Numerical study of water entry supercavitating flow around a vertical circular cylinder influenced by turbulent drag-reducing additives

C X Jiang, J P Cheng and F C Li
School of Energy Science and Engineering, Harbin Institute of Technology, Harbin 150001, China

E-mail: lifch@hit.edu.cn

Abstract. This paper attempts to introduce a numerical simulation procedure to simulate water-entry problems influenced by turbulent drag-reducing additives in a viscous incompressible medium. Firstly we performed a numerical investigation on water-entry supercavities in water and turbulent drag-reducing solution at the impact velocity of 28.4 m/s to confirm the accuracy of the numerical method. Based on the verification, projectile entering water and turbulent drag-reducing solution at relatively high velocity of 142.7 m/s (phase transition is considered) is simulated. The cross viscosity equation was adopted to represent the shear-thinning characteristic of aqueous solution of drag-reducing additives. The configuration and dynamic characteristics of water entry supercavity, flow resistance were discussed respectively. It was obtained that the numerical simulation results are in consistence with experimental data. Numerical results show that the supercavity length in drag-reducing solution is larger than one in water and the velocity attenuates faster at high velocity than at low velocity; the influence of drag-reducing solution is more obvious at high impact velocity. Turbulent drag-reducing additives have the great potential for enhancement of supercavity.

1. Introduction
The water-entry phenomenon, intrinsically a transient process, is an important topic in naval hydrodynamic sector. Scientific research of water entry phenomenon initiated from the beginning of the 19th century [1, 2]. With the development of CFD techniques, numerical simulation has become a reliable and appropriate tool for the investigation of this complex unsteady nonlinear flow. Ref. [3-6] presented numerical solutions on water-entry problems.

In the research formerly, our work has obtained important discovery that at the same initial velocity: the size of supercavity in drag-reducing solution was larger than that in water and maintaining downstream distance of supercavity is longer in surfactant solution as well [7]. In order to have a deep insight into the mechanism of water-entry supercavitating flow influenced by turbulent drag-reducing activities, we first simulated projectile downward entering drag-reducing solution in moderate velocity and compared with the experimental data to confirm the accuracy of the numerical method employed. And then we make further efforts to numerically investigate water-entry supercavitating flow around a

1 Corresponding author.
circular cylinder influenced by turbulent drag-reducing additives at high impact velocity and in which phase transition is considered.

2. Numerical simulation procedures

1.1. Governing equations
The volume of fluid (VOF) method is taken in this study to capture the interface between phases. The standard \( k-\varepsilon \) model with standard wall functions was adopted to provide turbulence closure. Schnerr-Sauer cavitation model [8] is adopted to deal with phase transition.

1.2. Cross viscosity model
The Cross viscosity model [9] was utilized herein to characterize the shear-thinning turbulent drag-reducing solution. The drag-reducing solution was characterized through modeling the dynamic viscosity by the Cross viscosity equation:

\[
\mu = \mu_0[1 + (\gamma \lambda)^m]^{1/m}.
\]

Where \( \lambda = 0.4 \mu_0 M \mu / R_c T \), \( \mu_0 \) is zero shear viscosity; \( \lambda \) is relaxation time; \( \gamma \) is shear rate; \( m \) is Cross power index and \( m = 0.3 \) was utilized here; \( M \) is the mean mole mass of the solute in g/mol; \( \mu \) is viscosity of solvent; \( c \) is the solution concentration in g/cm\(^3\); \( R_c = 8.31 \text{ J/mol-K} \); \( T \) is the absolute temperature in K; \( [\mu] \) is intrinsic viscosity and \( [\mu] = 1.03 \times 10^{-5} M^{0.78} \) in ml/g. Together with the Cross equation, a decreased surface tension coefficient was used to characterize the drag-reducing solution. \( \sigma_s = 0.036 \text{ N/m} \) (half of that of water) was chosen for the solution [10].

1.3. Numerical method and conditions
The simulation model was then simplified to be 2-Dimensional axially symmetric, as shown in Figure 1. Figure 1 (a) and (b) are the mesh systems for lower initial velocity and higher initial velocity, respectively. The User Defined Functions embedded in the FLUENT platform were used to control the movement of the projectile.

First, numerical simulations were performed for water and surfactant solution (\( \mu_0 = 0.01 \text{ Pa.s}, \lambda = 1 \text{ s}, m = 0.3 \) and \( \sigma_s = 0.036 \text{ N/m} \)) flow cases at the initial velocity \( V_0 = 28.4 \text{ m/s} \), respectively. In these cases, only liquid phase and air phase are considered. Based on the first step, we moved on to the typically high impact velocity \( V_0 = 142.7 \text{ m/s} \) in water and drag-reducing solution, and phase transition is added to this part.

![Figure 1](image)

Figure 1. Physical model, computational region and grid system.

3. Results and discussion
3.1 Lower impact velocity of $V_0 = 28.4$ m/s

In the discussion of this section, the experimental data and analytical calculation used for the comparisons with the numerical simulation are all 1000 ppm in concentration from Ref. [7].

Figure 2 shows a comparison of the supercavity forming process of water entry between numerical simulation and experiments for the 1000 ppm CTAC solution case. The significant phenomenon is that, shortly after the impact of projectile on the liquid surface, upward moving jetting and a laterally expanding splash are formed above the surface, and the projectile is suddenly surrounded by the splash (see Figure 2 (a)). As shown in Figure 2 (b) for the “open-cavity phase”: the flow separates from the projectile nose with the generation of the cavity; air rushes in from above the liquid surface to fill the cavity. In Figure 2 (c), it shows that the splash is forming a dome and closing over the entry point of the projectile to seal the cavity from the air above. As shown in Figure 2 (d), “surface closure” occurs and the cavity is then pushed down from the liquid surface. From this stage, the cavity has closed above the liquid surface and envelopes the projectile, forming the supercavity (see Figure 2 (e)). In the process of water-entry, the shape of the cavity in solution is consistent well with the experimental observations. It demonstrates that the presently conducted numerical simulation procedure is suitable for simulating the supercavitating projectile in drag-reducing solutions.

The numerical simulation results for the variation of projectile velocity as compared with the analytical predictions are shown in Figure 3. It can be seen that, for water-entry supercavitation at initial velocity $V_0 = 28.4$ m/s, the drag obtained from the numerical simulation is comparable to that predicted using the analytical model, with the maximum discrepancy of 25%. In consideration of the impact process, because the projectile is launched at some distance above the water surface, when the projectile impacts the water surface, it will lose more energy than the analytical formulation. From the numerical simulation results, we can see that the velocity in solution decays slower than in water, which means that, at the same instant, the projectile in solution flies faster than in water.
The numerical simulation results compared to the predicted penetration distance based on the analytical formulation and the experiment data are shown in Figure 4. It is seen that the projectile in solution moves in longer distance than in water at the same instant. Actually, the projectile cannot move as far as the analytical calculation due to the loss of projectile kinetic energy at the water-entry process. From the results mentioned above, it is clear that the numerical simulation approach developed in the present study allows one to investigate the characteristics of water-entry induced supercavitation influenced by turbulent drag-reducing additives.

Figure 5 plots drag coefficients versus time for both water and drag-reducing solution cases. The drag coefficient $C_d$ is defined below: $C_d = F / 0.5 \rho V^2 A_o$, where $F$ is the overall drag force. It can be seen that the drag coefficients rise sharply when the projectile impacts the liquid surface. The drag coefficient is obviously smaller in drag-reducing solutions than that in water. From these results, it can be conjectured that adding small amount of drag-reducing additives, projectile can move faster in the drag-reducing solution than in water provided the driving power is the same. It is expected that the actual drag coefficient would be greater than that predicted for a disk cavitator because there are drag forces on the projectile body.

3.2. Higher impact velocity of $V_0 = 142.7$ m/s

In this part we discuss the unsteady behavior of water-entry supercavitating flows in water and influenced by turbulent drag-reducing additives at the impact velocity of $V_0 = 142.7$ m/s. The experimental data and analytical calculation used for the comparisons with the numerical simulation are all from Ref. [11].

The numerical simulation results for the variation of projectile velocity in water as compared with the analytical predictions and experimental data are shown in Figure 6 (a). It can be seen that, for
water-entry supercavitation at initial velocity $V_0 = 142.7 \text{ m/s}$, the velocity attenuation obtained from the numerical simulation is comparable to that predicted using the analytical model and experimental data. It can demonstrate the accuracy of the numerical method for high velocity water entry with phase transition. Figure 6 (b) gives the projectile velocity attenuation in water and drag-reducing solution. From the numerical simulation results, we can see that the velocity in solution decays slower than in water, which is the same as lower impact velocity. Again, it is proved that the projectile moves faster in drag-reducing solution than in water, no matter the impact velocity is high or not. Compared with Figure 3, it can be seen that the velocity attenuates faster at high velocity than at low velocity. Meanwhile, when $t=4 \text{ ms}$, the velocity difference ($\Delta V$) between water and solution at $V_0 = 28.4 \text{ m/s}$ is $2.7 \text{ m/s}$, while at $V_0 = 142.7 \text{ m/s}$, $\Delta V = 3.3 \text{ m/s}$. So it can be conclude that the influence of drag-reducing solution is more obvious at high impact velocity.

The numerical simulation results compared to the predicted penetration distance based on the analytical formulation and the experiment data are shown in Figure 7. It is seen that the projectile in solution moves in longer distance than in water at the same instant. From the results mentioned above, it is clear that the numerical simulation approach developed in the present study allows one to investigate the characteristics of water-entry induced supercavitation considering phase transition influenced by turbulent drag-reducing additives.

Figure 7. Comparison of experiments, analytical and numerical simulation for penetration distance.

Figure 8. Comparisons of cavity configurations between water and drag-reducing solution.

Figure 8 shows a comparison of the supercavity forming process of the projectile entering water and drag-reducing solution. In consideration of the phase transition from water to vapor, we use the contour of the liquid phase to represent the cavity shape, in other words, the region under water where there is no water represents the cavity. At $t=1 \text{ ms}$, the free surface is already closed in water, meanwhile owning to the surface closure a jet forms towards air; nevertheless, the cavity is still open to the air in drag-reducing solution, and the jet is towards the cavity. At $t=2 \text{ ms}$, the reentrant jet forms in water and still, the intensity of the jet is weaker than that in solution and the aft part of the cavity is cupped in solution. Paying more attention to the process from $t=2 \text{ ms}$ to $t=3 \text{ ms}$ in solution, we’ll find that the surface closure is different. The actual process is that the closure is disturbed after $t=2 \text{ ms}$, and more air rushes into the cavity and then the cavity closed again, so the cavity becomes larger. From this process analyzed, the cavity does not close at one time, but first closes and then is disturbed and closes again. The closure process is periodic.

Figure 9 shows the components inside the cavity in water and in drag-reducing solution. It can be seen that the cavity is composed of vapor and air. The vapor is mainly distributed in the aft part and near the cavity boundary, and some other is behind the projectile. When the projectile impacts on the free surface, the air first rushes into the cavity, so most part of the cavity is immediately filled with air. With the movement of the projectile, the pressure inside the cavity decreases, due to the existence of
the air, the vapor cannot form in the center of the cavity. As the result, the vapor transformed from water first generates in the cavity boundary in the aft part and a vapor film is formed in the front part of the cavity. And then the vapor region becomes larger. After the formation of the reentrant jet into the cavity, the low pressure transforms the liquid jet to the vapor; this is where the vapor behind the projectile comes from. Comparing the vapor region in water and in solution, it can be seen that the vapor region in solution is larger than that in water, which indicates that the drag-reducing solution enhances the formation of vapor.

![Figure 9](image-url) The components inside the cavity in water and in drag-reducing solution. (a) vapor component, (b) air component.

4. Conclusions
This paper introduced a numerical simulation procedure to simulate water-entry supercavitating flow influenced by drag-reducing solution at typically low velocity and high velocity. The configuration and dynamic characteristics of water entry supercavity, flow resistance were discussed respectively. It was obtained that the numerical simulation results are in consistence with experimental data; the numerical procedure proposed is believable. Detailed conclusions are as follows:

1) The projectile moves faster in solution than in water.

2) At higher impact velocity, the vapor is mainly distributed in the aft part and near the cavity boundary, and some other is behind the projectile. The drag-reducing solution enhances the formation of vapor. The actual cavity closure process is that the cavity does not close at one time, but first closes and then is disturbed and closes again. The closure process is periodic.

3) The velocity attenuates faster at high velocity than at low velocity. And it can be conclude that the influence of drag-reducing solution is more obvious at high impact velocity.

Acknowledgments
This work is supported by Foundation for Innovative Research Groups of the National Natural Science Foundation of China (51121004), National Natural Science Foundation of China (51276046), Specialized Research Fund for the Doctoral Program of Higher Education of China (No. 20112302110020).

References
[1] Worthington A M and Cole R S 1897 *Philos. Trans. R. Soc. London, Ser. A* 189 137-48
[2] Worthington A M and Cole R S 1900 *Philos. Trans. R. Soc. London, Ser. A* 194 175-99
[3] Panahi R 2012 *J. Theor. Appl. Mech.* 42 (2) 79-92
[4] Kim Y W, Kim Y, Liu Y M and Yue D 2007 *Proc. of 9th Int. Conf. on Numerical Ship Hydrodynamics* (Michigan, USA, 5-8 Aug 2007)
[5] Walhorn E, Kö lke A, Hübner B and Dinkler D 2005 *Comput. Struct.* 83 2100-11
[6] Yang Q and Qiu W 2012 *J. Offshore. Mech. Arct. Eng.* 134 041101-1
[7] Jiang C-X and Li F-C 2014 *Adv. Mech. Eng.* 2014 280643
[8] Schnerr G H and Sauer J 2001 *4th Int. Conf. on Multiphase Flow* New Orleans USA
[9] Bird R B, Curtiss C F, Armstrong R C and Hassager O 1987 *Dynamics of Polymeric Liquids, Fluid Mechanics* Volume 1 (John Wiley & Sons)
[10] Li F C, Dong Y, Kawaguchi Y and Oshima M 2008 *Exp. Therm. Fluid. Sci.* 33 161-8
[11] Zhang W, Guo Z T, Xiao X K and Wang C 2011 *Explo. Shock. Wav.* 31(6) 579-84