Natural Cavitation in High Speed Water Entry Process

Yadong Wang
College of Marine
Northwestern Polytechnical University
Xi'an, China，(86)13572124326
roby868@163.com

Xulong Yuan
College of Marine
Northwestern Polytechnical University
Xi'an, China，(86)18691859360
yuanxulong@nwpu.edu.cn

Yuwen Zhang
College of Marine
Northwestern Polytechnical University
Xi'an, China，(86)13609110453

***Abstract*—Phase transition between liquid and vapor comes into existence when the velocity of the water entry device is sufficiently high, which will affect the flow field formation and development. Numerical simulations, implemented in the commercial CFD package FLUENT, with three popular cavitation models and some equation modifications by UDF, have been conducted to investigate the natural cavitation in high speed water entry process. Comparisons with experimental results were done to verify the applicability of each model. With the preferable model, the formation and development of natural cavitation is presented, with the pressures inside water entry cavity analyzed. Cases with three water entry velocity magnitudes were simulated specially to study its influence to natural cavitation and water entry process. The results show an order of the natural cavitation in high speed water entry process, and provide an effective method to simulate such problems.**

Keywords-high speed water entry; cavitation model; multiphase flow;natural cavity; pressure feature

I. INTRODUCTION

With the development of naval weapons, the problems encountered in the water entry process have become more notable, especially when the speed adds up to a very high level. The traditional research field of water entry focuses on the structure related problems[1,2], namely the impacting force. Usually it takes account of the water liquid phase and the air phase to construct the flow field of water entry, neglecting the potential natural cavitation. But unfortunately when the water entry speed becomes high enough, part of the body pressure distribution will drop below the saturation pressure, natural cavity starts to occur[3], which will change the mechanism of flow field formation. Especially, when the water entry problem of supercavitation projectile is studied, the formation of both the natural cavity and the air cavity is the main concern, and the interactions between phases become significant.

Traditionally, the water entry research is done to guide the design of air drop underwater weapons, so it mainly solves the problem of phase changing during its flight, but avoids the adverse affects of phase transition. However, we have to use the cavitation phenomenon in some applications, such as supercavitation vehicles. The natural cavitation tends to be a dominant role in water entry cavity for some devices with ultra high water entry speed. To sum up, it becomes extremely important to study the transition and interaction of different phases in high speed cases.

The problem contains three phases, we have to track the interfaces between different phases and meanwhile model the transition of liquid to vapor. Existing research approaches show that it is better to use the VOF method in free surface cases, but the homogeneous equilibrium flow method is popular due to its computationally inexpensive nature when modeling cavitaion[4]. For our problem, we focus on the multiphase flow field and especially the effects of phase transition, so we prefer to use the latter method mentioned previously, and use some techniques in meshing the computational domain to avoid the adverse aspects induced by the method.

In the scope of water entry related problems, various theory investigations and experiments have been carried out. Hydrodynamic load during water entry process is calculated using different methods[5,6]. Experimental investigations of different objects are done to aid modeling and verifying each theory[7,8]. Their investigations mainly concentrate on low speed water entry problem, where natural cavitation is beyond their consideration. For high speed cases, a supercavitation projectile water entry simulation was done by Michael[3], using the Kunz cavitation model. Many other cavitation models, such as Singhal et al Model[9], Schnerr-Sauer Model[10] and Zwart-Gerber-Belamri Model[11], have been set up to study cavitation problems of turbine machines and underwater vehicles, showing good accordance with experimental data. However, the successful applications of these models are for submerged devices, the applicability in water entry should be studied.

In this paper, the water/vapor/air multiphase model with three cavitation models was set up to simulate the water entry process, concentrating on the natural cavitation appeared. A comment on the applicability of cavitation models was made through comparisons of numerical with experimental results. The formation and development of multiphase flow field composed by three phases were specially discussed with figures and pressure curves. The influence of water entry speed to natural cavity characteristics were studied with further simulations, showing the necessity of vapor phase in high speed water entry investigations.

II. NUMERICAL PROCEDURE

A. Multiphase Model

The multiphase flow is modeled by the popular Mixture model. It uses a single-fluid approach, and allows the phases to be interpenetrating. The Mixture model solves the continuity equation for the mixture, the momentum equation for the mixture, the energy equation for the mixture, and the volume fraction equation for the secondary phases.

The continuity equation and the momentum equation for the mixture can be seen in [12].

The energy equation is

$$\frac{\partial}{\partial t}\sum_{i=1}^{3}(\alpha_i\rho_i E_i)+\nabla\bullet\sum_{i=1}^{3}(\alpha_i\vec{v}_i(\rho_i E_i+p))=\nabla\bullet\left(k_{ef}\nabla T\right)+S_E \quad (1)$$

where $k_{ef}=(\sum\alpha_i(k_i+k_t))$, k_t is the turbulent thermal conductivity. S_E is the user defined source term, for our case, $S_E=-r(R_e-R_c)$, r is the local specific latent heat.

In this paper, water liquid is selected to be the primary phase, water vapor and air as the secondary phases.

B. Compressible Effects

As the body moving velocity in air is about 0.3 Mach or above, compressible effects must be taken into account. All the gaseous phases are modeled as ideal gas.

$$P=\rho_g RT \quad (2)$$

where $R=\hat{R}/\hat{M}$ is universal gas constant divided by the molar mass. Treating the gaseous phase as an ideal gas makes it simplified to calculate enthalpy, for it is only a function of temperature.

C. Cavitation Models

The widely used cavitation models, including Singhal et al Model, Schnerr-Sauer Model and Zwart-Gerber-Belamri Model were utilized in current investigation.

The differences these models lay in the expressions of mass transfer equations.

The mass transfer terms of Singhal et al model are[9]:

$$R_e=F_v\rho_l\rho_v\frac{\max(1.0,\sqrt{k})}{\sigma}(1-f_v-f_g)\sqrt{\frac{2}{3}\frac{(P_v-P)}{\rho_l}} \quad (3)$$

$$R_c=F_c\rho_l\rho_v\frac{\max(1.0,\sqrt{k})}{\sigma}f_v\sqrt{\frac{2}{3}\frac{(P-P_v)}{\rho_l}} \quad (4)$$

where $F_v=0.02$ and $F_c=0.01$.

In the Schnerr-Sauer Model, they are[10]:

$$R_e=\frac{\rho_v\rho_l}{\rho}\alpha(1-\alpha)\frac{3}{R_B}\sqrt{\frac{2}{3}\frac{(P_v-P)}{\rho_l}} \quad (5)$$

$$R_c=\frac{\rho_v\rho_l}{\rho}\alpha(1-\alpha)\frac{3}{R_B}\sqrt{\frac{2}{3}\frac{(P-P_v)}{\rho_l}} \quad (6)$$

The expressions in Zwart-Gerber-Belamri Model are[11]:

$$R_e=F_{vap}\frac{3\alpha_{nuc}(1-\alpha_v)}{R_B}\sqrt{\frac{2}{3}\frac{(P_v-P)}{\rho_l}} \quad (7)$$

$$R_c=F_{cond}\frac{3\alpha_v\rho_v}{R_B}\sqrt{\frac{2}{3}\frac{(P-P_v)}{\rho_l}} \quad (8)$$

where $R_B=10^{-6},\alpha_{nuc}=5\times10^{-4},F_{vap}=50,F_{cond}=0.001$.

D. Turbulence Model

Three phase interactions are calculated here in this problem, and the huge density ratio between phases becomes a main difficulty in carrying out such simulations. To solve this, a modified RNG k-ε turbulence model is employed in this paper. The modified model can artificially restrict the turbulent viscosity in the mixture area[13], so it is proper to describe the unsteady cavitation flow during the water entry process, the form of turbulent viscosity is

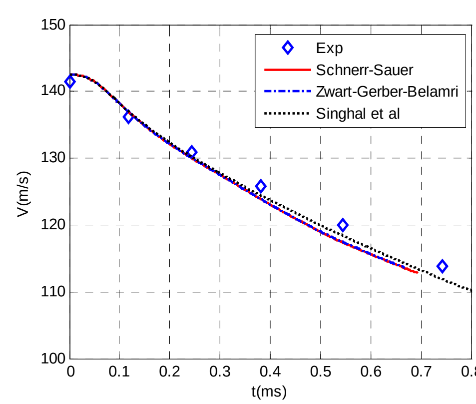
$$\mu_t=f(\rho)C_\mu\frac{k^2}{\varepsilon} \quad (9)$$

$$f(\rho)=\rho_v+(1-\alpha)^n(\rho_l-\rho_v),n>1 \quad (10)$$

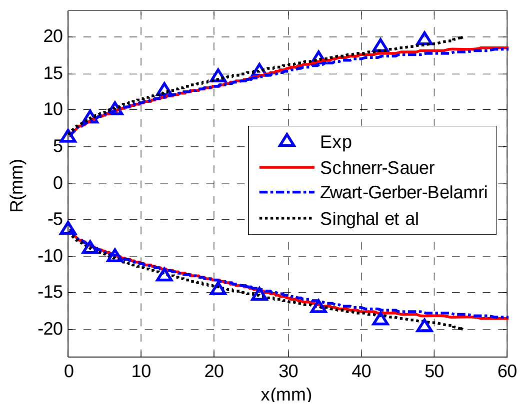
According the conclusion of previous work[4], n is set to 10 in this article.

III. VALIDATION OF CAVITATION MODELS

First, a series of simulations for validation were done, using the test model of experimental work by Zhang[14]. The test model is a 25.4×Φ12.65mm cylinder, the weight is 22.52g, and the selected initial velocity is 142.7m/s. It is difficult to mark out the volume fraction of natural cavitation in water entry cavity through experimental work, and indeed in their paper, they only provided the images of water entry cavities and speed curve. The formation and development of natural cavitation is one of the causes of these parameters, so they can still be used to verify the applicability of cavitation models.



a) Curve of velocity vs. time



b) Water entry cavity profile (t=0.5 ms)

Figure 1. Comparsions with experimental results by Zhang et al[14]

Figure 1 shows the comparison of simulations utilizing the mentioned three cavitation models with experimental result. Through the comparison, we can see that all these models present a nearly same result with experiment, but it appears that the Singhal et al model predicts a better cavity profile than the other models, and the velocity drop agrees better, which means the force feature it obtains is more reliable. So in our further investigation, we choose the Singhal et al cavitation model.

IV. NATRAL CAVITATION DEVELOPMENT

A. Model Configuration

In the further study, we choose another model which is a cylinder with a higher slenderness ratio, with size 120×Φ20mm, 0.292kg in weight. Utilizing this model, a series of simulations were carried out to investigation the multiphase flow field development and effect of initial speed.

To study the pressure distribution along water entry object and also the pressure inside cavity during the water entry process, an array of pressure monitors were setup along body surface, as shown in Figure 2. The distance of P1 to nose is 10mm, and the gap between each adjacent monitor is 10mm.

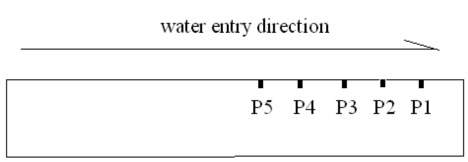


Figure 2. Pressure monitor configuration

Three velocity magnitudes were chosen in these simulations, 50m/s, 100m/s and 150m/s, and 100m/s is considered to be the typical speed.

B. Flow Field Development

Details of the multiphase flow field formation will be discussed here, especially the development of natural cavitation. The typical water entry speed of 100m/s case is addressed herein. Figure 3 is a series of air phase contours in different depth points of water entry process, the accurate water entry depths of the pictures are separately 39.9mm, 49.7mm, 59.5mm, 69.3mm, 79.2mm and 117mm.

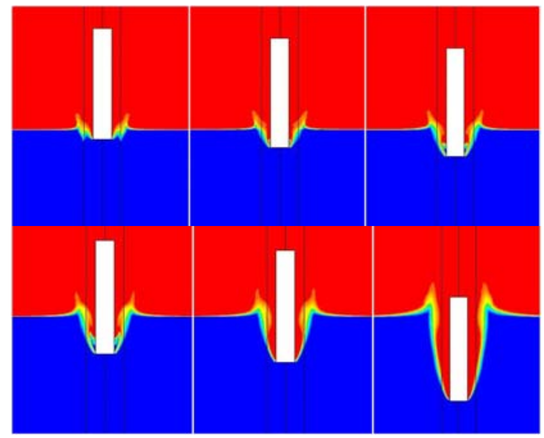


Figure 3. Contour of air at different stages

At the depth of 39.9mm, the dropping body has contacted with water surface, and the water begins to move away. Vapor comes into existence since 49.7mm, and small cavity starts to form in the body shoulder. The size of vapor cavity grows up at 59.5mm and 69.3mm, but it collapses at 79.2mm due to pressure increasing. The body penetrates into the water surface wholly at 117mm, surrounded by a large air cavity.

The development of multiphase flow field, the formation and collapse of natural cavity can also be judged by pressure distribution in body surface as shown in Figure 4. It can be seen in this figure that for a specified depth, pressure inside the cavity are not equal, pressures near body shoulder tend to be much lower. The drop magnitude of pressure is much lower than $0.5\rho_{air}v^2$, which is the air dynamic pressure drop according to Bernoulli equation. That means body shoulder contacts with liquid, and then the pressure drop below the saturation pressure (2367Pa for our simulations), phase transition comes into existence. P1 begins to suffer pressure 2372Pa in 50mm depth, and it continues to about 75mm, which is consistent with previous conclusion of natural cavitation formation and collapse. The inner cavity pressures illustrate the development of multiphase flow field in a more specific way.

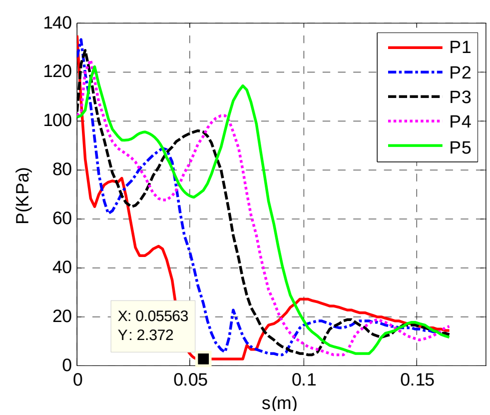


Figure 4. Pressures vs. time of 100m/s

C. Effect of Water Entry Speed

Various speed cases are set to study the effect of water entry speed to the three phase flow development. Results are shown in Figure 5 in order of 50m/s, 100m/s and 150m/s.

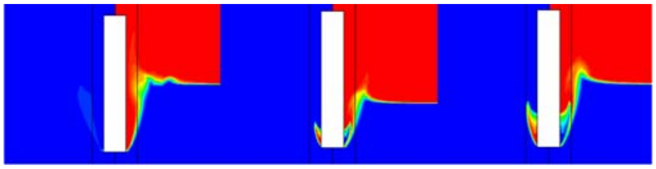


Figure 5. Profiles of phase interface (depths at the maximal natural cavity)

The left side of each picture is the contour of vapor phase, and the right side is the contour of air phase, showing the weight of each gaseous phase and the water entry cavity profile. It is quite clear that the fraction of vapor phase arises with speed, making it more important in the formation of flow. The peak size of vapor cavity appears in different penetrating depths. The results also indicate that for the model we investigated, if the speeds equal 50m/s or less, the effect of vapor phase is neglectable.

More specific information can be obtained if we examine the pressures for each case as shown in Figure 6, Figure 4 and Figure 7.

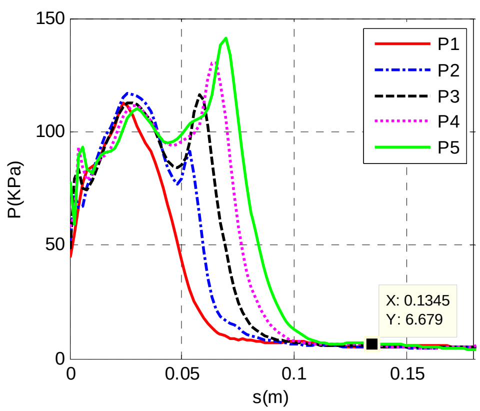


Figure 6. Pressures vs. time of 50m/s

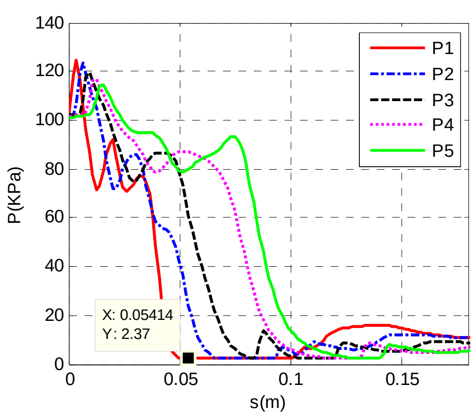


Figure 7. Pressures vs. time of 150m/s

All pressures of 50m/s case have not drop to the saturation pressure, which shows there is no macroscopical natural cavity. Pressure features for 150m/s case are mostly same with the 100m/s case, for both of them have pressures bellow saturation pressures, and the subsequent variations due to natural cavity collapse are nearly same. However, the wider low pressure region of 150m/s case indicates a longer natural cavity.

V. CONCLUSION

The main conclusions for can be drawn as:

1) Applied to simulate natural cavitation in high speed water entry process, Singhal et al cavitation Model predicts better cavity profile and force features than Schnerr-Sauer Model and Zwart-Gerber-Belamri Model.

2) Pressure inside water entry cavity are not necessarily equal everywhere, and the low pressure in the body shoulder region allows for the appearance of natural cavitation.

3) The natural cavitation volume fraction adds with water entry speed. Pressure features are mostly same for cases with natural cavitation, but if velocity is less than 50m/s for the model studied in this paper, the effect of vapor phase can be neglected.

REFERENCES

[1] WANG Yong-hu and SHI Xiu-hua, “Review on research and development of water-entry impact problem,” Explosion and Shock Waves,vol 28, pp. 276-282, 2008. (In Chinese)

[2] QIN Hong-de, ZHAO Lin-yue and SHEN Jing, “Review of water entry problem,” Journal of Harbin Institute of Technology, vol 43, pp. 152-157, 2011. (In Chinese)

[3] Michael Dean Neaves, “Time-accurate calculations of axisymmetric water entry for a supercavitating projectile,” 34th AIAA Fluid Dynamics Conference and Exhibit, Oregon, Portland, 2004.

[4] Zhao Wei-guo, ZHANG Ling-xin and SHAO Xue-ming, “Numerical simulation of cavitation flow under high pressure and temperature,” Journal of Hydrodynamics Ser. B, vol 23, pp. 289-294, 2011.

[5] A.A. Korobkin and A. Iafrati, “Hydrodynamic loads during initial stage of floating body impact,” Journal of Fluids and Structure,vol 21, pp. 413-427, 2005.

[6] D. Battistin and A. Iafrati, “Hydrodynamic loads during water entry of two-dimensional and axisymmetric bodies,” Journal of Fluids and Structure, vol 17, pp. 643-664, 2003.

[7] Jeffrey M. Aristoff and John W.M. Bush, “Water entry of small hydrophobic spheres,” J. Fluid Mech., vol 619, pp. 45-78, 2009.

[8] Tadd T. Truscott and Alexandra H. Techet, “Water entry of spinning spheres,” J. Fluid Mech., vol 625, pp. 135-165, 2009.

[9] Singhal A.K., Athavale M., LI H.Y. and Y. J., “Mathematical basis and validation of the full cavitation model,” J. Fluid Eng., vol 124, pp. 617-624, 2002.

[10] G.H. Schnerr and J.Sauer, “Physical and Numerical Modeling of Unsteady Cavitation Dynamics,” Fourth International Conference on Multiphase Flow, New Orleans, USA, 2001.

[11] P.J. Zwart, A.G. Gerber, and T.Belamri, “A Two-Phase Flow Model for Predicting Cavitation Dynamics,” Fifth International Conference on Multiphase Flow, Yokohama, Japan, 2004.

[12] ANSYS Inc, FLUENT 6.3 user guide. Pennsylvania: ANSYS Inc. 2006.

[13] CHEN Ying, LU Chuan-jing and WU Lei, “Modeling and computation of unsteady turbulent cavitation flows,” Journal of Hydrodynamics Ser. B, vol 18, pp. 559-566, 2006.

[14] ZHANG Wei, GUO Zi-tao, XIAO Xin-ke and WANG Cong, “Experimental investigations on behaviors of projectile high-speed water entry,” Explosion and Shock Waves, vol 31, pp. 579-584, 2011. (In Chinese)