

Numerical Study on the Water Entry Process of Vehicle Based on the Structured Arbitrary Lagrangian-Eulerian Method

Zhenpeng Liu*, Yao Shi*, Qiaogao Huang*, Guang Pan*

Corresponding author: shiyao@nwpu.edu.cn

* Northwestern Polytechnical University, China.

Abstract: Water entry is a critical technical issue in nature, daily life, and engineering fields. In this study, a numerical model for the water entry of large-scale vehicle (with a diameter of 533 mm) was established based on the structured arbitrary Lagrangian-Eulerian (S-ALE) method. The cavity evolution and time-domain characteristics of acceleration during the water entry process were obtained. Furthermore, the frequency-domain characteristics of acceleration were derived by combining modal analysis and shock response spectrum analysis. The relevant research conclusions can provide supplements to the existing numerical simulation results of vehicle water entry.

Keywords: Numerical Simulation, S-ALE Method, Water Entry, Cavity, Acceleration.

1. Introduction

Water entry is a phenomenon that widely exists in nature and daily life, and it is also a key technical issue in fields such as naval architecture and ocean engineering, as well as aerospace engineering. In nature, behaviors including raindrop falling, fish leaping out of and re-entering water, and birds diving to prey all fall into the category of typical water entry events. In daily life, activities such as athletes diving, rescue equipment being thrown into water, and the design of water slides in water parks are all closely related to water entry problems. In the fields of naval architecture and ocean engineering, and aerospace engineering, research on issues such as the water entry of air-dropped vehicles and marine detection equipment holds significant engineering practicality and strategic importance. Research on water entry problems mainly focuses on two aspects: cavity evolution and load characteristics.

Owing to the strong nonlinearity and unsteadiness of water entry problems, early research on water entry cavity relied primarily on experimental methods. The research objects were mainly cylindrical bodies with shapes similar to those of vehicles or projectiles, and the water entry velocities of the objects in the experiments were relatively low. He et al.[1] conducted studies on the low-speed water entry of cylindrical bodies, analyzed a series of flow phenomena including the generation, development, and closure of cavity during the water entry process, and summarized the cavity closure laws. Lu et al.[2] performed low-speed oblique water entry experiments on projectiles based on the high-speed photography technology, and derived the influence laws of head shapes on water entry cavity, motion velocity, pitch angle, and drag coefficient. In terms of theoretical research, most studies have been expanded based on the principle of independent expansion of cavity sections. Mirzaei et al.[3] modified the principle of independent expansion of cavity sections, considered the effects of cavity memory effect, gravity, and angle of attack on the axial deformation of cavity, and proposed an empirical model for predicting the cavity shape of cylindrical bodies during water entry. Jiao et al.[4] established an unsteady supercavitation calculation method for the high-speed water entry of axisymmetric bodies based on the principle of independent expansion of cavity sections, and realized the solution of the motion parameters of axisymmetric bodies by combining with the dynamic model of axisymmetric bodies. With the advancement of computer technology, numerical simulation has enriched the research methods for water entry cavity problems. Wang et al.[5] carried out numerical research on the vertical water entry process of slender projectiles at a constant velocity, analyzed the variation laws of projectile drag coefficient, flow field pressure distribution, water entry cavity evolution, and internal flow characteristics of cavity, and obtained the relationship

between the above parameters and Froude number. Chang et al.[6] performed numerical calculations on the oblique water entry process of cylindrical bodies, studied the formation and development of water entry cavity, and derived the effects of parameters such as head shape, water entry velocity, water entry angle, and fluid density on the water entry depth and inclination angle of cylindrical bodies.

In the early stage, research on water entry impact loads mainly focused on the problems of aircraft ditching and ship slamming, with most studies simplifying the research objects into simple geometric shapes such as wedges and cylinders. Von Karman[7] simplified the aircraft ditching problem into a two-dimensional wedge slamming onto a calm water surface. Neglecting factors such as the gravity of the wedge and the surface tension of water, he solved the slamming pressure of the wedge during water entry based on the law of conservation of momentum. On the basis of von Karman's work, Wagner[8] introduced a wave correction factor to account for the water surface uplift during the wedge water entry process, and obtained the pressure distribution on the wetted surface of the wedge that was more consistent with the experimental test results. Oliver[9] derived the second-order correction for the force at the impact point and in the root region of the high-pressure jet during structural water entry through systematic matched asymptotic analysis, expanding the Wagner model and improving its prediction accuracy. Kou et al.[10] replaced the equivalent flat plate theory in the Wagner model with elliptical fitting, and obtained the pressure distribution on the wetted surface when the wedge entered water at a constant velocity. With the development of technologies related to air-dropped vehicles and rocket-assisted vehicles, the water entry velocity of vehicles has been further increased, necessitating attention to the impact of impact loads induced by high-speed water entry on vehicles. In addition to focusing on the time-frequency domain characteristics of impact loads themselves, it is also necessary to study the dynamic response of vehicle shell structures excited by such impact loads. Sun et al.[11] established a fluid-structure interaction numerical model for the high-speed water entry of cylinders, and obtained the oscillatory variation characteristics of impact loads as well as the deformation and effective stress distribution of cylinders during water entry. Wang et al.[12] performed numerical simulations on the water entry process of large-caliber flat-nosed projectiles under different velocities, angles, and angles of attack, and derived the effects of different initial conditions on the axial and radial loads of projectiles during water entry.

To date, research on vehicle water entry has predominantly focused on small-scale vehicles (maximum diameter ≤ 324 mm), whereas investigations into the water entry of large-scale vehicles (maximum diameter ≥ 533 mm) at velocities ≥ 100 m/s remain insufficient and thus demand further exploration. Against this backdrop, the present study aims to address the water entry issue of air-dropped vehicles. A numerical model is established based on the structured arbitrary Lagrangian-Eulerian (S-ALE) method. The core objectives are to capture the cavity evolution characteristics and acceleration responses during the water entry process, thereby supplementing and enriching the existing body of research findings. The subsequent structure of this paper is organized as follows: Section 2 elaborates on the establishment of the numerical model, including the fundamental theory of the S-ALE method (encompassing governing equations, fluid-structure interaction algorithms, and interface reconstruction algorithms) as well as the detailed parameter settings of the model. Section 3 presents the analysis and discussion of the simulation results, focusing on water entry cavity characteristics, time-domain and frequency-domain characteristics acceleration. Finally, Section 4 summarizes the main conclusions derived from this study.

2. Set-up of Numerical Model

2.1 Theory of S-ALE Method

The governing equations of the S-ALE method are presented as follows:

$$\frac{\partial \mathbf{J}}{\partial t} = \mathbf{J} \frac{\partial v_i}{\partial x_i} \quad (1)$$

Where $\mathbf{J}$ denotes the Jacobian matrix, representing the relative volume differential between the two

domains.

The material time derivative in the S-ALE method is expressed as follows:

$$\frac{\partial b^r}{\partial t} = b - w \frac{\partial f}{\partial x} \quad (2)$$

where b^r denotes the reference domain function of b .

The flow field governing equations of the S-ALE method consist of the mass conservation equation (equation (3)), momentum conservation equation (equation (4)) and energy conservation equation (equation (5)):

$$\frac{\partial \rho^r}{\partial t} = -\rho \frac{\partial u_i}{\partial x_i} - w_i \frac{\partial \rho}{\partial x_j} \quad (3)$$

$$\rho \frac{\partial u_i^r}{\partial t} = (\sigma_{ij,j} + \rho b_i) - \frac{\partial u_i}{\partial x_i} - \rho w_j \frac{\partial u_i}{\partial x_j} \quad (4)$$

$$\rho \frac{\partial e^r}{\partial t} = (\sigma_{ij} u_{i,j} + \rho b_i u_i) - \rho w_j \frac{\partial e}{\partial x_j} \quad (5)$$

Where u_i denotes the material velocity. w_i represents the relative velocity, with $w_i = u_i - v_i$. v_i stands for the mesh velocity. σ_{ij} is the stress tensor, $\sigma_{ij} = -p\delta_{ij} + \mu(v_{ij} + v_{ji})$. b_i refers to the body force per unit volume. δ_{ij} denotes the Kronecker delta function.

The mesh governing equation of the S-ALE method is expressed as follows:

$$\frac{\partial f(X_i, t)}{\partial t} = \frac{\partial(x_i, t)}{\partial t} + w_i \frac{\partial(x_i, t)}{\partial x_j} \quad (6)$$

Where X_i denotes the Lagrangian coordinate.

For the fluid-structure interaction (FSI) algorithm, a penalty-based coupling method is adopted. The magnitude of the coupling force is proportional to the penetration depth of the solid into the fluid, and this force restricts the penetration of solid into the fluid. Within a single time step, the coupling force acts simultaneously on the nodes involved in the FSI, preventing the penetration of the node interface. Assuming the relative velocity between the structural node and the fluid node is $(v_s - v_f)$, the structural node penetrates the free surface at this relative velocity, and the penetration depth is calculated using equation (7).

$$d^n = d^{n-1} + (v_s - v_f) \cdot \Delta t \quad (7)$$

The magnitude of the coupling force is calculated using equation (8) and equation (9).

$$F = kd \quad (8)$$

$$k = p_f \frac{KA^2}{V} \quad (9)$$

where k denotes the coupling stiffness. d represents the penetration depth of the solid into the fluid. K is the bulk modulus of the element. A stands for the area of the coupling segment. V denotes the volume of the main segment. p_f is the penalty factor, with the value range satisfying $0 \leq p_f \leq 1$.

For the problem of water entry impact, when the coupling force is extremely large, penetration and separation may occur within the same time step, resulting in numerical simulation instability.

To avoid numerical instability, the magnitude of the coupling force is limited, as expressed in equation (10).

$$F \leq \frac{M_s M_m (v_s - v_f)}{\Delta t (M_s + M_m)} \quad (10)$$

where M_s denotes the mass of the structural node. M_m represents the mass of the fluid node.

The interface reconstruction algorithm adopts the Volume of Fluid (VOF) method. In the S-ALE method, the fluid domain is discretized using structured orthogonal meshes, which facilitates the implementation of interface reconstruction via the VOF method. The principle of the VOF method is illustrated in Figure 1. A piecewise linear interface reconstruction method is employed to reconstruct the fluid interface. In a three-dimensional element, the fluid interface is a planar surface, whose determination requires two key parameters: one is the fluid volume fraction within the element, and the other is the normal vector of the fluid interface. Specifically, the fluid volume fraction is used to define the position of the fluid interface, while the normal vector of the fluid interface serves to determine the orientation of the fluid interface.

The specific implementation process of fluid interface reconstruction is to calculate the fluid volume in each element and determine the fluid volume fraction within each time step. For the solution of the fluid interface in a specific element, the fluid volume fractions of both the target element and its adjacent elements are required. The gradient of the fluid volume fraction in the target element is calculated using the finite difference method, and the direction of the normal vector of the fluid interface can be obtained by normalizing the aforementioned gradient. The fluid interface can be defined by equation(11).

$$\mathbf{n} \cdot \mathbf{x} - d = 0 \quad (11)$$

where d denotes the shortest distance between the fluid interface and the coordinate origin, and its value is determined by the fluid volume fraction within the element.

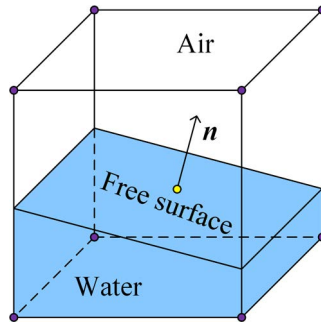


Figure 1: Schematic diagram of the interface reconstruction algorithm.

2.2 Settings of Numerical Model

The shell of the vehicle is made of 6061-T6 aluminum alloy, and an elastic-plastic constitutive model is adopted for the material. The density of 6061-T6 aluminum alloy is 2686 kg/m³, the elastic modulus is 72.4 GPa, the Poisson's ratio is 0.32, the yield strength is 286.8 MPa, and the tangent modulus is 542.6 MPa.

The air is described by a linear polynomial equation of state, as shown in equation(12).

$$p_a = C_0 + C_1 \mu_a + C_2 \mu_a^2 + C_3 \mu_a^3 + (C_4 + C_5 \mu_a + C_6 \mu_a^2) e_a \quad (12)$$

where p_a denotes the air pressure. $C_0, C_1, C_2, C_3, C_4, C_5, C_6$ are polynomial coefficients, where C_0 is used to define the initial pressure and C_1 represents the volume viscosity. e_a is the internal energy per unit volume of air. $\mu_a = \frac{1}{\psi_a} - 1$, with ψ_a being the relative volume of air, which is employed to characterize the compression state of air. The values of the coefficients are set as follows: $C_0 = C_1 = C_2 = C_3 = 0$, $C_4 = C_5 = 0.4$, $C_6 = 0$. The initial internal energy per unit volume of air e_{a0} is 250 kJ/m^3 , the density is 1.25 kg/m^3 , and the dynamic viscosity coefficient is $1.747 \times 10^{-5} \text{ Pa}\cdot\text{s}$.

The water is described by the Grüneisen equation of state, as shown in equation (13).

$$p_w = \frac{\rho_w c^2 \mu_w \left(1 + \left(1 - \frac{\gamma_0}{2}\right) \mu_w - \frac{\alpha}{2} \mu_w^2\right)}{\left(1 - (S_1 - 1) \mu_w - S_2 \frac{\mu_w^2}{\mu_w + 1} - S_3 \frac{\mu_w^3}{(\mu_w + 1)^3}\right)^2} + (\gamma_0 + \alpha \mu_w) e_w \quad (13)$$

$$\mu_w = \frac{1}{\psi_w} - 1 \quad (14)$$

where p_w denotes the water pressure. c is the intercept of the u_s-u_p curve, representing the velocity of sound. γ_0 is the Grüneisen constant. α is the first-order volume correction of γ_0 . e_w is the volumetric internal energy of water. S_1, S_2, S_3 are the coefficients of the slope of the u_s-u_p curve. ψ_w is the relative volume of water, which is used to characterize the compression state of water. The parameter values are set as follows: $c = 1480 \text{ m/s}$, $S_1 = 2.56$, $S_2 = -1.986$, $S_3 = 0.226$, the initial volumetric internal energy of water $e_{w0} = 0$, the initial relative volume $\psi_{w0} = 1$, the density is 998 kg/m^3 , and the dynamic viscosity coefficient is $8.684 \times 10^{-4} \text{ Pa}\cdot\text{s}$.

The overall length of the vehicle is 5.85 m, with a maximum diameter of 0.533 m and a total mass of 1582 kg. The vehicle is discretized using hexahedral meshes with an average mesh size of 0.01 m. The fluid domain is configured with dimensions of $6 \text{ m} \times 6 \text{ m} \times 9 \text{ m}$, and the average mesh size of the fluid domain is 0.02 m. Details regarding mesh independence verification can be referred to previous work[13].

A global coordinate system and a local coordinate system are defined to describe the position and attitude of the vehicle, as well as to analyze its mechanical and motion characteristics during the water entry process, as illustrated in Figure 2.

The origin O_0 of the global coordinate system ($O_0-x_0y_0z_0$) is set at the water entry point. The O_0x_0 -axis lies in the ground plane and points to the direction of vehicle navigation. The O_0y_0 -axis is perpendicular to the ground plane and points upward. The O_0z_0 -axis lies in the ground plane, perpendicular to the $x_0O_0y_0$ plane, and its direction is determined according to the right-hand coordinate rule.

The origin O of the local coordinate system ($O-xyz$) is set at the centroid of vehicle. The Ox -axis lies in the symmetric plane of the vehicle, coincides with the geometric axis of symmetry, and points toward the head. The Oy -axis lies in the longitudinal symmetric plane of the vehicle, is perpendicular to the Ox -axis, and points upward. The Oz -axis is perpendicular to the xOy plane and points to the right when viewed from the tail.

The water entry angle θ is defined as the included angle between the velocity vector v of the vehicle and the horizontal plane.

The angle of attack α is defined as the included angle between the projection of the velocity vector v onto the symmetric plane and the Ox -axis, with the nose-up attitude designated as the positive direction.

2.3 Experimental verification

Furthermore, experimental verification was carried out on the established numerical model in an indoor water tank with a depth of 15 m. The experimental setup and test apparatus are illustrated in Figure 3. The experiment model adopted the fore-body of the vehicle and was scaled down to a maximum diameter

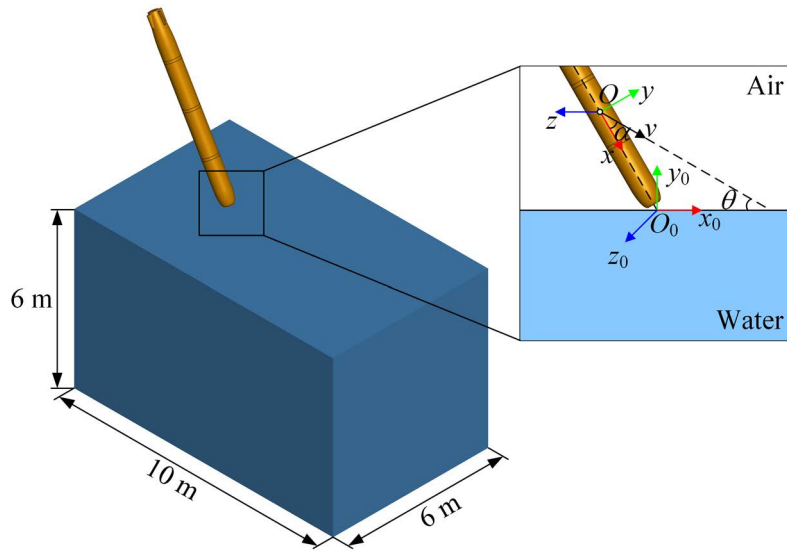


Figure 2: Definition of coordinate system.

of 200 mm. Fabricated from aluminum alloy, the experiment model has a total mass of 15 kg. An acceleration acquisition system was installed inside the experiment model with a sampling frequency of 60 kHz. The experiment model was driven and accelerated by high-pressure gas to reach the preset entry velocity. Meanwhile, a high-speed camera operating at a frame rate of 1000 fps was deployed in the tank to record the evolution of cavity.



(a) Experimental model

(b) Data acquisition system

(c) Underwater high-speed camera

Figure 3: Experimental model and test apparatus.

Figure 4 presents a comparison of cavity between simulation and experiment. Using digital image processing technology, the pixel scale of images was calibrated according to the model dimensions, and the outlines of cavity were extracted and plotted in the same coordinate system. The instant when the model impacts the water surface is defined as time zero, and the results at 30 ms are displayed herein. By comparing the cavity profiles obtained from simulation and experiment, good agreement in morphology is observed. It is verified that the adopted numerical method is applicable to subsequent research on water entry cavity.

Figure 5 compares the acceleration results obtained from simulation and experiment. For intuitive comparison, the acceleration curves are normalized using equation (15). The original acceleration curves reveal two distinct stages: the acceleration stage and the water entry stage. In the acceleration stage, the model is driven by high-pressure gas and exhibits negative acceleration. During the water entry stage, the model collides with the water surface at high speed and experiences impact force, generating a large positive acceleration, which indicates an opposite direction to that in the acceleration stage. Furthermore, the acceleration curves of the water entry stage are intercepted and filtered for comparison with the numerical results. It is found that the two curves share basically the same variation trend. The experimental curve shows more obvious oscillations than the simulated one, which is attributed to the

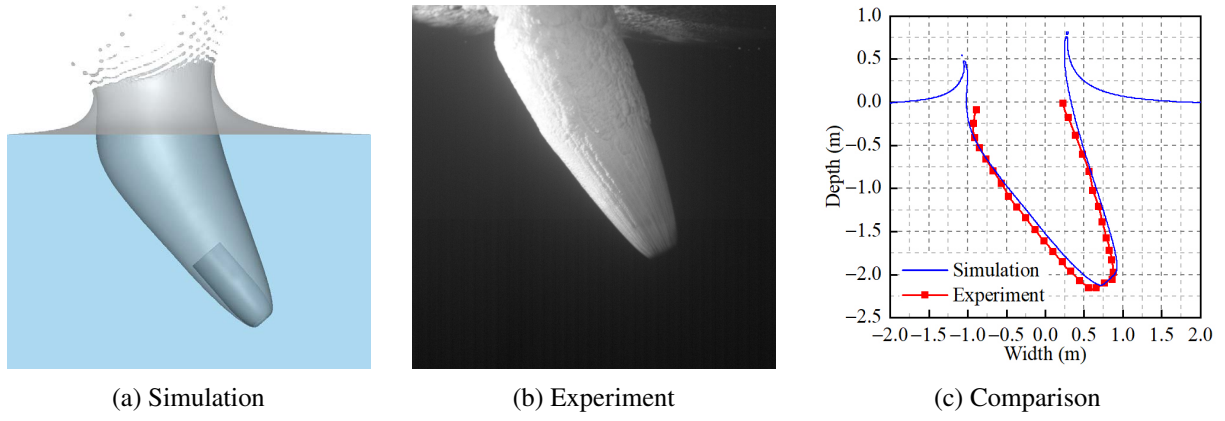


Figure 4: Comparison of cavity between simulation and experiment.

vibration of the data acquisition device fastened by screws during the water entry process. The peak value of the normalized acceleration coefficient from numerical simulation is 0.792, while that from the experiment is 0.756. Taking the experimental data as the true value, the relative error is 4.76%. Consequently, the proposed numerical method is validated for subsequent research on water entry loads.

$$C_d = \frac{2ma}{\rho\pi R^2 V_0^2} \quad (15)$$

Where C_d denotes the dimensionless acceleration coefficient. m is the mass of the model. a represents the acceleration of the model. ρ stands for the density of water. R is the maximum radius of the model. V refers to the initial water-entry velocity of the model.

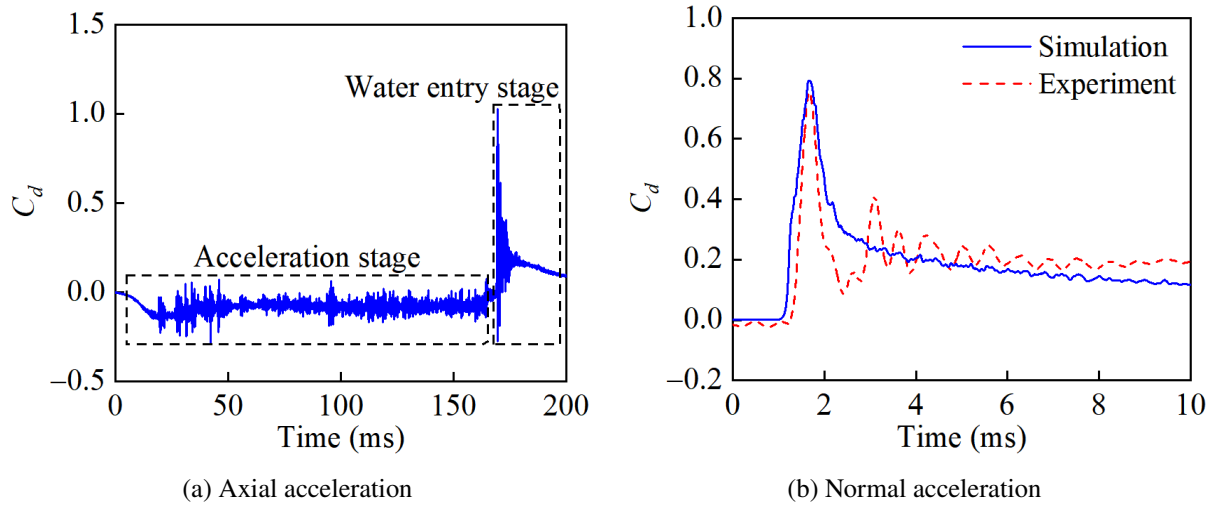


Figure 5: Acceleration curve of vehicle oblique water entry ($v = 100 \text{ m/s}$, $\theta = 60^\circ$).

3. Analysis and Discussion

3.1 Analysis of cavity

Based on the observed flow phenomena and the motion characteristics of the vehicle, the water entry process can be roughly divided into four stages: the water entry impact stage, the cavity opening stage, the cavity closure stage, and the cavity collapse stage, as shown in Figure 6.

During the water entry impact stage, the vehicle rapidly transitions from the low-density air into the high-density water, with its head colliding against the free surface. The water medium surrounding the head acquires a certain amount of kinetic energy, which is transferred and dissipated in the form of wave motion, thereby establishing a perturbed flow field characterized by the head geometry in the originally stationary water. As the water body moves, the flow separates from the vehicle head and a cavity is generated, with only the front end of the vehicle head being wetted at this moment. When the vehicle enters the water at an inclined angle, the cavity exhibits an asymmetric morphology with a smaller scale on the lower side and a larger scale on the upper side, accompanied by large-area droplet splashing on one side.

During the cavity opening stage, the water surface adjacent to the vehicle head rises and induces liquid splashing, generating a crescent-shaped water curtain. As the water entry depth of the vehicle increases, the cavity formed in the previous stage further expands radially and elongates axially. In this stage, the cavity remains open to the atmosphere, allowing air to enter the cavity from the water surface. Given that only the vehicle head was in a wetted state in the preceding stage, the normal acceleration acting on it creates a counterclockwise deflection moment relative to the centroid of vehicle, which drives the vehicle to rotate counterclockwise. Consequently, the lower part of the vehicle comes into contact with the cavity wall, resulting in a wall-sliding phenomenon and accelerating the tensile deformation of the cavity.

During the cavity closure stage, owing to the rapid expansion of the cavity, the speed at which atmospheric air enters the cavity is relatively low. Consequently, the pressure inside the cavity gradually decreases and falls below that of the surrounding water, causing the cavity wall to be compressed by the surrounding water medium and thus undergo contraction. When the cavity wall converges to a single point, cavity closure occurs, dividing the cavity into upper and lower parts. The upper cavity remains connected to the atmosphere, while the lower cavity attaches to the vehicle and moves continuously with it.

During the cavity collapse stage, the concave air-water interface of the upper cavity moves upward under the combined action of elastic force and surface tension, forming a Worthington jet. For the lower cavity, as the vehicle submerges to increasing depths, the hydrostatic pressure surrounding the cavity gradually rises. This causes the volume of the lower cavity to decrease continuously until it collapses completely, resulting in the entire vehicle being fully wetted.

Figure 7 depicts the curves of water entry trajectory, velocity, and pitch angle of the vehicle at an initial velocity of 100 m/s and an entry angle of 60° . In the initial stage of water entry, the vehicle head displaces the water medium and forms a cavity. After cavity formation, the vehicle is enveloped within the cavity, with only the head maintaining a wetted state. During this period, the vehicle is primarily subjected to axial acceleration, while large-amplitude normal acceleration occurs only when the lower end of the vehicle head impacts the water and the rear part of the shell makes contact with the water. The duration of normal acceleration action is relatively short in the initial stage, resulting in a slow variation of the pitch angle of vehicle.

As the water entry process proceeds, the vehicle undergoes counterclockwise deflection under the moment induced by normal acceleration and gravity. This deflection leads to the contact between the lower part of the vehicle shell and the cavity wall, accompanied by the occurrence of wall-sliding phenomenon. When wall-sliding takes place, the entire vehicle is subjected to significant normal acceleration, generating a larger deflection moment, which causes a rapid change in the pitch angle. At 500 ms, the pitch angle of the vehicle decreases from the initial 60° to 12° , and the water entry trajectory gradually flattens.

According to the curve of velocity, the velocity of vehicle decays rapidly in the early stage of water entry and slowly in the late stage. This phenomenon is mainly attributed to the following mechanisms: in the early stage, the vehicle needs to displace the water medium to form the cavity, during which a large amount of its kinetic energy is converted into the kinetic energy of the surrounding water medium. Moreover, after cavity formation, the characteristic cross-sectional area of the vehicle increases, leading to a rise in water entry resistance and thus rapid velocity attenuation. In the late stage, after the cavity is torn off, the attached cavity of the vehicle collapses continuously, resulting in a reduction in the characteristic

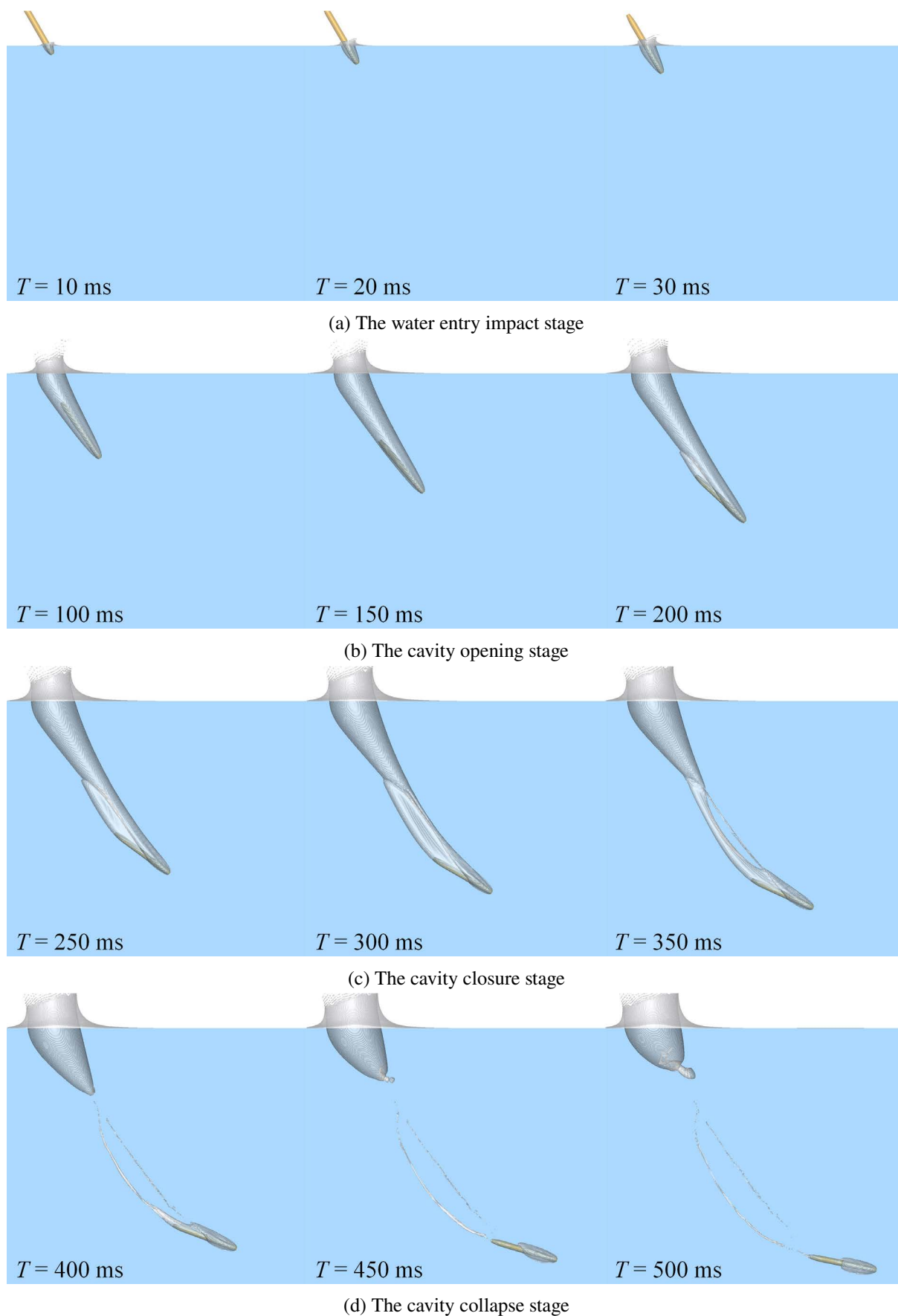


Figure 6: Cavity of vehicle water entry at different moments($v = 100$ m/s, $\theta = 60^\circ$).

cross-sectional area of the vehicle, a decrease in water entry resistance, and consequently a slowdown in velocity attenuation. At 500 ms, the velocity of the vehicle attenuates from the initial 100 m/s to 40 m/s.

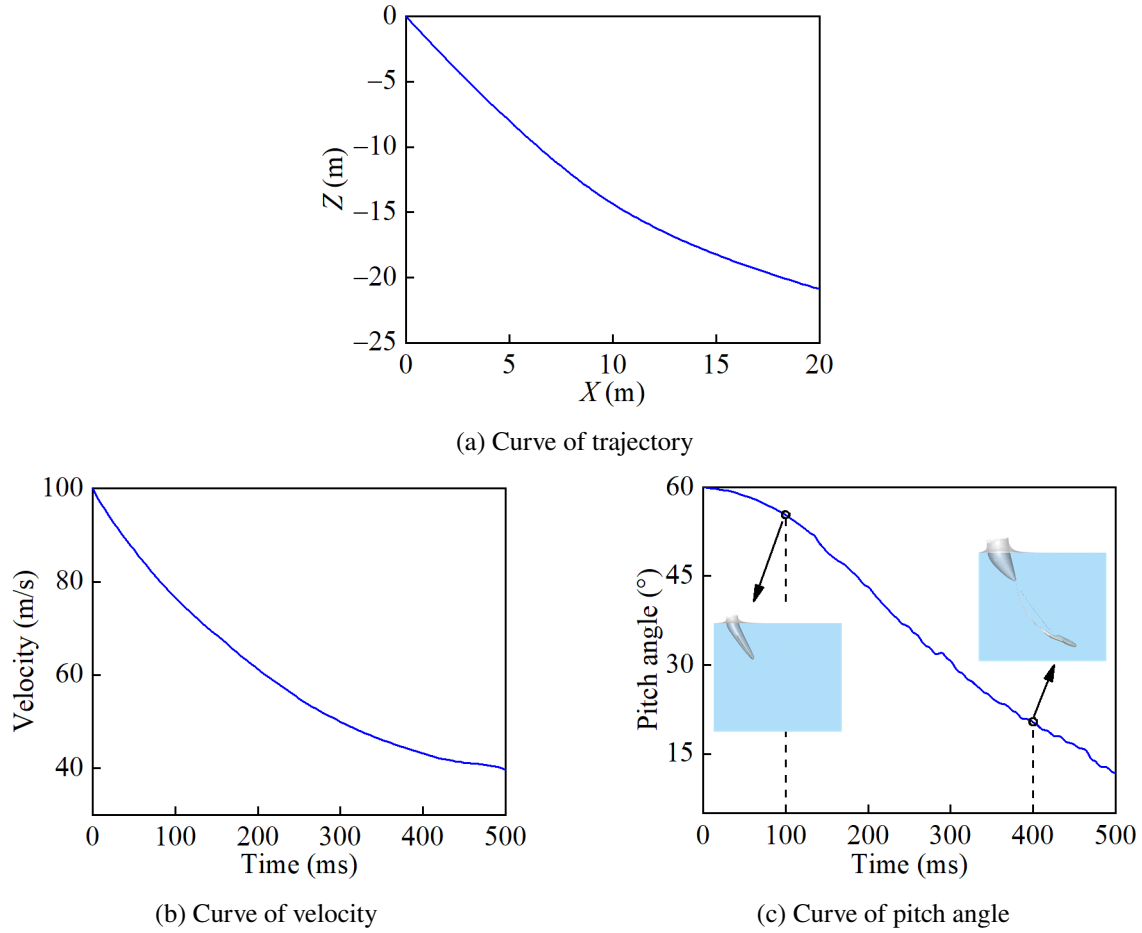


Figure 7: Curves of water-entry trajectory, velocity and pitch angle($v = 100$ m/s, $\theta = 60^\circ$).

3.2 Time-domain characteristics of acceleration

Throughout the entire water entry process, the acceleration acting on the vehicle can be primarily decomposed into axial acceleration and normal acceleration. Axial acceleration originates from the vehicle head and propagates along the longitudinal axis toward the tail. It is the dominant factor causing structural damage to the shell and malfunction of internal components. The formation of normal acceleration is mainly induced by the circumferential non-uniform distribution of the wetted surface of the vehicle. This non-uniform distribution generates a deflection moment at the centroid of vehicle, resulting in transverse angular acceleration of the vehicle and thus leading to changes in its pitching attitude.

Figure 8 depicts the acceleration curves of the vehicle during oblique water entry at an initial velocity of 100 m/s and an entry angle of 60° . During oblique water entry, the magnitude of the normal acceleration is smaller than that of the axial acceleration. After the vehicle head comes into contact with the free surface, the axial acceleration rises rapidly, reaches a peak value, then drops sharply and exhibits an oscillatory attenuation behavior, which is a typical characteristic of collision. Upon the completion of collision, the vehicle continues to move forward, displacing the water medium surrounding the head and forming a cavity. The axial acceleration is then gradually converted into the motion resistance of the vehicle navigating with the attached cavity, and its magnitude decreases progressively as the motion velocity of the vehicle attenuates. The normal acceleration reaches a large peak at the initial stage of water entry when the lower end of the vehicle head impacts the free liquid surface. Since the formation of normal acceleration is mainly induced by the circumferential non-uniform distribution of the wetted

surface, the peak of normal acceleration generally occurs earlier than that of axial acceleration. With the formation of the cavity, most of the vehicle shell is enveloped in the cavity, with only the head maintaining a symmetric wetted state. Consequently, the normal acceleration decreases rapidly and fluctuates around zero. As the water entry process proceeds, the vehicle undergoes a certain degree of attitude deflection, which causes the rear part of the shell to come into contact with water and thus induces fluctuations in the normal acceleration.

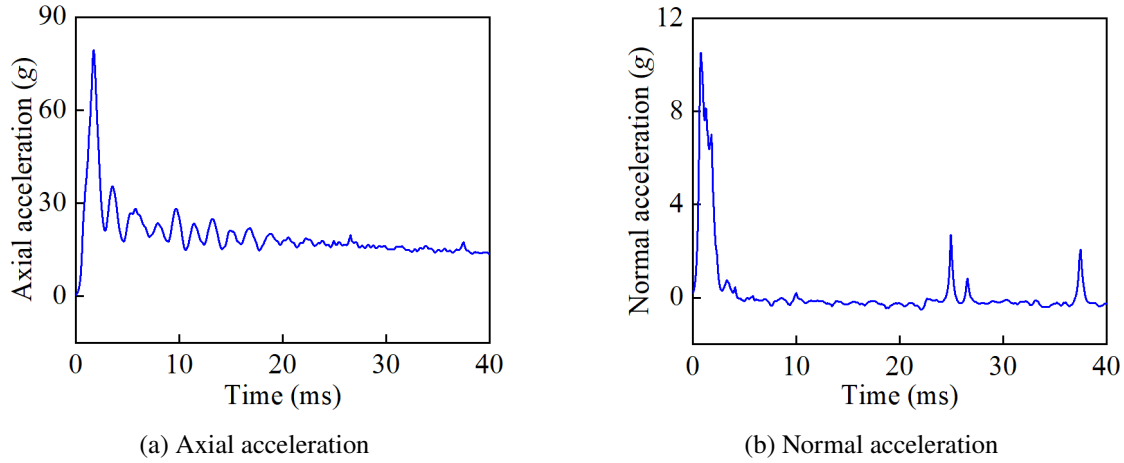


Figure 8: Acceleration curve of vehicle oblique water entry($v = 100$ m/s, $\theta = 60^\circ$).

3.3 Modal analysis of vehicle

A mode refers to the inherent, global dynamic characteristic of an elastic structure. When a structural system undergoes motion under external excitation, it will vibrate naturally at a specific frequency, which is defined as the natural frequency of the structure. The corresponding vibration configuration is termed the mode shape. A structure possesses multiple modes, with each mode characterized by a unique natural frequency and mode shape. In the presence of external excitation, the structural vibration response is generated in accordance with its natural frequencies.

Modes can be classified into dry modes and wet modes according to the environment where the structure is located. A dry mode is defined as the mode of a structure in a vacuum environment, without considering the influence of the surrounding fluid. For instance, air has relatively low density and viscosity, leading to a negligible effect on the structural modes; therefore, dry modal analysis is generally adopted for structures in air. A wet mode refers to the mode of a structure in a fluid environment, such as the mode of a structure fully submerged in water. In this case, water exerts a significant influence on the natural frequency and mode shape of the structure, and thus the effect of water must be taken into account.

The modes of a structure are independent of external excitation and represent an inherent property of the structure itself. When solving for the natural frequencies and mode shapes of a structure, there is no need to consider external excitation, and the governing equation of motion for the solution is as follows:

$$[m][\ddot{x}] + [c][\dot{x}] + [k][x] = [0] \quad (16)$$

For modal analysis, the modal results are correlated with the mass, damping, and stiffness of the structure. In the case of ideal free vibration where the damping effect is neglected, the governing equation of motion for the solution is as follows:

$$[m][\ddot{x}] + [k][x] = [0] \quad (17)$$

The equation for a structure undergoing simple harmonic motion at a constant frequency is expressed as

follows:

$$\begin{cases} [x] = [\varphi_i] \sin(\omega_i t + \theta_i) \\ [\dot{x}] = \omega_i [\varphi_i] \cos(\omega_i t + \theta_i) \\ [\ddot{x}] = -\omega_i^2 [\varphi_i] \sin(\omega_i t + \theta_i) \end{cases} \quad (18)$$

Substituting the above equation into the governing equation of motion and performing simplification, the final expression is derived as follows:

$$[k] - \omega_i^2 [m] [\varphi_i] = [0] \quad (19)$$

The condition for the above equation to hold is that $[\varphi_i] = [0]$ (indicating no vibration) or the following equation is satisfied:

$$\det([k] - \omega_i^2 [m]) = 0 \quad (20)$$

Solving equation (19), ω_i is obtained as the natural angular frequency of the structure, from which the natural frequency f_i can be derived using the relation $f_i = \omega_i/2\pi$. The corresponding eigenvector ϕ_i represents the vibration state of the structure oscillating at the natural frequency f_i . By solving for different angular frequencies and their associated eigenvectors, the modes and mode shapes of each order for the structure can be acquired.

During the water entry process of the vehicle, the head collides with the free liquid surface and excites the dynamic response of the shell structure. In this scenario, the head is in a wetted state while most of the shell is exposed to the air. Therefore, for modal analysis, the head is defined as a fixed constraint to solve for the natural frequencies of each mode of the vehicle and their corresponding mode shapes. For dry modal analysis, it is assumed that the vehicle is surrounded by a vacuum environment. In contrast, for wet modal analysis, the vehicle is assumed to be immersed in a water medium and in a fully wetted state, as illustrated in Figure 9.

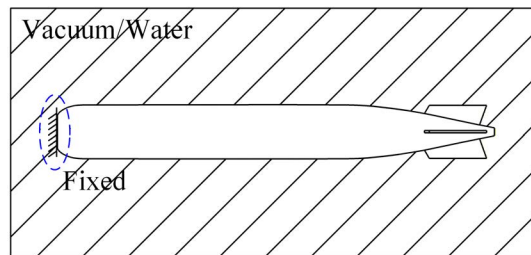


Figure 9: Schematic diagram of fixed constraints on vehicle head.

Table 1 presents a comparison of the natural frequencies between the dry modes and wet modes of the vehicle. Compared with the dry modes of each order, the natural frequencies of the wet modes of each order are all reduced. This is mainly because, with the structural parameters and external constraint conditions unchanged, the dominant factor affecting the natural frequencies of each mode of the vehicle is the mass of the vehicle. When the vehicle is fully submerged in water and undergoes vibration, water exerts a certain damping effect on the vibration of the vehicle. In this case, it is necessary to introduce the added mass induced by water into the mass of the vehicle, which leads to an increase in the equivalent mass of the vehicle. Consequently, the natural frequencies of each mode of the vehicle are reduced.

Figure 10 illustrates the first 20 mode shapes of the vehicle obtained via dry modal analysis. As the modal order increases, the mode shapes become increasingly complex. The low-order modes exhibit simple deformation patterns such as swinging, bending, compression, and tension, while the high-order modes

Table 1: Natural frequency of the vehicle

Modal order	Natural frequency of dry modal (Hz)	Natural frequency of wet modal (Hz)
1st	5.56	4.68
2nd	5.56	4.73
3rd	42.15	32.80
4th	42.16	32.90
5th	78.06	78.62
6th	118.30	91.71
7th	118.77	92.26
8th	118.78	92.37
9th	123.54	103.07
10th	123.56	103.53
11th	131.51	110.88
12th	131.63	111.49
13th	154.90	123.06
14th	155.31	130.09
15th	195.43	130.62
16th	196.51	161.59
17th	220.65	162.15
18th	220.65	173.02
19th	234.91	173.09
20th	247.36	186.46

are dominated by deformation patterns including bending and torsion. Moreover, the number of nodes corresponding to bending and torsion increases with the rise of modal order.

Figure 11 illustrates the first 20 mode shapes of the vehicle obtained via wet modal analysis. Most of these mode shapes are similar to those derived from dry modal analysis, including swinging, bending, and torsion, and the majority of mode shapes also appear in pairs. The key difference from dry modal analysis is that the vibration amplitude of the overall structure is significantly smaller. The damping effect of water reduces the natural frequency and vibration amplitude of the vehicle during vibration. Additionally, several mode shapes that do not appear in dry modal analysis are observed, such as the 6th-order mode and the 20th-order mode. This indicates that the influence of water on the overall structural response must be considered when the vehicle is fully submerged in water.

3.4 Frequency-domain characteristics of acceleration

The shock response spectrum (SRS) correlates shock acceleration with system response, which can evaluate the effects of shock loads acting on structures. Moreover, the SRS possesses the characteristic of response equivalence, enabling it to better approximate the actual shock environment. Therefore, the calculation of the shock response spectrum is carried out.

In simple terms, the shock response spectrum (SRS) is a curve plotted in the Cartesian coordinate system, where the natural frequency of a single-degree-of-freedom (SDOF) system is taken as the abscissa and the peak response value is taken as the ordinate. Before solving the shock response spectrum of the entire system, it is necessary to derive the shock response function of the SDOF system. The physical model of the SDOF system is illustrated in Figure 12, with the external excitation assumed to be a half-sine excitation.

The mathematical equation of the single-degree-of-freedom (SDOF) system is expressed as follows:

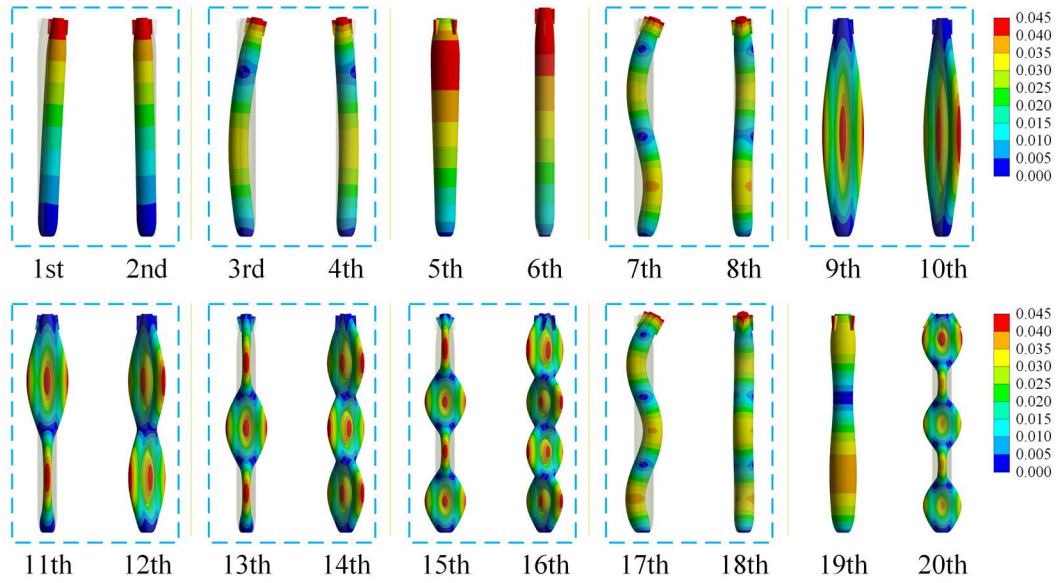


Figure 10: First twenty order modal shapes of vehicle of dry modal analysis.

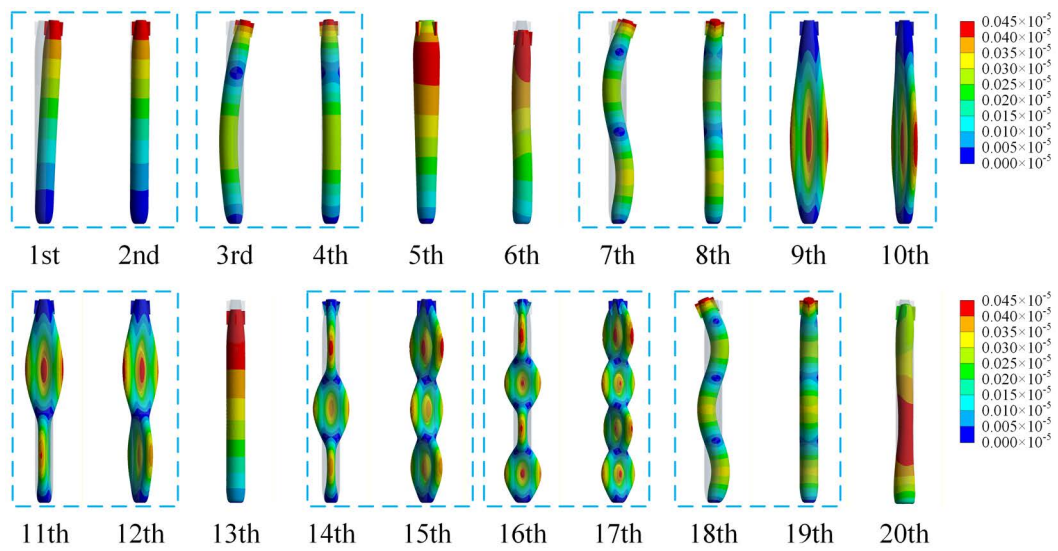


Figure 11: First twenty order modal shapes of vehicle of wet modal analysis.

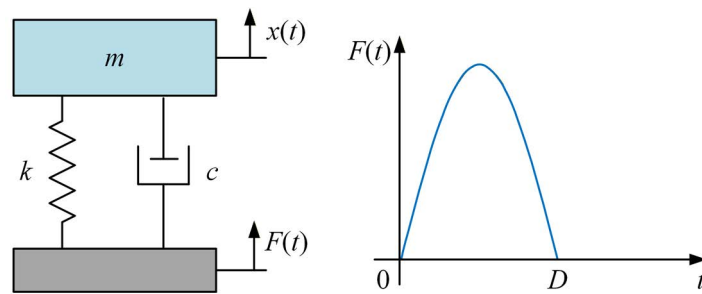


Figure 12: Physical model of a single-degree-of-freedom system.

$$m\ddot{x}(t) + c\dot{x}(t) + kx(t) = F(t) \quad (21)$$

where m denotes the mass of the system. c denotes the damping coefficient of the system. k denotes the stiffness coefficient of the system.

The form of the mathematical equation for the single-degree-of-freedom (SDOF) system (Equation (20)) is similar to that of the governing equation of motion adopted for solving each mode of the vehicle (Equation (15)). However, when solving the shock response spectrum (SRS), the external excitation acceleration is incorporated into the equation. In addition to this difference, the SRS solution is conducted for an SDOF system, whereas modal analysis requires consideration of a multi-degree-of-freedom (MDOF) system, which is decoupled into a series of SDOF systems for analysis.

When calculating the SRS, it is only necessary to obtain the characteristics of the external excitation acceleration, rather than the properties of the actual structure. This is because, after transformation, the description of an SDOF mass-damping system only requires two parameters: the natural frequency and damping ratio of the system. When the damping ratio of the system is assumed to be a constant, the SRS data within the analysis frequency band can be derived by traversing the data within the upper and lower limits of the analysis frequency at a specific octave interval.

For equation (20), it is assumed that

$$\begin{cases} c/m = 2f_n\xi \\ k/m = f_n^2 \\ f(t) = F(t)/m \end{cases} \quad (22)$$

where f_n denotes the natural frequency of the system. ξ denotes the damping ratio of the system.

Then the mathematical equation of the system can be transformed into the following form:

$$\ddot{x}(t) + 2f_n\xi\dot{x}(t) + f_n^2x(t) = f(t) \quad (23)$$

The general solution of the equation is expressed as follows:

$$X(t) = -\frac{1}{2\pi f_n\sqrt{1-\xi^2}} \times \int_0^t f(\tau) \times e^{-2\pi f_n\xi\tau} \sin\left[2\pi f_n\sqrt{1-\xi^2}(t-\tau)\right] d\tau \quad (24)$$

The above equation represents the relational expression of the system's displacement response with respect to time t and natural frequency f_n , denoted as $X(t, f_n)$. By taking the second-order derivative of the above equation, the relational expression of the acceleration response with respect to time t and natural frequency f_n can be obtained, denoted as $\ddot{X}(t, f_n)$.

Assume that the duration of the pulse is t_d and the natural frequency corresponding to the single-degree-of-freedom (SDOF) system is f_n . First, let $f_{n1} = f_n$ and $t = t_a$, with time increasing at a step size of Δt .

$$\begin{cases} t_1 = t_a \\ t_2 = t_1 + \Delta t \\ t_3 = t_2 + \Delta t \\ \vdots \end{cases} \quad (25)$$

Substitute t_1, t_2, t_3 , etc. into the equation respectively to calculate the corresponding acceleration

responses.

$$\begin{cases} \ddot{X}_1(t_1, f_{n1}) \\ \ddot{X}_2(t_2, f_{n1}) \\ \ddot{X}_3(t_3, f_{n1}) \\ \vdots \end{cases} \quad (26)$$

Compare all the acceleration responses $\ddot{X}_i(t_i, f_{n1})$ and obtain the maximum value among them, denoted as $\ddot{X}_i(t_i, f_{n1})_{\max}$. Then, $\ddot{X}_i(t_i, f_{n1})_{\max}$ and the natural frequency f_{n1} form a corresponding set of data. Subsequently, increase the natural frequency by a step size of Δf (where Δf is calculated according to the octave interval).

$$\begin{cases} f_{n1} = f_{nk} \\ f_{n2} = f_{n1} + \Delta f \\ f_{n3} = f_{n2} + \Delta f \\ \vdots \end{cases} \quad (27)$$

A series of data points corresponding to the maximum acceleration response values $\ddot{X}_{\max}$ and the natural frequencies f_n can thus be obtained, and the shock response spectrum curve is generated by connecting these points.

The shock response spectrum (SRS) can be divided into the initial shock response spectrum and the residual shock response spectrum. The initial shock response spectrum is derived by extracting the corresponding peak values within the duration of the shock load, while the residual shock response spectrum is obtained by extracting the corresponding peak values after the termination of the shock load. The spectrum curve obtained through the aforementioned process refers to the initial shock response spectrum. When calculating the residual shock response spectrum, it is necessary to assume that there is no external excitation acting on the system, as expressed by the following equation:

$$\ddot{x}(t) + 2f_n\xi\dot{x}(t) + f_n^2x(t) = 0 \quad (28)$$

Obtain the general solution of this equation and follow the aforementioned steps to derive the residual shock response spectrum.

Taking the shock acceleration time-history curve illustrated in Figure 8 as an example, the calculation of the shock response spectrum is performed to obtain the initial shock response spectrum and the residual shock response spectrum.

Figure 13 presents the calculated results of the shock response spectrum (SRS), where the solid red line denotes the initial SRS and the dashed blue line represents the residual SRS. The subsequent analysis is focused on the initial SRS. Based on the response amplitude of the SRS, the curve can be roughly divided into three regions, namely the buffer region, amplification region, and constant-impulse region.

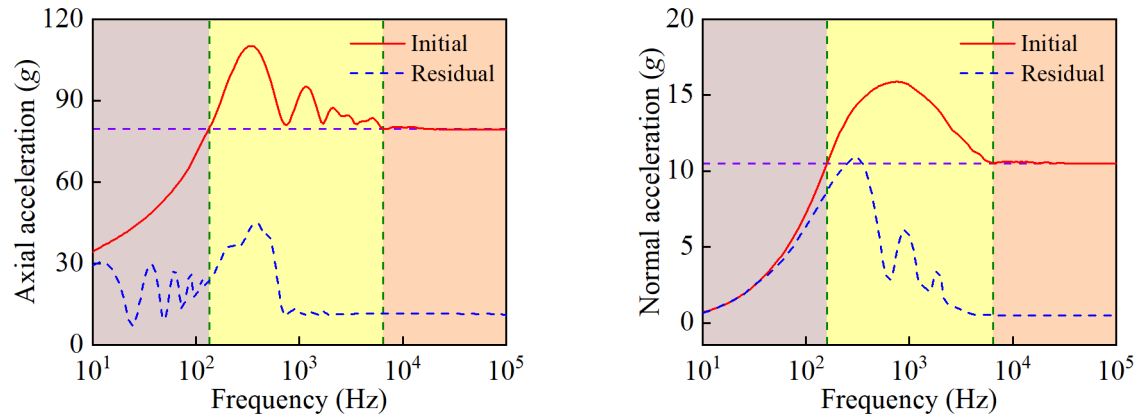
The buffer region corresponds to low natural frequencies, where the system response amplitude is lower than the pulse peak, exhibiting a buffering characteristic. The amplification region corresponds to medium-to-high natural frequencies, where the system response amplitude exceeds the pulse peak, showing an amplifying effect. The constant-impulse region corresponds to the highest natural frequencies, where the shock response amplitude of the system is approximately equal to the peak acceleration of the shock pulse, demonstrating a constant-impulse behavior.

An observation of the overall variation trend of the SRS reveals that with the increase in the natural

frequency of the single-degree-of-freedom (SDOF) mass-damping system, the resulting peak response first increases and then decreases, until it approximates the peak acceleration of the shock pulse. A comparative analysis of the SRSs derived from axial acceleration and normal acceleration indicates that the peak response of the axial acceleration reaches 110.19 g at a corresponding natural frequency of 339.03 Hz, while the peak response of the normal acceleration is 15.88 g at a corresponding natural frequency of 761.09 Hz. It follows that the peak response of the axial acceleration SRS is higher than that of the normal acceleration SRS, whereas its corresponding natural frequency is lower.

The time-domain characteristics of the axial acceleration and normal acceleration indicate that both the peak value and peak pulse width of the axial acceleration are larger than those of the normal acceleration. This reveals that, in the time domain, a larger peak value of the shock acceleration corresponds to a greater peak structural response excited in the frequency domain. In contrast, a narrower peak pulse width of the shock acceleration in the time domain leads to a higher natural frequency corresponding to the peak structural response excited in the frequency domain.

It can thus be concluded that for a single shock acceleration: assuming the pulse width remains constant, an increase in the peak value will enhance the damage potential of the acceleration to the system; assuming the peak value is fixed, a wider peak pulse width will amplify the damage potential to systems with low natural frequencies, whereas a narrower peak pulse width will intensify the damage potential to systems with high natural frequencies.



(a) Shock response spectrum of axial acceleration

(b) Shock response spectrum of normal acceleration

Figure 13: Solution results of shock response spectrum($v = 100$ m/s, $\theta = 60^\circ$).

Different initial water entry conditions exert a significant influence on the peak response of the shock response spectrum (SRS) and its corresponding natural frequency. Combined with the results of the dry modal analysis of the vehicle, it is found that the natural frequencies corresponding to the dominant modes involved in the vibration of the vehicle shell are basically located in the buffer region of the SRS. In contrast, the internal instruments and equipment of the vehicle, owing to their smaller mass relative to the shell, possess higher natural frequencies, which may fall within the amplification region and constant-impulse region of the SRS.

4. Conclusions

This study focuses on the water entry problem of vehicles. A numerical model was established based on the S-ALE method, and numerical simulations were carried out. The main conclusions are as follows:

- (1) The water entry process of the vehicle can be divided into four phases: water-entry impact phase, cavity opening phase, cavity closure phase, and cavity collapse phase. For oblique water entry, the cavity presents an asymmetric distribution pattern.

(2) The water entry acceleration of the vehicle can be decomposed into axial acceleration and normal acceleration. When the vehicle enters water at an inclined angle, its tail may come into contact with the cavity wall, resulting in fluctuations in the normal acceleration.

(3) The natural frequencies corresponding to the dominant modes involved in the vibration of the vehicle shell are basically located in the buffer region of the shock response spectrum.

This study provides a referable numerical method for investigating the high-speed water entry of large-scale vehicle. Due to the limitations of current conditions, simplified modeling is adopted for the vehicle herein. Accordingly, the research conclusions cannot be directly applied to engineering design. Future work will focus on establishing refined models for large-scale vehicle to obtain results that better conform to practical conditions.

Acknowledgements

This study was funded by the Fundamental Research Funds for the Central Universities (Grant No. 2026KY07002) and the National Natural Science Foundation of China (Grant No. U2341285), and sponsored by Innovation Foundation for Doctor Dissertation of Northwestern Polytechnical University.

References

- [1] He C.T., Wang C., He Q.K., and Qiu Y. Low speed water-entry of cylindrical projectile(in chinese). *Acta Phys. Sin.*, 61(13):281–288, 2012.
- [2] Lu L.R., Wei Y.J., Wang C., and Song W. C. Experimental investigation into the cavity and ballistic characteristics of low-speed oblique water entry of revolution body(in chinese). *Acta Armamentarii*, 39(7):1364–1371, 2018.
- [3] Mirzaei M., Taghvaei H., and Golneshan A.A. Improvement of cavity shape modeling in water-entry of circular cylinders by considering the cavity memory effect. *Appl. Ocean Res.*, 97:102073, 2020.
- [4] Jiao W.P., Huang Z.G., Chen Z.H., Liu X.Y., Wang H., and Zhou K. Trajectory model of high-speed water entry of the body of revolution(in chinese). *Equip. Environ. Eng.*, 19(5):30–38, 2022.
- [5] Wang Y.F., Ye B.S., Wang Z.Y., Huang J., Wang Y.W., and Huang C.G. Vertical water entry of projectiles with surface seal. *Ocean Eng.*, 216:107606, 2020.
- [6] Chang Y.N. and Tong A.Y. A numerical study on water entry of cylindrical projectiles. *Phys. Fluids*, 33(9):093304, 2021.
- [7] Von Karman T.H. The impact on seaplane floats during landing. Technical report, NACA, 1929.
- [8] Wagner H. Phenomena associated with impacts and sliding on liquid surfaces. *Appl. Math. Mech.*, 12(4):193–215, 1932.
- [9] Oliver J.M. Second-order wagner theory for two-dimensional water-entry problems at small deadrise angles. *J. Fluid Mech.*, 572:59–85, 2007.
- [10] Kou Y., Wang B.S., and Chen W.Q. A study of theoretical method for two-dimensional water impact problems(in chinese). *J. Ship Mech.*, 21(4):383–389, 2017.
- [11] Sun T.Z., Zhou L., Yin Z.H., and Zong Z. Cavitation bubble dynamics and structural loads of high-speed water entry of a cylinder using fluid-structure interaction method. *Appl. Ocean Res.*, 101:102285, 2020.
- [12] Wang Z., Wu M.L., and Dai W.L. Study on load characteristics of high-speed water-entry of large caliber projectile(in chinese). *J. Ballist.*, 32(1):15–22, 2020.
- [13] Shi Y., Liu Z.P., Zhao H.R., Pan G., and Qin D.H. Numerical study on load and deformation characteristics of vehicle during high-speed water entry. *J. Fluids Struct.*, 141:104473, 2026.