q.L.; data curation, Q.L.; writing—original draft preparation, Q.L. and L.L.; writing—review and editing, Q.L. and L.L.; visualization, Q.L. and L.L.; supervision, Q.L.; project administration, Q.L.; funding acquisition, Q.L. and L.L. All authors have read and agreed to the published version of the manuscript.

**Funding:** This research was funded by the National Natural Science Foundation of China, grant number 51175481, the Natural Science Foundation of Shanxi Province of China, grant number 201801D221038, the University Technological Innovations Plan of Shanxi Province of China, grant number 201802080 and the Science Research Foundation of North University of China, grant number 2017002.
Conflicts of Interest: The authors declare no conflicts of interest

References

1. Gilbarg, D.; Anderson, R.A. Influence of atmospheric pressure on the phenomena accompanying the entry of spheres into water. J. Appl. Phys. 1948, 19, 127–139.
2. May, A.; Woodhull, J.C. The virtual mass of a sphere entering water vertically. J. Appl. Phys. 1950, 21, 1285–1289.
3. May, A. Effect of surface condition of a sphere on its water-entry cavity. J. Appl. Phys. 1951, 22, 1219–1222.
4. Wu, G.X.; Sun, H.; He, Y.S. Numerical simulation and experimental study of water entry of a wedge in free fall motion. J. Fluid Struct. 2004, 19, 277–289.
5. Wu, G.X. Numerical simulation of water entry of twin wedges. J. Fluid Struct. 2006, 22, 99–108.
6. Zhu, X.; Faltinsen, O.M.; Hu, C. Water entry and exit of a horizontal circular cylinder. J. Offshore Mech. Arct. Eng. 2007, 129, 253–264.
7. Li, D.; Zhang, J.; Zhang, M.; Huang, B.; Ma, X.; Wang, G. Experimental study on water entry of spheres with different surface wettability. Ocean Eng. 2019, 187, 106123.
8. Shentu, J.; Zhao, T.; Li, D.; Zhao, X. Numerical simulations for water entry of hydrophobic objects. Ocean Eng. 2019, 190, 106485.
9. Mirzaei, M.; Taghvaei, H.; Golneshan, A.A. Improvement of cavity shape modeling in water-entry of circular cylinders by considering the cavity memory effect. Appl. Ocean Res. 2020, 97, 102073.
10. Scolan, Y.M. Oblique water entry of a three dimensional body. Int. J. Naval. Archit. Ocean Eng. 2014, 6, 1197–1208.
11. Gu, H.B.; Qian, L.; Causon, D.M.; Mingham, C.G.; Lin, P. Numerical simulation of water impact of solid bodies with vertical and oblique entries. Ocean Eng. 2014, 75, 128–137.
12. Abraham, J.; Gorman, J.; Reseghetti, F.; Sparrow, E.; Stark, J.; Shepard, T. Modeling and numerical simulation of the forces acting on a sphere during early-water entry. Ocean Eng. 2014, 76, 1–9.
13. Erfanian, M.R.; Anbarsooz, M.; Rahimi, N.; Zare, M.; Moghiman, M. Numerical and experimental investigation of a three dimensional spherical-nose projectile water entry problem. Ocean Eng. 2015, 104, 397–404.
14. Iranmanesh, A.; Passandideh-Fard, M. A three-dimensional numerical approach on water entry of a horizontal circular cylinder using the volume of fluid technique. Ocean Eng. 2017, 130, 557–566.
15. Chen, C.; Sun, T.; Wei, Y.; Wang, C. Computational analysis of compressibility effects on cavity dynamics in high-speed water-entry. Int. J. Naval Architec. Ocean Eng. 2019, 11, 495–509.
16. Amin, K.; Mac Huang, J.; Hu, K.J.; Zhang, J.; Ristroph, L. The role of shape-dependent flight stability in the origin of oriented meteorites. Proc. Natl. Acad. Sci. USA 2019, 116, 16180–16185.
17. Jalalisendi, M.; Shams, A.; Panciroli, R.; Porfiri, M. Experimental reconstruction of three-dimensional hydrodynamic loading in water entry problems through particle image velocimetry. Exp. Fluids. 2015, 56, 41.
18. Panciroli, R.; Shams, A.; Porfiri, M. Experiments on the water entry of curved wedges: High speed imaging and particle image velocimetry. Ocean Eng. 2015, 94, 213–222.
19. Shams, A.; Jalalisendi, M.; Porfiri, M. Experiments on the water entry of asymmetric wedges using particle image velocimetry. Phys. Fluids 2015, 27, 027103.
20. Shi, H.H.; Takami, T. Hydrodynamic behavior of an underwater moving body after water entry. Acta. Mech. Sin. 2001, 17, 35–44.
21. Barjasteh, M.; Zeraatgar, H.; Javaherian, M.J. An experimental study on water entry of asymmetric wedges. Appl. Ocean Res. 2016, 58, 292–304.
22. Chen, T.; Huang, W.; Zhang, W.; Qi, Y.; Guo, Z. Experimental investigation on trajectory stability of high-speed water entry projectiles. Ocean Eng. 2019, 175, 16–24.
23. Alauoi, A.E.M.; Nême, A.; Tassin, A.; Jacques, N. Experimental study of coefficients during vertical water entry of axisymmetric rigid shapes at constant speeds. Appl. Ocean Res. 2012, 37, 183–197.
24. Nila, A.; Vanlanduit, S.; Vepa, S.; Van Paepegem, W. A PIV-based method for estimating slamming loads during water entry of rigid bodies. Meas. Sci. Technol. 2013, 24, 045303.
25. Van Nuffel, D.; Vepa, K.S.; De Baere, I.; Lava, P.; Kersemans, M.; Degrieck, J.; De Rouck, J.; Van Paepegem, W. A comparison between the experimental and theoretical impact pressures acting on a horizontal quasi-rigid cylinder during vertical water entry. Ocean Eng. 2014, 77, 42–54.
26. Dong, C.; Sun, S.; Song, H.; Wang, Q. Numerical and experimental study on the impact between a free falling wedge and water. *Int. J. Naval Architec. Ocean Eng.* 2018, 1-11.
27. Menter, F.R. Two-equation eddy-viscosity turbulence models for engineering applications. *AIAA J.* 1994, 32, 1598-1605.
28. Singhal, A.K.; Athavale, M.M.; Li, H.; Jiang, Y. Mathematical basis and validation of the full cavitation model. *J. Fluids Eng.* 2002, 124, 617-624.
29. Watanabe, T.; Kawamura, T.; Takekoshi, Y.; Maeda, M.; Rhee, S.H. Simulation of steady and unsteady cavitation on a marine propeller using a RANS CFD code. In Proceedings of the 5th International Symposium on Cavitation, Osaka, Japan, 1-5 November 2003.
30. Salvatore, F.; Streckwall, H.; Van Terwisga, T. Propeller cavitation modelling by CFD-results from the VIRTUE 2008 Rome workshop. In Proceedings of the First International Symposium on Marine Propulsors, Trondheim, Norway, 22-24 June 2009.
31. Zwart, P.J.; Gerber, A.G.; Belamri, T. A two-phase flow model for predicting cavitation dynamics. In Proceedings of the 5th International Conference on Multiphase Flow, Yokohama, Japan, 30 May–3 June 2004.
32. Guo, Z. Research on Characteristics of Projectile Water Entry and Ballistic Resistance of Targets under Different Mediums. Ph.D. Thesis, Harbin Institute of Technology, Harbin, China, 2012. (In Chinese)
33. Polovnev, A.A.; Khasiakhmetov, V.S. U.S. Patent No. 8,082,851. Washington, DC: U.S. Patent and Trademark Office, 2011.
34. Huang, C. Research of Trajectory Characteristics of Supersonic-Supercavitating Projectiles. Ph.D. Thesis, Northwestern Polytechnical University, Xi’an, China, 2017. (In Chinese)
35. Huang, C.; Luo, K.; Bai, J.; Wang, Z.; Jiang, B. Influence of liquid’s compressibility on supercavitating flow. *J. Shanghai Jiao Tong Univ. (Sci.)* 2016, 50, 1241–1245. (In Chinese)
36. Huang, C.; Dang, J.J.; Li, D.J.; Luo, K. Influence of the transonic motion on resistance and cavitation characteristics of projectiles. *Acta Armamentarii* 2016, 37, 1482–1488. (In Chinese)
37. Chen C. Multiphase Flow Characteristics of Subsonic and Transonic Water-entry for Small Moving Body. Ph.D. Thesis, Harbin Institute of Technology, Harbin, China, 2019. (In Chinese)
38. Li, D.; Huang, B.; Zhang, M.; Wang, G.; Liang, T. Numerical and theoretical investigation of the high-speed compressible supercavitating flows. *Ocean Eng.* 2018, 156, 446-455.
39. Liang, T.; Zhang, M.; Li, D.; et al. Numerical simulation of high speed compressible supercavitation flow. In: Proceedings of the 14th National Congress on Hydrodynamics & 28th National Conference on Hydrodynamics, Changchun, China, August 8-10, 2017; pp. 357–362 (in Chinese).

© 2020 by the authors. Licensee MDPI, Basel, Switzerland. This article is an open access article distributed under the terms and conditions of the Creative Commons Attribution (CC BY) license (http://creativecommons.org/licenses/by/4.0/).