

Article

Water Entry Characteristics of a Truncated-Cone Object with the Effect of Compressibility

Ping Liu ^{1,†}, Mengcheng Zeng ^{2,†}, Yi Shen ¹, Jiahao Huang ², Zhi Yan ² and Yongliang Xiong ^{2,*} ¹ Institute of System Engineering, Mianyang 621900, China² School of Aerospace Engineering, Huazhong University of Science and Technology, Wuhan 430074, China

* Correspondence: xylcfd@hust.edu.cn

† These authors contributed equally to this work.

Abstract

Trans-medium vehicles, including supercavitating torpedoes, submarine-launched projectiles, and high-speed hydroballistic bodies, demonstrate increasingly diverse applications in crossing the air–water interface. For such vehicles, the truncated-cone (flat-headed) configuration represents a geometry of significant engineering relevance, as it is widely adopted in the nose sections of supercavitating projectiles and certain underwater ballistic penetrators where the flat head promotes rapid vaporization and cavity generation during high-speed water entry. The air-to-water transition process typically generates extreme hydrodynamic impact loads due to complex multiphase flow and fluid–structure coupling interactions, with water compressibility playing a significant role under hydroballistic conditions. This study focuses on the water-entry regime at velocities ranging from 300 to 1100 m/s, corresponding to hydroballistic speeds relevant to supercavitating vehicles (e.g., the Shkval torpedo operates at approximately 370 m/s) and the initial impact phase of high-speed trans-medium projectiles. Parametric studies are conducted with varying entry velocities (Mach 0.20~0.73 in water), impact angles, and structural dimensions to systematically investigate their effects on the peak slamming overload, using a dynamic mesh technique coupled with a VOF multiphase model with compressibility effects for both air and water phases. The results demonstrate that compressibility effects induce a pronounced air cushion effect during water impact, wherein compression waves generated during high-speed entry dominate the load formation process. Velocity is identified as the most sensitive factor affecting peak overload, followed by structural size parameters. The findings provide valuable guidance for the protective design of high-speed water-entry structures operating in the hydroballistic regime.

Keywords: water entry; multiphase flow; impact load; numerical simulation; compressibility; slamming overload; trans-medium



Academic Editor: Kung-Ming Chung

Received: 14 June 2026

Revised: 2 August 2026

Accepted: 22 August 2026

Published: 27 August 2026

Copyright: © 2026 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 \(CC BY\)](https://creativecommons.org/licenses/by/4.0/) license.

1. Introduction

Water entry refers to the process by which an object impacts and penetrates the water surface from air. Currently, trans-medium craft capable of crossing the air–water interface demonstrate asymmetric combat advantages by leveraging their dual-domain maneuverability [1]. The impact during this process may result in tremendous hydrodynamic loads acting on the projectile, potentially leading to structural failure. The water entry problem involves complex physical processes across solid, liquid, and gas phases, including interactions between the multiphase flow field and structures. This process is usually

accompanied by flow separation, jet evolution, and cavity formation, exhibiting characteristics of high coupling, strong nonlinearity, unsteadiness, and intense disturbance. The water entry process represents a critical stage in cross-media navigation. Therefore, it is essential to investigate the underlying physics of water entry and the primary factors influencing the structural response of vehicles during this process.

The early studies of water entry of structures were carried out by analytical analyses and experimental observation. In 1929, Von Kármán [2] derived the formula for water entry impact based on the principle of momentum conservation, and pointed out that the reason for the development of the impact load during water entry is that the vehicle needs to accelerate the water at the head and inhale the air at the rear. In 1932, Wagner [3] refined Von Kármán's theoretical results by considering the rise of the water level that is in contact with the surface of the object. Logvinovich [4] developed a hydrodynamic theory of free-boundary flows by analyzing the free-boundary flow problems during cavitation evolution and water entry. From the experimental perspective, in 1908, Worthington [5] utilized stroboscopic photography to document splash phenomena on the water surface and captured images of underwater cavity development, systematically characterizing the evolution of spray dynamics and submersed cavity morphology to investigate the influences of sphere surface roughness and liquid viscosity on the cavity evolution during water entry of spheres and liquid droplets, respectively.

Subsequently, a large number of studies have adopted similar experimental techniques to study the water entry process. For example, Bivin et al. [6] experimentally investigated the vertical water entry of spheres and disks with different masses, quantifying drag and added-mass effects and deriving analytical expressions for cavity evolution. Lee et al. [7,8] developed a modeling framework to simulate cavity evolution during high-speed water entry of rigid projectiles and investigated cavitation dynamics of spheres, with the obtained computational predictions showing strong alignment with experimental observations. Jalalisendi et al. [9] experimentally investigated the three-dimensional hydrodynamic load distribution during the vertical water entry of a rigid wedge. Using planar particle image velocimetry (PIV) and the incompressible Navier–Stokes equations, they reconstructed the three-dimensional flow field and pressure distribution, providing an effective approach for analyzing unsteady fluid–structure interactions. Zhao et al. [10] experimentally investigated the high-speed vertical water entry of spheres, focusing on the influence of density and velocity on cavitation evolution, and established a predictive model based on the experimental results.

With the rapid development of computational capabilities, numerical simulation has become an important approach for studying water entry problems. Stenius et al. [11] used a multi-material Arbitrary Lagrangian–Eulerian (ALE) method to simulate the impact loads and structural responses of elastic wedges under different entry velocities, impact angles, and material properties. Panciroli et al. [12] investigated the hydroelastic water entry of elastic wedges through experiments and numerical simulations, validating SPH for modeling hydroelastic effects with weak liquid–gas coupling.

Yang [13] incorporated the time-varying rate of added mass into simulations of submarine-launched projectile water exit, improving the treatment of complex fluid–structure interactions and simulation accuracy. Yang [14] investigated the viscoelastic water entry of a three-dimensional sphere, showing that viscoelasticity significantly affects the sphere's stress distribution, free-surface pressure, and cavity-wall stability under high-speed impact. Zhang [15] proposed a diffusive wetting model for WCSPH that accurately simulates wetting effects, flow separation, and free-surface breaking throughout the water-entry and water-exit processes.

Moreover, research on water entry of complex structures and specific operating conditions is also ongoing. Hou et al. [16] experimentally investigated the oblique water entry of hollow cylinders, while Yuan et al. [17] investigated the simultaneous water entry of two hollow cylinders by numerical simulation. Qi et al. [18] studied the impact hydrodynamics of a typical hull obliquely entering water via drop tests. Li et al. [19] used the smoothed particle hydrodynamics (SPH) method to study the influence of the angle of entry and different initial velocities on the 3D real-scale lifeboat. Gan et al. [20] numerically investigated the water-entry impact performance of a bionic aerial-underwater vehicle mimicking flying fish. Wang et al. [21] studied water entries in flowing water of twin spheres side-by-side. Wang et al. [22,23] experimentally investigated asynchronous parallel high-speed vertical and oblique water entry of cylinders. Furthermore, scholars [24–27] have also conducted research on the water-entry process with consideration of floating ice. These studies show that numerical methods can reproduce important transient features of water entry under various geometric and operating conditions.

The velocity range of 300–1100 m/s investigated in this study corresponds to the hydroballistic regime of water entry, which is distinct from the conventional splashdown conditions associated with low-speed water entry scenarios. This velocity range is physically motivated by several engineering applications. The maximum velocity studied is well within the achievable range of existing high-speed experimental facilities (e.g., gas-gun launched objects or water-entry test rigs). Moreover, the speed of sound in water is approximately 1500 m/s, meaning that 1100 m/s remains subsonic relative to the water medium. Thus, the flow is strongly compressible but does not cross the sonic threshold in water. This velocity was selected precisely because its physical mechanism represents a realistic upper bound for many practical hydroballistic and supercavitating scenarios, while still allowing us to focus on the initial stage of water impact before cavitation and structural deformation dominate [28,29]. Advanced conceptual designs for supercavitating vehicles have proposed velocities exceeding 600 m/s [30]. Recent studies by Wu et al. [31] and Yang et al. [32] have investigated comparable velocity ranges (subsonic to supersonic in water) for axisymmetric and rectangular projectiles, respectively, demonstrating that compressibility effects become dominant at these hydroballistic speeds. Previous studies of high-speed axisymmetric and rectangular projectiles have shown that compressibility can affect compression-wave generation, air cushioning, cavity evolution, and impact loading. In contrast, studies conducted at much lower velocities may not fully represent the physical mechanisms governing hydroballistic water entry.

The existing literature indicates that the compressibility of water and air can modify the air-cushioning effect and generate compression waves during impact. These waves influence the pressure field, the timing of contact between the body and water, cavity development, and the evolution of slamming loads [31–38]. However, most existing studies have focused on individual factors or on geometries such as wedges, spheres, axisymmetric bodies, and rectangular projectiles. A systematic assessment of how entry velocity, nose area, and half-cone angle jointly affect the transient impact dynamics of meter-scale truncated-cone bodies is still lacking.

Accordingly, the specific research gap addressed in this study is the lack of a systematic and quantitatively focused analysis of compressible water-entry impact for truncated-cone configurations in the hydroballistic velocity regime. In particular, the relationships among the geometric parameters of the flat nose, the half-cone angle, the compressibility-induced flow features, and the resulting peak slamming loads remain insufficiently established. Addressing this gap is important because the truncated-cone geometry is directly relevant to high-speed cross-media vehicles, whereas the load-generation mechanisms at these velocities cannot be inferred reliably from low-speed water-entry results alone.

This study establishes a truncated-cone object model at the meter scale to investigate the cross-medium process in the hydroballistic velocity regime. The Volume of Fluid (VOF) method is employed to resolve gas–liquid phase interactions and track the multiphase interface, while incorporating the Tait equation to account for water compressibility. The numerical simulations primarily focus on analyzing compressibility effects and evaluating the influence of key physical parameters, including entry velocity, truncated-cone object nose area, and half-cone angle, on the impact dynamics. Special emphasis is placed on characterizing the evolution of slamming loads during water entry, with the parametric analysis providing critical references for the design optimization of cross-media truncated-cone objects operating in the hydroballistic regime.

The remainder of this paper is organized as follows: Section 1 introduces the engineering background and reviews the current state of research in the relevant field. Section 2 describes the numerical model and computational methods employed in the simulations. Section 3 presents and analyzes the simulation results, with particular emphasis on the effects of fluid compressibility, water-entry conditions, and projectile geometry on the water-entry response. Finally, Section 4 summarizes the main findings and conclusions of this study.

2. Numerical Method and Verification

2.1. Mathematical Formulation

This study employs the Volume of Fluid (VOF) model to simulate multiphase flow. The VOF model assumes immiscibility between phases within the computational domain, introducing a respective volume fraction for each phase, and determines phase distribution by calculating these volume fractions. Taking the water volume fraction α_l and air volume fraction α_g as examples, they satisfy

$$\alpha_l + \alpha_g = 1$$

When $\alpha_l = 1$, it means that the inside of the grid is completely liquid phase; otherwise, it is filled with air. In the VOF model, phase interface tracking is achieved by solving continuity equations involving the volume fractions of the phases. The continuity equation for the mixture phase is

$$\frac{\partial \rho_m}{\partial t} + \frac{\partial (\rho_m v_i)}{\partial x_i} = 0$$

The momentum equation of the mixture phase is

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

In the equations, v_i denotes the velocity components; ρ_m , the density of the mixture phase, is given by $\rho_m = (1 - \alpha_g)\rho_l + \alpha_g\rho_g$; $\mu_m = (1 - \alpha_g)\mu_l + \alpha_g\mu_g$ represents the dynamic viscosity of the mixture phase; and $\mu_t = \frac{\rho_m C_\mu k^2}{\varepsilon}$ is the turbulent viscosity coefficient, where C_μ is determined by the turbulence model employed, k is the turbulent kinetic energy, and ε is the turbulent dissipation rate.

This study employs the Realizable k- ε turbulence model, with its transport equations as follows:

$$\frac{\partial}{\partial t} (\rho_m k) + \frac{\partial}{\partial x_i} (\rho_m k u_i) = \frac{\partial}{\partial x_i} \left[\left(\mu + \frac{\mu_t}{\sigma_k} \right) \frac{\partial k}{\partial x_i} \right] + G_k + G_b - \rho_m \varepsilon - Y_M$$

$$\frac{\partial}{\partial t}(\rho_m \epsilon) + \frac{\partial}{\partial x_i}(\rho_m \epsilon u_i) = \frac{\partial}{\partial x_i} \left[\left(\mu + \frac{\mu_t}{\sigma_\epsilon} \right) \frac{\partial \epsilon}{\partial x_i} \right] + \rho C_1 S \epsilon - \rho C_2 \frac{\epsilon^2}{k + \sqrt{\nu \epsilon}} + C_{1\epsilon} \frac{\epsilon}{k} C_{3\epsilon} G_b$$

In these equations,

$$C_1 = \max \left[0.43, \frac{\eta}{\eta + 5} \right], \quad \eta = S \frac{k}{\epsilon}, \quad S = \sqrt{2 S_{ij} S_{ij}}$$

G_k and G_b represent the turbulent kinetic energy production terms due to mean velocity gradients and buoyancy, respectively; Y_M denotes the contribution of fluctuating dilatation in compressible turbulence to the total dissipation rate. These terms are defined as

$$G_k = \mu_t S^2$$

$$G_b = \beta g_i \frac{\mu_t}{Pr_t} \frac{\partial T}{\partial x_i}$$

$$Y_M = 2 \rho_m \epsilon M_t^2$$

$Pr_t = 0.85$ is the turbulent Prandtl number for energy, g_i denotes the components of the gravity vector, β represents the thermal expansion coefficient, and M_t , the turbulent Mach number, is related to the speed of sound a and defined as

$$\beta = -\frac{1}{\rho_m} \left(\frac{\partial \rho_m}{\partial T} \right)_p$$

$$M_t = \sqrt{\frac{k}{a^2}}$$

For ideal gas, G_b is simplified as

$$G_b = -g_i \frac{\mu_t}{\rho_m Pr_t} \frac{\partial \rho_m}{\partial x_i}$$

$C_{1\epsilon}$, C_2 , σ_k , σ_ϵ are constants:

$$C_{1\epsilon} = 1.44, \quad C_2 = 1.9, \quad \sigma_k = 1.0, \quad \sigma_\epsilon = 1.2$$

$C_{3\epsilon}$ is defined as follows:

$$C_{3\epsilon} = \tanh \left| \frac{v}{u} \right|$$

In this paper, the state of air during the water entry process is described using the ideal gas equation, while the properties of the liquid are characterized using the Tait equation [28]:

$$p = a + b \rho_l^n$$

In the formula, the coefficients a and b are determined by assuming that the bulk modulus is a linear function of pressure.

The simplified form of the Tait equation is

$$\left(\frac{\rho_l}{\rho_{l0}} \right)^n = \frac{K}{K_0}, \quad K = K_0 + n \Delta p, \quad \Delta p = p - p_0$$

p_0 represents the reference pressure; K_0 is the bulk modulus at p_0 ; n is the density exponent; p is the liquid pressure (absolute pressure); ρ_l is the liquid density at pressure p ; K is the bulk modulus at pressure p .

2.2. Verification

To validate the applicability of the numerical model developed in this study for simulating the water-entry process of a body of revolution, the numerical results were compared with the experimental data reported in Ref. [39]. The water tank used in the experiment had dimensions of 210 mm × 260 mm × 320 mm. The wedge-shaped body had a semi-vertex angle of 90°, a width of 36 mm, and a length of 150 mm. Its initial water-entry velocity was 1.93 m/s. The geometric dimensions and initial conditions in the numerical model were consistent with those used in the experiment. The results are shown in Figure 1.

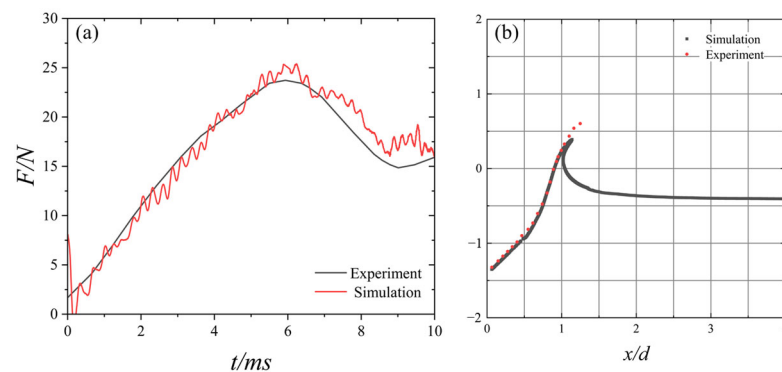


Figure 1. Comparison of force history (a) and cavity interface (b) between simulation and experiment.

The comparison focused primarily on the impact–load response during the water-entry process, together with the general evolution of the cavity and jet. The results show good agreement between the numerical and experimental data during the rising stage of the impact load, indicating that the model can reasonably capture the main characteristics of the initial impact process. A small difference is observed in the peak load and the subsequent relatively stable load value, although the overall variation trends are consistent. The numerical results also reproduce the basic development of the cavity and jet, although their local structural details are less refined than those observed experimentally.

It should be noted that this validation case is mainly intended to assess the capability of the numerical model to predict the impact–load response during the initial water-entry stage. The focus of this study is the formation and evolution of the impact load, rather than the detailed reconstruction of the local cavity structure. Therefore, a high level of accuracy in capturing the fine-scale features of the cavity and jet is not required. Overall, the model is capable of reproducing the primary physical characteristics of the water-entry impact and provides sufficient accuracy for the subsequent numerical simulations and analysis in this study.

2.3. Geometrical Model

As shown in Figure 2, this study employs a near-actual-scale truncated-cone object model for simulation, representing a geometry typical of supercavitating projectile nose sections and underwater ballistic penetrators. The computational domain for the numerical simulation is a 3D domain with dimensions of $100 D \times 100 D \times 100 L$, where L denotes the truncated-cone object length, and D represents its maximum diameter. This domain size ensures that boundary effects do not influence the water entry process during the simulation time. The entire computational domain is divided into three subdomains interconnected via mesh interfaces: Region 1 is the static region, while the truncated-cone model is positioned in Region 3, which moves synchronously along the $-Y$ direction with

Region 2 during the water entry process. The top boundary is specified as a pressure outlet, while all other boundaries are defined as no-slip walls.

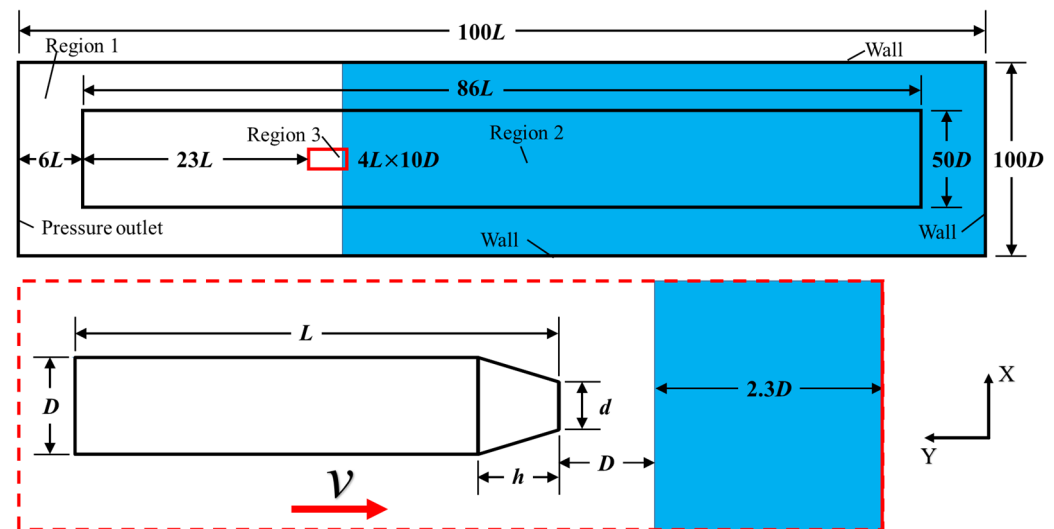


Figure 2. Truncated-cone object model and computational domain (Region 1 represents the static zone, while Region 2 and Region 3 are motion regions with truncated-cone object model in Region 3).

2.4. Grid Convergence and Verification

The geometric model in this case is relatively large. To minimize computational resource consumption, a symmetric computational model was adopted, and the computational domain was divided into three regions, each of which was meshed separately, as shown in Figure 3. To ensure accurate data transfer between adjacent regions, the mesh sizes on both sides of the interfaces were kept consistent, thereby avoiding additional numerical errors caused by mesh-size mismatches. Three meshes with different grid densities were used to discretize the computational domain: a coarse mesh with 3.98 million nodes, a medium mesh with 4.23 million nodes, and a fine mesh with 5.70 million nodes. The results of the grid independence verification are shown in Figure 4. A grid convergence study was then conducted using the Grid Convergence Index (GCI) methodology, which provides a quantitative assessment of discretization errors. The GCI values for the peak drag coefficient between the coarse and medium meshes and between the medium and fine meshes were 8.2% (GCI_{23}) and 3.1% (GCI_{12}), respectively, indicating that the medium mesh provides satisfactory grid independence for engineering applications. The apparent order of convergence was further calculated as $p = 1.8$, which is consistent with the second-order spatial discretization scheme employed by the solver.

To further validate the computational model, a quantitative comparison was performed against the experimental data reported by Bivin et al. [6] and the high-speed water entry experimental results of Zhao et al. [10]. The numerical predictions show good agreement with experimental measurements, with a maximum deviation of approximately 12% for the peak drag coefficient. This level of agreement is acceptable for hydroballistic water entry simulations, where experimental uncertainty is inherently higher than in low-speed conditions due to the extremely short impact duration and complex multiphase interactions.

Additionally, a time-step independence study was conducted to ensure that temporal discretization does not affect the results. Three time-step sizes were tested: $\Delta t_1 = 1.0 \times 10^{-6}$ s, $\Delta t_2 = 5.0 \times 10^{-7}$ s, and $\Delta t_3 = 2.5 \times 10^{-7}$ s. The peak overload coefficients computed with the three time steps agreed within 3%, confirming that the selected

time step (Δt_2) is adequate for resolving the unsteady compressible flow physics during water entry.

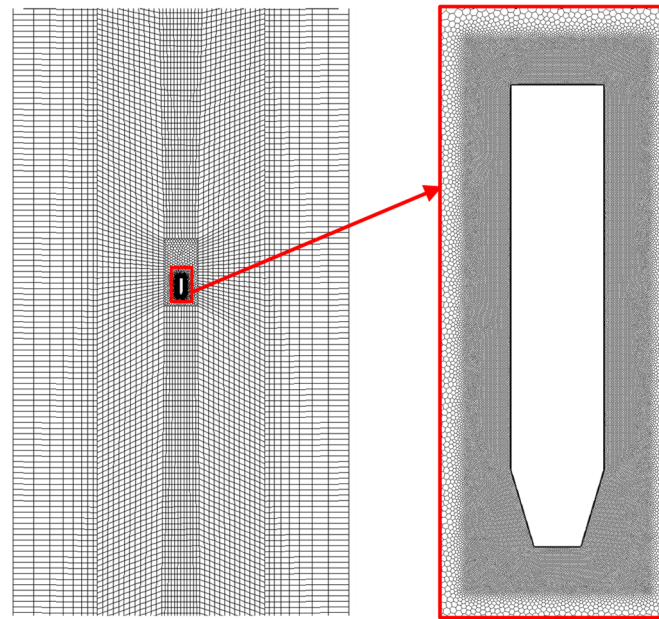


Figure 3. Meshing diagram of the computation domain. **(Left)** Global view illustrating the unstructured grid distribution surrounding the truncated cone, highlighting the refined mesh clustering at the water-entry interface to resolve flow gradients **(Right)**.

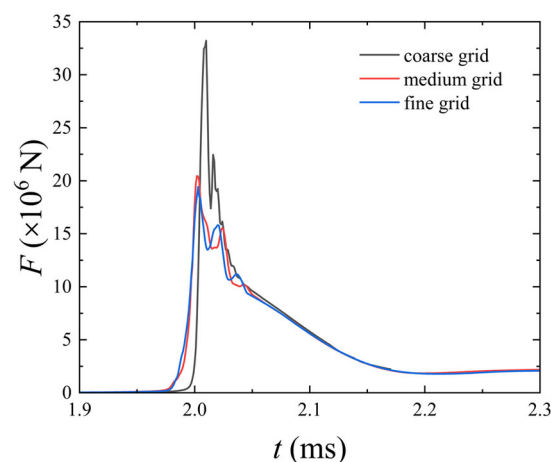


Figure 4. Grid independence verification.

In the water entry process, the truncated-cone object quickly reaches the stable sailing condition after impacting water. In the water entry process, the truncated-cone object quickly reaches the stable sailing condition after impacting water. Using Newton's second law, the truncated-cone object's overload (n) at the sailing condition can be calculated as follows:

$$n = \frac{\rho v^2 S}{2mg} + 1$$

The overload becomes stable after $t^* = 1.5$ ms, where $t^* = 0$ denotes the instant of the truncated-cone object's impact on the water surface. For a more detailed method for theoretical and numerical overload calculation, refer to our previous study [40,41]. The error between the average overload of the two results is within 0.2%, as shown in Figure 5, demonstrating the reliability of the computational model for predicting the hydrodynamic

loads during the water entry and subsequent sailing phases. This validation, combined with the experimental comparison presented in Section 2.3, provides confidence that the numerical methodology is capable of accurately simulating the hydroballistic water entry process.

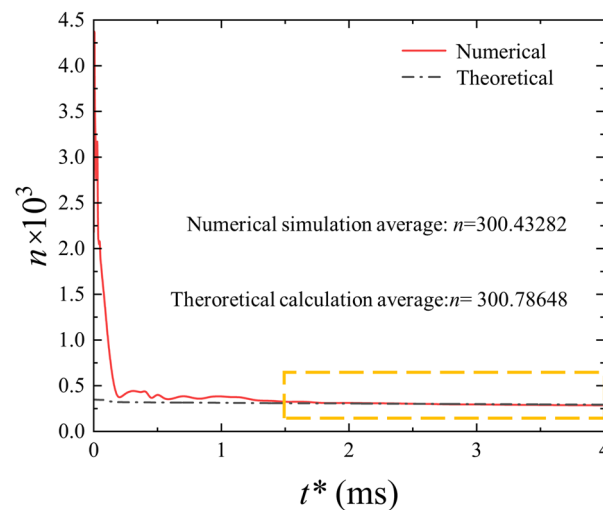


Figure 5. Comparison of overload (n) variation during the water entry.

3. Result and Discussion

3.1. Water Entry Process

Based on the grid convergence analysis presented in Section 2.3, the medium grid model is employed for large-scale parametric computations. In the initial state, the truncated-cone object model is positioned above the water surface at a distance D . During computation, it moves along the $-Y$ direction at $Fr = 174.87$ (Froude number, $Fr = \sqrt{\frac{v_0^2}{gd}}$), subsequently contacts the water surface, and submerges into the water (as shown in Figure 6).

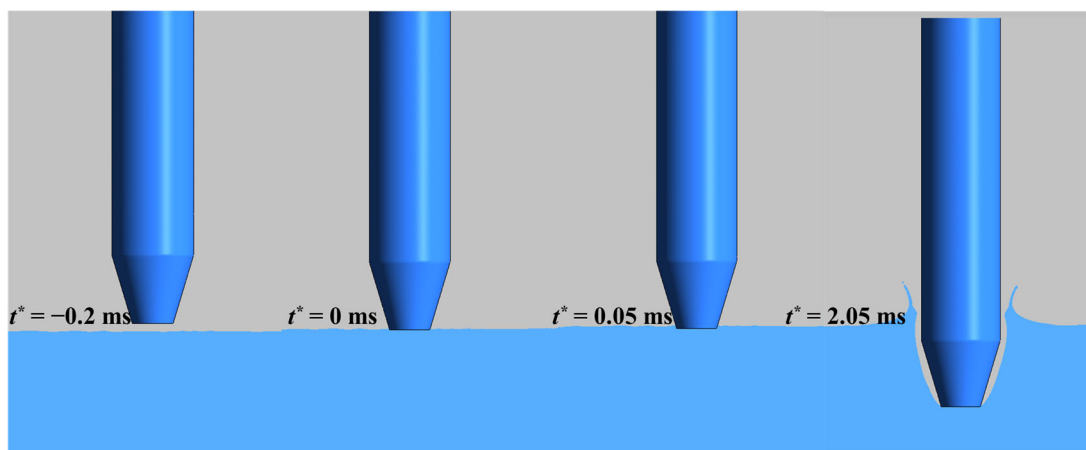


Figure 6. Water entry process of truncated-cone object model ($t^* = 0$ refers to the time of truncated-cone object impact on the water surface).

The water entry process of a truncated-cone object at hydroballistic velocities is characterized by distinct compressibility-dominated phenomena that differ fundamentally from low-speed entry scenarios. Upon initial contact with the free surface, the flat nose of the truncated-cone object generates a strong compression wave that propagates into the water

medium, creating a localized region of extremely high pressure at the impact interface. Simultaneously, the rapid displacement of the free surface produces an air cushion effect, wherein a thin layer of compressed air is trapped between the object's nose and the advancing water front. This air cushion significantly modifies the pressure distribution on the truncated-cone object's nose plane compared to incompressible assumptions, resulting in an uneven pressure distribution with peak values concentrated at the periphery of the flat head.

When the truncated-cone object slams onto the water surface, due to the large density gradient in the water-entry process, a large impact load will be generated. For the truncated-cone object, its nose, which is close to the flat plate slamming the water surface vertically, will make the maximum load peak pulse of the truncated-cone object more obvious. While the drag of the object moving in pure air medium and complete water medium is stable, as shown in Figure 7.

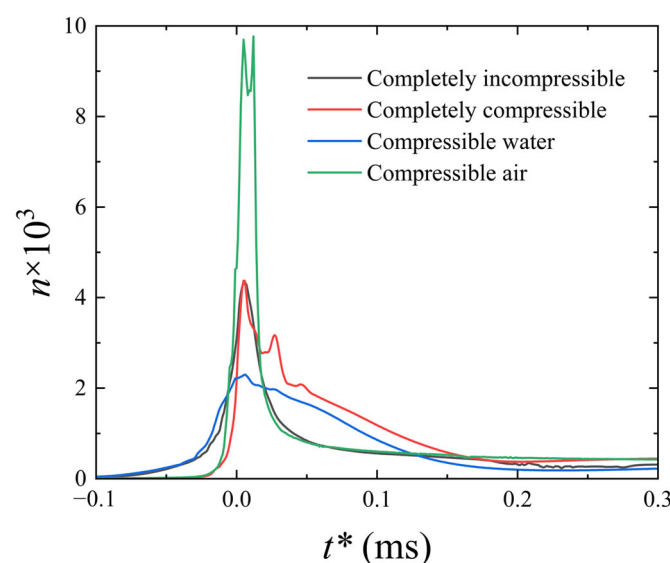


Figure 7. Overload evolving over time.

Figure 7 illustrates the evolution of the overload on the truncated-cone object model during water entry under different conditions. Considering the compressibility of water and air, the simulation is carried out under four conditions: compressible water, compressible air, completely compressible and completely incompressible. The results in Figure 7 demonstrate that, compared to the completely incompressible solution, air's compressibility elevates peak loads, while water's compressibility reduces peak load value, thereby inducing similar results in peak slamming loads between completely compressible and incompressible simulations. The compressibility of air induces a dual-peak characteristic in slamming loads; incompressible air causes premature load rise, and water compressibility extends load pulse duration.

To assess the influence of air's compressibility on computational results, density gradient contours of air are analyzed in Figure 8. The results reveal that during navigation in air, localized compression waves formed near the nose of the truncated-cone object. As the truncated-cone object continues to navigate along the $-Y$ direction, these compression waves interact with the water surface ahead of the truncated-cone object's impact, imparting velocity to portions of the water surface and establishing undulations. This causes asynchronous contact between the truncated-cone object's nose surface and the water, producing localized high-pressure zones. When the truncated-cone object passes through the undulating area and the water surface is fully contacted, the pressure distri-

bution transitions to a centrosymmetric pattern with higher pressure at the center and lower values at the periphery, as illustrated in Figure 9. The evolution characteristics of pressure distribution account for the dual-peak phenomenon (Figure 7) in slamming load magnitude during water entry.

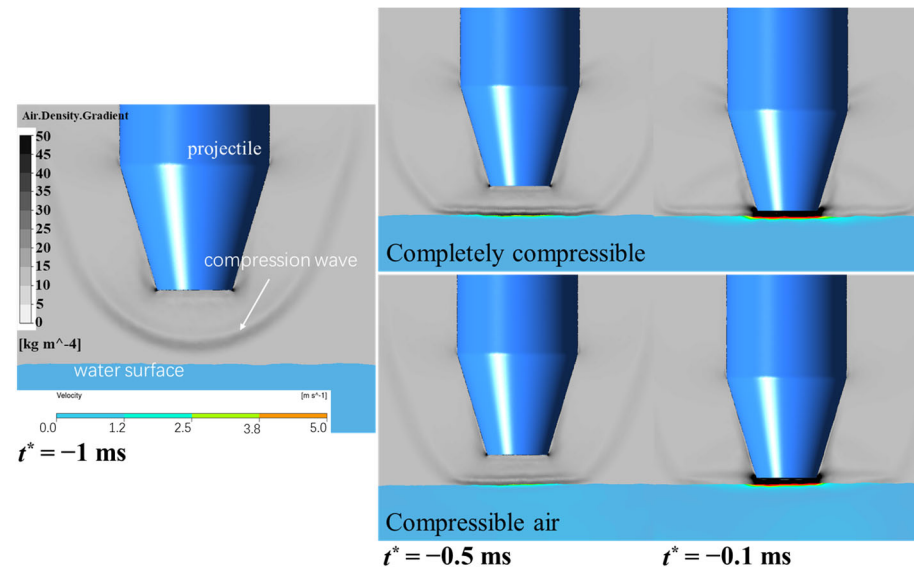


Figure 8. Compression wave impact the water surface.

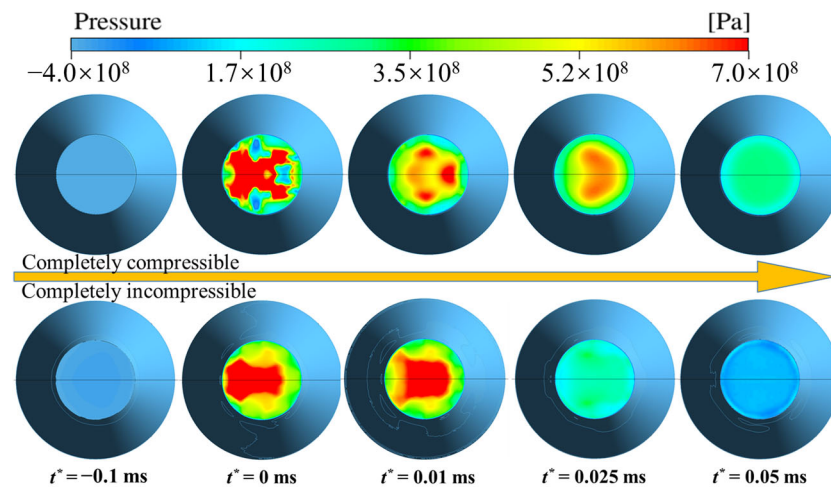


Figure 9. Pressure distribution on the head of the truncated-cone object during water entry process.

As shown in Figure 10, compared with the incompressible model, the compressible model exhibits a more complex surface-pressure evolution during the early stage of water entry. The pressure distribution gradually changes from irregular, localized high-pressure regions to a more centralized pattern, while the high-pressure state persists for a longer duration. This behavior is mainly attributed to compression waves generated in the air layer between the structure and the free surface. Their propagation disturbs the free surface, causing the structure to initially contact the water locally. As the entry progresses, the contact area expands and gradually develops into a nearly full-contact state, resulting in a more continuous and centralized pressure distribution. In contrast, the incompressible model neglects these air-compression-wave effects, leading to a more regular initial contact process and a faster evolution of the localized high-pressure regions.

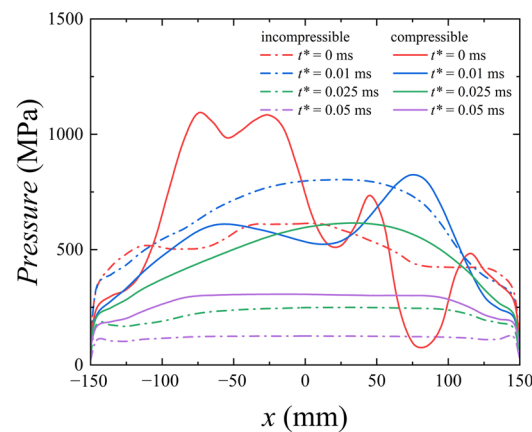


Figure 10. Pressure distribution on the head of the truncated-cone object.

The compressibility of air induces a cushioning effect, where the compression wave of air precedes the truncated-cone object's contact with the water surface, thereby generating initial disturbances. In contrast, for incompressible air, the air ahead of the truncated-cone object cannot be compressed before water impact and instead impacts the water surface in a quasi-rigid manner. This leads to an earlier pressure buildup at the truncated-cone object nose prior to water entry, accompanied by greater thrust on the water surface that generates higher initial velocity (Figure 11). Consequently, the peak load during subsequent water entry exhibits a corresponding reduction.

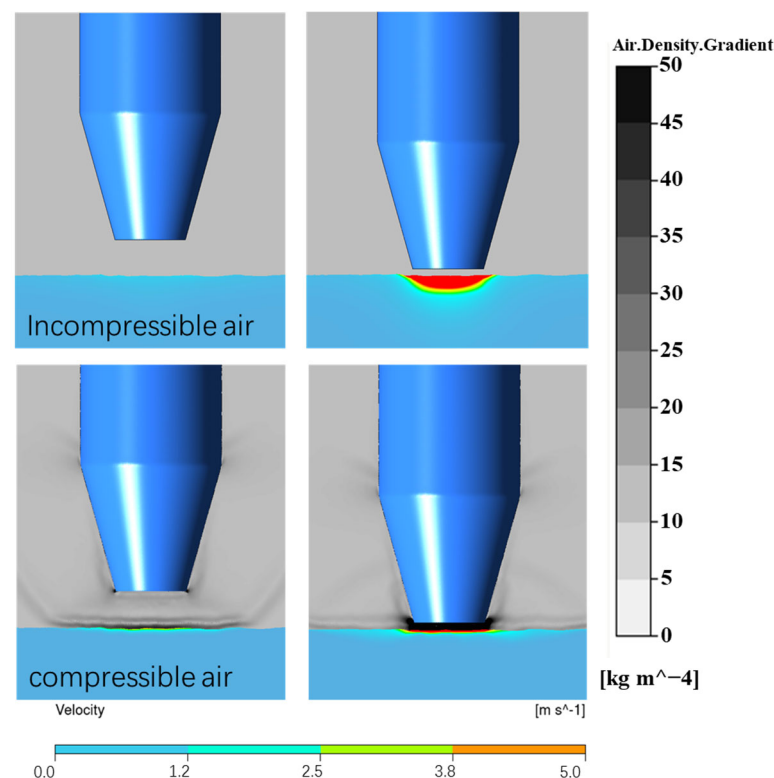


Figure 11. The difference in water surface impact between compressible and incompressible air.

After entering the water, the compressibility of the water body is mainly reflected in the generation of underwater compression waves. For incompressible water, the impact generated by the water impact process is effective for the entire water body at the same time. The pressure distribution of the water body is almost unchanged throughout the process, similar to rigid body collision (Figure 12), so its peak value is higher and the pulse

time is shorter. For compressible water, due to the generation of a compression wave, the high-pressure area in the process of water impact gradually transfers downward with the compression wave, and the high-pressure area is distributed in the front of the compression wave (Figure 13). At this time, the impact energy release is slower, the load peak is low, and the pulse time is longer.

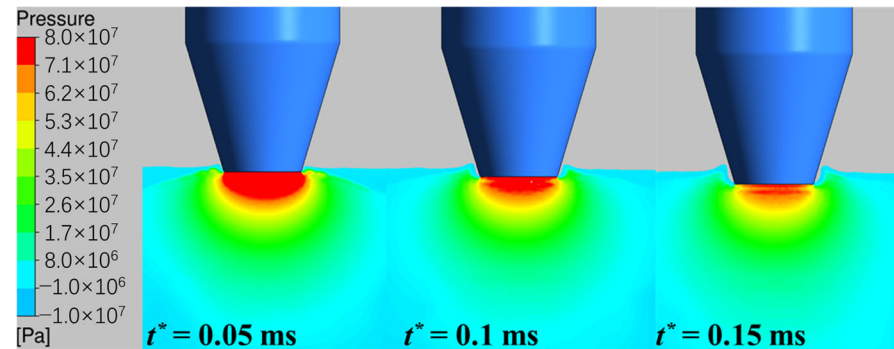


Figure 12. Pressure distribution in incompressible water.

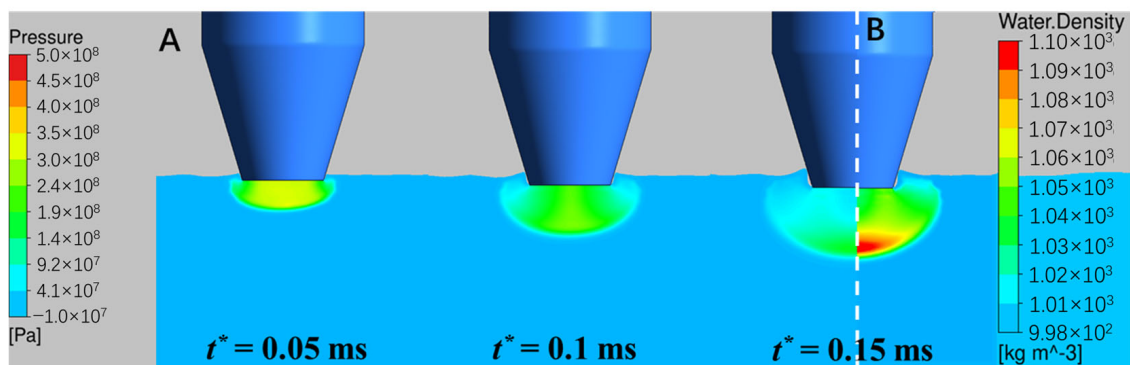


Figure 13. Pressure distribution in compressible water ((A) pressure contour; (B) density contour).

These findings are physically consistent with the operational conditions of supercavitating projectiles, where the initial impact phase generates the highest structural loads that the vehicle must withstand. The air cushion phenomenon observed in the simulations has direct implications for the nose design of such vehicles: an optimal nose geometry must balance the need for rapid cavity generation (favored by flat or nearly flat nose profiles) against the mitigation of peak impact loads (which may be achieved through carefully designed nose modifications). The results presented in this section provide fundamental physical insights that can guide the design of nose sections for supercavitating torpedoes and hydroballistic penetrators.

3.2. The Influencing Factors of Water-Entry Peak Load

Given the high-magnitude and short-duration characteristics of slamming loads for truncated-cone objects during water entry, this study focuses on the peak load magnitude, analyzing the influence of key parameters on its magnitude.

3.2.1. Influence of Impact Velocity

The geometric model and mesh were maintained constant. The simulations were performed for varying water entry velocities to investigate their influence on the peak slamming overload while the geometric model and other parameters were maintained constant. It should be noted that the highest velocity considered in this study (1100 m/s) is

chosen to investigate the strongly compressible, shock-dominated impact physics rather than to represent the operating condition of any specific existing vehicle. The results are shown in Figure 14.

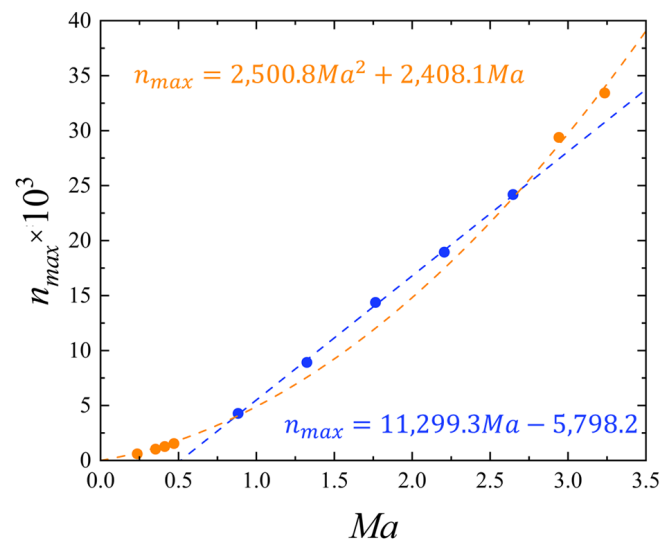


Figure 14. Relationship between peak overload and Mach number of truncated-cone objects.

As shown in Figure 14, the peak overload coefficient exhibits distinct trends with the truncated-cone object's Mach number: when the water entry Mach number of the truncated-cone object is between 0.88 and 2.64, the peak value of the water entry overload and the Mach number of the truncated-cone object show an obvious linear relationship, which satisfies the compression wave dominant mode in the previous article. This aligns with the impact peak pressure estimation formula [29]:

$$p_{\max} = \rho_l c_l c_g Ma$$

At low velocities, the compression of the water medium is insufficient to generate strong compression waves. Consequently, the additional mass of the displaced water corresponds to the top water layer propelled by the truncated-cone object's motion, resulting in a quadratic relationship between peak overload and velocity.

Figure 15 illustrates the compression wave states at the truncated-cone object nose under identical immersion depths ($\Delta h = 10$ mm) during water entry at varying velocities. Higher velocities produce more pronounced compression waves, while they are virtually absent at lower velocities. It should be noted that different truncated-cone object velocities result in varying times to reach the same immersion depth. At higher velocities, the time available for compression wave propagation is shorter, leading to a smaller propagation distance of the compression wave (relative to the truncated-cone object nose position), despite the wave propagation speed being similar across different cases.

Figure 16 illustrates the compression wave states at the truncated-cone object nose after equal water entry durations ($t^* = 0.01$ ms). At low and medium velocities, compression wave propagation rates remain consistent, while at high velocities, wave expansion accelerates significantly. This indicates that the truncated-cone object's high velocity actively drives compression wave propagation, linking the additional displaced water mass to the entry velocity and resulting in a quadratic relationship between peak load and velocity as shown in Figure 14.

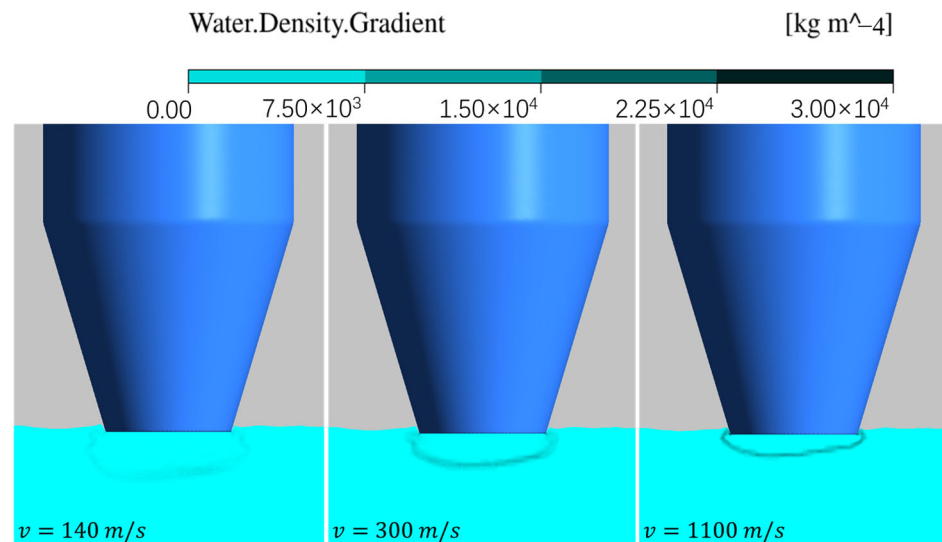


Figure 15. Water density gradient contours during entry at different velocities ($\Delta h = 10$ mm).

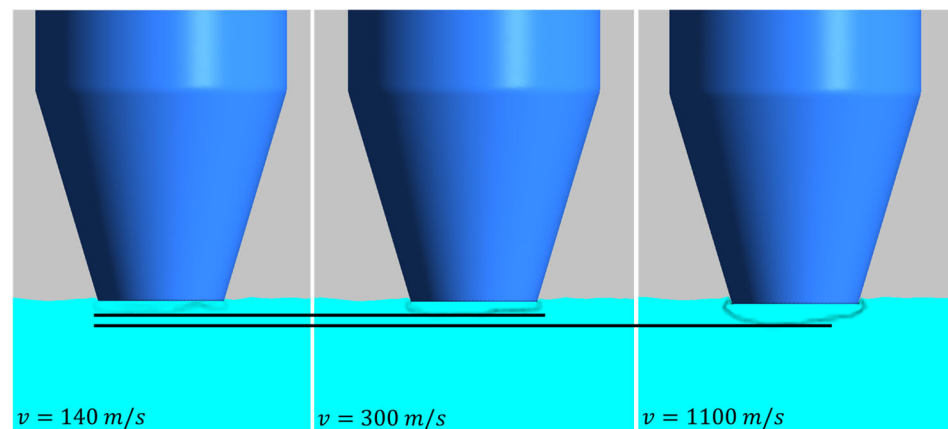


Figure 16. Compression wave states at the truncated-cone object head after identical water entry durations ($t^* = 0.01$ ms).

In the velocity range of 300–900 m/s (Mach 0.20–0.60 in water), the peak slamming overload during water entry is governed by underwater compression waves generated by the object's interaction with the water medium. Within this regime, the peak overload coefficient exhibits an approximately linear relationship with entry velocity. This linear dependence arises because the intensity of the compression wave scales directly with the impact velocity, while the duration of the pressure pulse remains relatively constant. The physical mechanism is consistent with the compressible water entry behavior observed by Wu et al. [31] for axisymmetric bodies and Yang et al. [32] for rectangular projectiles at comparable velocity ranges.

At velocities exceeding 900 m/s (Mach > 0.60), the rate of increase in peak overload with velocity diminishes. This saturation effect is attributed to the increasing dominance of the air cushion phenomenon at higher impact velocities: as the velocity increases, the trapped air layer beneath the object's nose becomes more compressed, effectively distributing the impact load over a larger area and reducing the peak pressure. Additionally, at these elevated velocities, vaporization effects at the nose surface begin to contribute to load alleviation through partial cushioning by the vapor phase.

The velocity sensitivity identified in this study has direct engineering implications for supercavitating vehicle design. The operational velocity of the Shkval torpedo (~370 m/s, Mach ~0.25 in water) falls squarely within the linear regime, where small variations in entry

velocity produce proportional changes in peak load. This suggests that design optimization efforts for such vehicles should prioritize velocity management during the water-entry phase, as even modest reductions in impact velocity can yield significant reductions in peak structural loads. For next-generation hydroballistic vehicles targeting velocities in the 600–900 m/s range, the design must account for the increasingly complex interplay between compression waves and air cushion effects.

3.2.2. Effect of Truncated-Cone Object Nose Area

For a truncated-cone object during water entry, its flat head induces a water impact behavior analogous to flat-plate entry, making the head area a critical factor influencing the peak load. To isolate this effect while controlling variables, simulations maintained a constant semi-cone angle, varying only the truncated-cone height h to change the head area. The water entry velocity was fixed at 300 m/s.

Figure 17 demonstrates the variation in peak slamming load for a truncated-cone object with different nose radii. Accounting for air-induced perturbations that create uneven load distribution across the nose plane (shown in Figure 9), the results exhibit an approximate quadratic dependence on the nose radius. Specifically, the peak overload shows a linear relationship with the nose planar area. It should be noted that this analysis considers only the effect of the nose radius R (with other parameters fixed), rather than a combined variation in both R and velocity v . A more rigorous approach would involve dimensional analysis using dimensionless parameters such as v/c (velocity normalized by the speed of sound in water) and R/h (radius-to-height ratio) to establish a unified fitting formula.

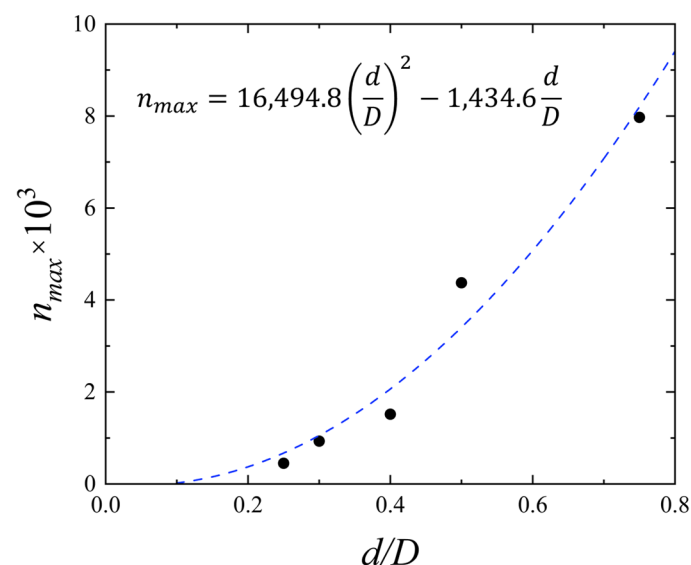


Figure 17. Variation in peak overload with d/D .

Figure 18 illustrates the compression waves generated by water states at the head of truncated-cone objects during high-speed water entry, analyzed under the condition of water compressibility. The scale at the top represents the water density gradient in $\text{kg} \cdot \text{m}^{-4}$, visually quantifying the intensity of the pressure disturbance. The simulation demonstrates that for a given high-speed entry, a sharper truncated cone ($d/D = 0.25$) generates a more concentrated and intense compression wave, whereas a blunter geometry ($d/D = 0.75$) produces a broader, less focused pressure disturbance. This illustrates how macroscopic geometric parameters directly govern the localized hydrodynamic loading and energy transfer characteristics during unsteady water-entry scenarios.

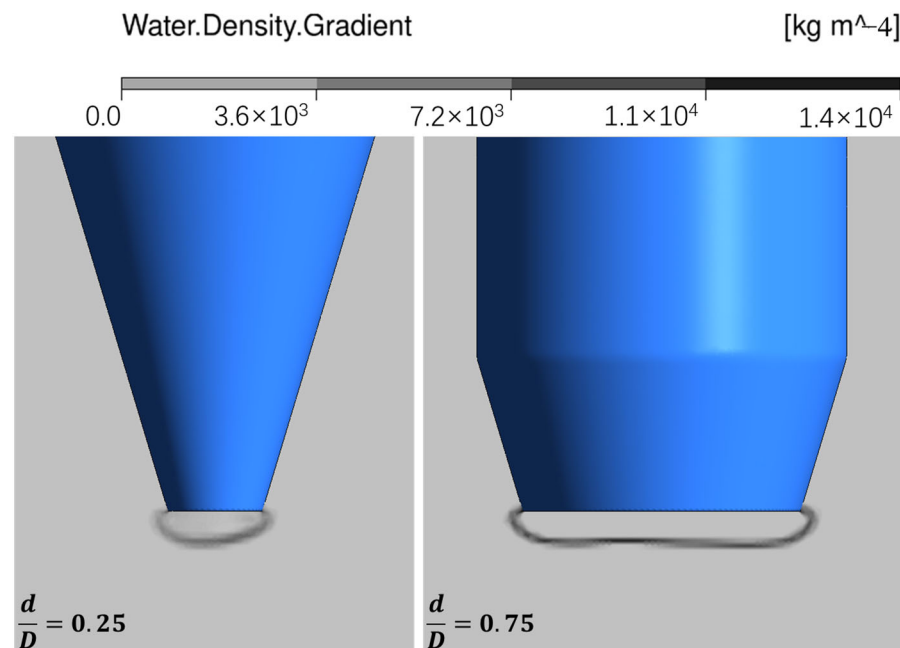


Figure 18. Compression wave of water states at the truncated-cone object head at 0.01 ms.

The parametric study reveals that the peak overload exhibits a quadratic dependence on the nose radius (or equivalently, on the planar head area). This quadratic relationship arises from two contributing factors: (1) the increased frontal area directly intercepts a larger volume of water during impact, generating stronger compression waves; and (2) the larger diameter enhances the air cushion effect by trapping a greater mass of air beneath the nose, which subsequently compresses and modifies the pressure distribution. This finding is particularly relevant for the design of supercavitating projectile nose sections, where the head diameter is a primary design parameter that must be optimized to achieve reliable cavity generation while maintaining acceptable impact loads. For typical supercavitating vehicles with head diameters in the range of 0.2–0.5 m, the results indicate that a 20% reduction in nose radius can yield approximately a 36% reduction in peak overload, providing valuable quantitative guidance for structural optimization.

3.2.3. Effect of Half-Cone Angle

The half-cone angle of the truncated-cone object β represents another important geometric parameter that influences the water entry dynamics. In the present study, the semi-vertex angle, β , was defined according to $\tan \beta = (D - d)/(2h)$. To ensure that the frontal area remained unchanged, the parameters d and h were kept constant, while only D was varied to control the value of β . The effects of varying $\tan \beta$ within the range of 0.3–0.9 were investigated at 300 m/s.

The results presented in Figure 19 indicate that the half-cone angle has no pronounced effect on the water-entry load. Combined with the density-gradient contours in Figure 20, this suggests that its influence is mainly associated with the shape of the air-compression wave and its disturbance of the free surface. As the half-cone angle varies, the propagation characteristics and spatial distribution of the air-compression wave near the projectile nose change accordingly. In particular, when $\tan \beta = 0.6$, the air-compression wave is located relatively close to the projectile nose and acts on the free surface for a shorter duration. Consequently, the disturbance to the free surface is weakened, resulting in relatively smaller surface deformations and fluctuations. Under these conditions, the local contact interaction between the projectile and the free surface is reduced, and the double-peak feature in the load-time history disappears, giving way to fluctuations in the vicinity of the peak load.

These results indicate that the half-cone angle affects the local impact load during the initial water-entry stage indirectly by modifying the interaction between the air-compression wave and the free surface. Following the impact stage, the load generally increases with increasing half-cone angle β . This trend is mainly attributed to the increase in D . As β increases, D also increases, thereby enlarging the contact area between the projectile and the water and consequently increasing the post-impact load level.

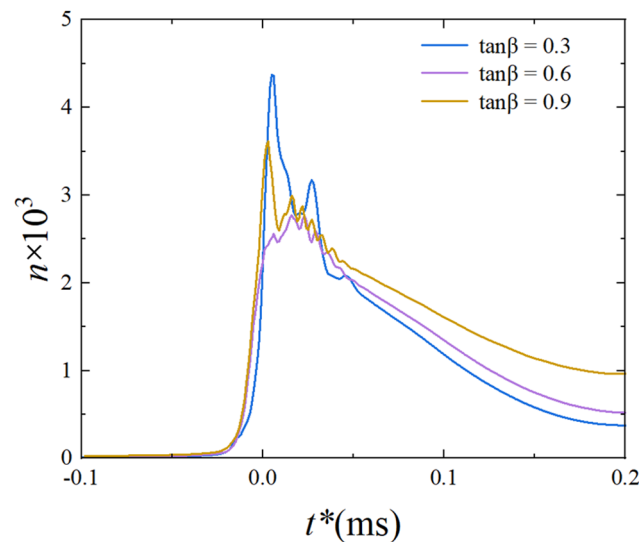


Figure 19. Overload evolving over time with varying half-cone angle.

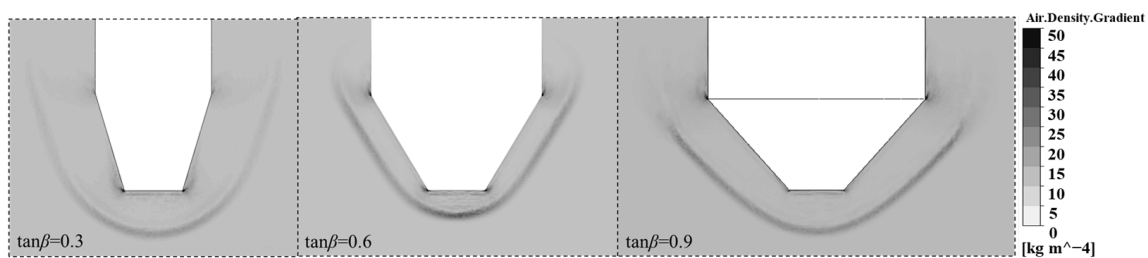


Figure 20. Air compression waves at different half-cone angles.

3.2.4. Effect of Impact Angle

The load characteristics of the projectile under oblique water-entry conditions were investigated. Figure 21 illustrates the temporal evolution of the free surface during water entry at an entry angle of 70° . As shown in Figure 22, the time–history curves indicate that, as the entry angle deviates further from the vertical direction, the effective axial projected area of the flat nose decreases, while the immersion velocity is reduced. Consequently, the axial overload decreases significantly, and its duration is prolonged. Meanwhile, an increase in the deviation of the entry angle from the vertical direction leads to a higher radial overload, which may adversely affect the attitude and trajectory stability of the projectile. Therefore, the effects of reduced axial overload, prolonged loading duration, and increased radial overload should be considered comprehensively in the design and guidance-control of projectiles undergoing oblique water entry.

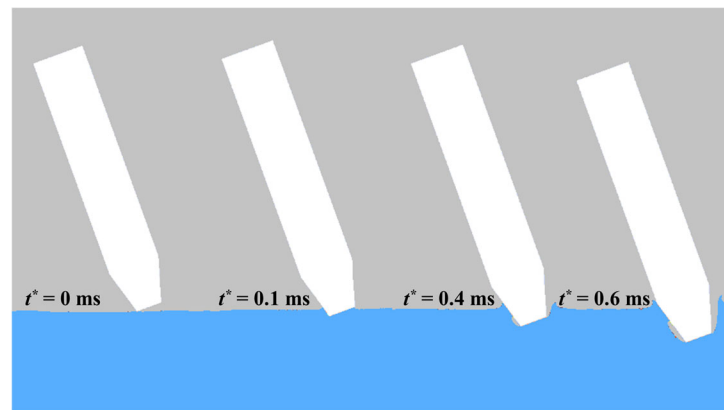


Figure 21. Water entry process of a truncated-cone object at an impact angle of 70° .

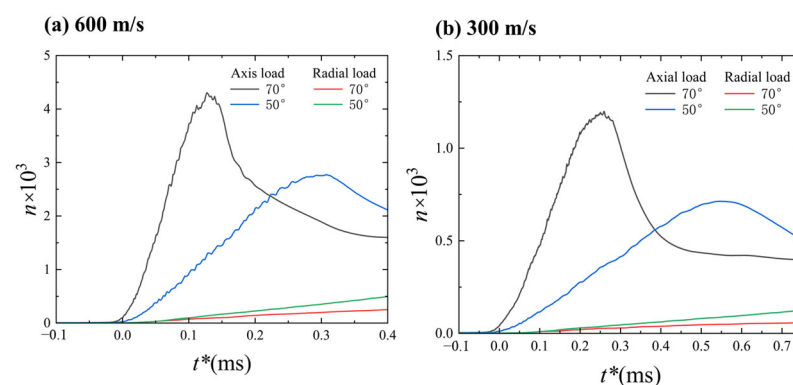


Figure 22. Time histories of axial and radial overloads.

4. Conclusions

This study employs a VOF multiphase flow method incorporating fluid compressibility to simulate the load characteristics during high-speed vertical and oblique water entry of a truncated-cone object, representing a geometry relevant to supercavitating projectiles and hydroballistic vehicles. Based on the grid-converged and validated model, large-sample parametric computations were conducted to systematically analyze the effects of entry velocity, impact angle, and structural dimensions on peak slamming overload. The following conclusions are drawn:

(1) The water entry of a truncated-cone object at hydroballistic velocities (300–1100 m/s) is governed by compressibility-dominated phenomena. Air shockwaves striking the water surface generate a pronounced air cushion effect, causing uneven pressure distribution across the truncated-cone object's nose plane during water impact. Underwater compression waves generated by the object's interaction with the compressible water medium dominate the load formation process, producing high-peak, short-pulse load characteristics that are qualitatively distinct from low-speed entry scenarios.

(2) Parametric studies demonstrate that entry velocity is the most sensitive factor affecting peak overload. Within the range of 300–900 m/s (Mach 0.20–0.60 in water), the peak load exhibits an approximately linear relationship with velocity, which is consistent with the operational regime of existing supercavitating vehicles such as the Shkval torpedo. At velocities exceeding 900 m/s, the rate of load increase diminishes due to the increasing dominance of air cushion effects and vaporization phenomena.

(3) The nose radius (or planar head area) is the second most influential factor, with peak load showing a quadratic dependence on nose radius. This finding provides quantitative guidance for the nose design of supercavitating projectiles, where a 20% reduction in nose

radius can yield approximately a 36% reduction in peak overload. The half-cone angle has a moderate effect on peak load but primarily influences post-impact cavity evolution and stability.

(4) Oblique water entry reduces the peak overload due to the decreased effective projected area but introduces asymmetric loading that generates pitching moments. This trade-off between load reduction and moment generation must be considered in the guidance and control system design for trans-medium vehicles.

These findings provide quantitative guidance for the protective design and structural optimization of high-speed water-entry structures operating in the hydroballistic regime, including supercavitating torpedoes, submarine-launched projectiles, and next-generation trans-medium vehicles. Future work should extend the current analysis to include structural elasticity effects and three-dimensional trajectory dynamics for complete vehicle configurations.

In the present study, structural deformation and cavitation effects are not considered. This simplification is justified by our focus on the instantaneous impact upon water entry, during which the air entrained by the penetrating object dominates the loading process. As the object submerges further, its shoulder region becomes fully immersed and moves away from the free surface; at that stage, cavitation dynamics are expected to play a progressively more significant role. Accordingly, the current work temporarily neglects both cavitation and structural deformation, leaving these aspects to be addressed in future investigations.

Author Contributions: Conceptualization, P.L. and Y.X.; methodology, M.Z. and Y.S.; software, M.Z.; validation, M.Z., J.H. and Y.X.; formal analysis, M.Z. and Y.S.; investigation, M.Z.; resources, Y.X.; data curation, M.Z. and Z.Y.; writing—original draft preparation, M.Z. and Z.Y.; writing—review and editing, M.Z., Z.Y., P.L. and Y.X.; visualization, M.Z. and J.H.; supervision, Y.X.; project administration, P.L., Y.S. and Y.X.; funding acquisition, P.L. All authors have read and agreed to the published version of the manuscript.

Funding: This work was supported by the President Fund of the China Academy of Engineering Physics (Grant No. YZJJZQ2023011) and the Young Scientists Fund of the National Natural Science Foundation of China (Grant No. 12102412).

Data Availability Statement: The data that support the findings of this study are available from the corresponding author upon reasonable request.

Conflicts of Interest: The authors declare no conflicts of interest. The funders had no role in the design of the study; in the collection, analysis, or interpretation of data; in the writing of the manuscript; or in the decision to publish the results.

References

1. Feng, J.; Hu, J.; Qi, D. Study on development needs and key technologies of air-water trans-media vehicle. *J. Air Force Eng. Univ.* **2019**, *20*, 8–13.
2. Von Karman, T. *The Impact on Seaplane Floats During Landing*; National Advisory Committee for Aeronautics Technical: Washington, DC, USA, 1929.
3. Wagner, H. Phenomena associated with impacts and sliding on liquid surfaces. *J. Appl. Math. Mech.* **1932**, *12*, 193–215.
4. Logvinovich, G.V. *Hydrodynamics of Free-Boundary Flows*; Israel Program for Scientific Translations: Jerusalem, Israel, 1972.
5. Worthington, A.M.; Cole, R.S.V. Impact with a liquid surface, studied by the aid of instantaneous photography. *Philos. Trans. R. Soc. London. Ser. A* **1897**, *189*, 137–148. [[CrossRef](#)]
6. Bivin, Y.K.; Glukhov, Y.M.; Permyakov, Y.V. Vertical entry of solids into water. *Fluid Dyn.* **1985**, *20*, 835–841. [[CrossRef](#)]
7. Lee, M.; Longoria, R.G.; Wilson, D.E. Cavity dynamics in high-speed water entry. *Phys. Fluids* **1997**, *9*, 540–550. [[CrossRef](#)]
8. Lee, M.; Longoria, R.G.; Wilson, D.E. Ballistic waves in high-speed water entry. *J. Fluids Struct.* **1997**, *11*, 819–844. [[CrossRef](#)]
9. 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. [[CrossRef](#)]

10. Zhao, G.; Guo, Z.; Chen, T.; Zhang, W. Experimental investigation on high-speed vertical water entry of spheres with different densities and velocities. *Ocean Eng.* **2024**, *295*, 116775. [\[CrossRef\]](#)
11. Stenius, I.; Rosén, A.; Kutteneuler, J. Hydroelastic interaction in panel-water impacts of high-speed craft. *Ocean Eng.* **2011**, *38*, 371–381. [\[CrossRef\]](#)
12. Panciroli, R.; Abrate, S.; Minak, G.; Zucchelli, A. Hydroelasticity in water-entry problems: Comparison between experimental and SPH results. *Compos. Struct.* **2012**, *94*, 532–539. [\[CrossRef\]](#)
13. Yang, J.; Feng, J.; Li, Y.; Liu, A.; Hu, J.; Ma, Z. Water-exit process modeling and added-mass calculation of the submarine-launched projectile. *Pol. Marit. Res.* **2017**, *24*, 152–164. [\[CrossRef\]](#)
14. Yang, L.; Wei, Y.; Wang, C.; Xia, G. Viscoelasticity dependence on hydrodynamic responses during water entry. *Ocean Eng.* **2023**, *272*, 113890. [\[CrossRef\]](#)
15. Zhang, S.; Fan, Y.; Zhang, C.; Adams, N.A.; Hu, X. A diffusive wetting model for water entry/exit based on the weakly-compressible SPH method. *Ocean Eng.* **2025**, *325*, 120818. [\[CrossRef\]](#)
16. Hou, Y.; Huang, Z.; Chen, Z.; Guo, Z.; Xu, Y. Experimental investigations on the oblique water entry of hollow cylinders. *Ocean Eng.* **2022**, *266*, 112800. [\[CrossRef\]](#)
17. Yuan, H.; Zhou, X.; Ling, H.; Sun, Q.; Yang, C. Numerical simulation of cavity evolution and motion characteristics of double hollow cylinders in oblique water entry. *Ocean Eng.* **2025**, *324*, 120695. [\[CrossRef\]](#)
18. Qi, X.; Peng, H.; Liu, F.; Xie, H.; Li, Y. Analysis of load and free surface behaviors of a hull-shaped body during asymmetrical water entry. *Ocean Eng.* **2024**, *310*, 118556. [\[CrossRef\]](#)
19. Li, M.Y.; Cui, J.; Chen, X.; Gu, C.J.; Masvaya, B. Kinematic response analysis of a three-dimensional lifeboat water entry based on smoothed particle hydrodynamics. *Ocean Eng.* **2024**, *295*, 116904. [\[CrossRef\]](#)
20. Gan, W.; Zhuang, J.; Zhang, Y.; Zuo, Z.; Xiang, J. Numerical investigation of the water-entry impact performance of a bionic unmanned aerial-underwater vehicle. *Ocean Eng.* **2024**, *303*, 117458. [\[CrossRef\]](#)
21. Wang, X.; Zhao, J.; Tan, X.; Qi, C.; Zhao, A.; Li, H.; Sun, R.; Lyu, X. Numerical study of consecutive water entries in flowing water with twin spheres side-by-side. *J. Fluids Struct.* **2024**, *131*, 104218. [\[CrossRef\]](#)
22. Wang, Y.; Wei, Y.; Wang, C.; Wen, Y. Experimental study on asynchronous parallel high-speed oblique water entry of cylinders. *Appl. Ocean Res.* **2024**, *146*, 103945. [\[CrossRef\]](#)
23. Wang, Y.; Wei, Y.; Wang, C. Study on cavity evolution of asynchronous parallel high-speed vertical water entry of cylinders. *Int. J. Multiph. Flow* **2025**, *186*, 105164. [\[CrossRef\]](#)
24. Lu, L.; Cheng, Y.; Chen, K.; Yu, Z.; Yang, Z. Experimental investigation on the water-entry characteristics of the structure passing through the broken ice field. *Ocean Eng.* **2024**, *313*, 119348. [\[CrossRef\]](#)
25. Cui, W.; Kong, D.; Sun, T.; Yan, G. Coupling dynamic characteristics of high-speed water-entry projectile and ice sheet. *Ocean Eng.* **2023**, *275*, 114090. [\[CrossRef\]](#)
26. Hu, X.; Wei, Y.; Wang, C. Study on water entry impact characteristics and parameter influence analysis of the vehicle passing through the thin crushed ice zone in a polar environment. *Appl. Ocean Res.* **2024**, *153*, 104250. [\[CrossRef\]](#)
27. Hu, X.; Wei, Y.; Wang, C. Dynamic analysis in polar exploration: Fluid-structure interaction modeling of projectile colliding with floating ice during water entry. *Mar. Struct.* **2025**, *100*, 103729. [\[CrossRef\]](#)
28. Cole, R.H.; Weller, R. Underwater explosions. *Phys. Today* **1948**, *1*, 35. [\[CrossRef\]](#)
29. Xu, X.Z.; Ouyang, L.V.; Yan, Z.H. *Torpedo Mechanics*; National Defense Industry Press: Beijing, China, 1992.
30. Yang, D.; Xiong, Y.L.; Guo, X.F. Drag reduction of a rapid vehicle in supercavitating flow. *Int. J. Nav. Archit. Ocean Eng.* **2017**, *9*, 35–44. [\[CrossRef\]](#)
31. Wu, Q.; Wang, K.; Kong, D.; Zhang, J.; Liu, T. Numerical simulation of subsonic and transonic water entry with compressibility effect considered. *Ocean Eng.* **2023**, *281*, 114984. [\[CrossRef\]](#)
32. Yang, L.; Xiang, J.; Zhang, S.; Kang, H.; Wang, X.; Xia, W. Compressibility effects on cavity dynamics and shock waves in high-speed water entry. *Phys. Fluids* **2024**, *36*, 042104. [\[CrossRef\]](#)
33. Egorov, L.T. Impact on a compressible fluid. *Prikladnaia Mat. Mekhanika* **1958**, *20*, 67–72.
34. Eroshin, V.A.; Romanenkov, N.I.; Serebriakov, I.V.; Iakimov, I.L. Hydrodynamic forces produced when blunt bodies strike the surface of a compressible fluid. *Akad. Nauk Sssr Izv. Mekhanika Zhidkosti Gaza* **1980**, *15*, 44–51. [\[CrossRef\]](#)
35. Korobkin, A.A.; Peregrine, D.H. The energy distribution resulting from an impact on a floating body. *J. Fluid Mech.* **2000**, *417*, 157–181. [\[CrossRef\]](#)
36. Oger, G.; Doring, M.; Alessandrini, B.; Ferrant, P. Two-dimensional SPH simulations of wedge water entries. *J. Comput. Phys.* **2006**, *213*, 803–822. [\[CrossRef\]](#)
37. Chen, C.; Sun, T.; Wei, Y.; Wang, C. Computational analysis of compressibility effects on cavity dynamics in high-speed water-entry. *Int. J. Nav. Archit. Ocean. Eng.* **2019**, *11*, 495–509. [\[CrossRef\]](#)
38. Wang, S.; Soares, C.G. Effects of compressibility, three-dimensionality and air cavity on a free-falling wedge cylinder. *Ocean Eng.* **2020**, *217*, 107589. [\[CrossRef\]](#)

39. Vincent, L.; Xiao, T.; Yohann, D.; Jung, S.; Kanso, E. Dynamics of water entry. *J. Fluid Mech.* **2018**, *846*, 508–535. [[CrossRef](#)]
40. Wang, Y.; Xiong, Y.; Tian, X.; Zhou, F.; Liu, A.; Sun, G. Comparison of High-Speed Water Entry Movement Process of Axisymmetric Bodies with Different Head Shapes. *J. Unmanned Undersea Syst.* **2024**, *32*, 451–462.
41. Liu, P.; Huang, J.; Wang, X. Rapid Prediction and Numerical Verification of High-Speed Water-Entry Impact Response of Spherical-Nosed Cones. *J. Unmanned Undersea Syst.* **2026**, *34*, 109–119.

Disclaimer/Publisher’s Note: The statements, opinions and data contained in all publications are solely those of the individual author(s) and contributor(s) and not of MDPI and/or the editor(s). MDPI and/or the editor(s) disclaim responsibility for any injury to people or property resulting from any ideas, methods, instructions or products referred to in the content.