CAV2021
11th International Symposium on Cavitation
May 10-13, 2021, Daejeon, Korea

Multiphase Flows Analysis of an Axisymmetric Body Using Improved Interface Capturing Method with a Cavitation Model

Trong-Nguyen Duy[1], Van-Tu Nguyen[1], Thanh-Hoang Phan[1], Quang-Thai Nguyen[1], Warn-Gyu Park [1*]

[1]School of Mechanical Engineering, Pusan National University, Busan, Korea

Abstract: The objective of this study is to examine the flow field surrounding a cylinder with different cavitator shapes to validate the accuracy of a newly developed solver based on an open-source platform OpenFOAM. The developed solver is based on an improvement to a coupling technique for interface simulation, and an enhancement of the Merkel cavitation model for estimation of the phase change rate. The cavity shape is sharpened and smoothed based on the utilization of the continuity of the Level-Set (LS) function while the mass of mixture flow in the interface cells is maintained based on the VOF approach of a coupling technique. The dependency on the empirical constant of the phase change rate of the Merkle cavitation model is significantly reduced based on an improvement in which the evaporation rate is proportional to the vapor density. The proposed methods are then integrated into an incompressible Navier-Stokes solver of OpenFOAM platform to simulate flow fields around asymmetric bodies. Good agreement between the simulated results and the measured data demonstrates the capability of the proposed method in simulations of practical cavitating flows.

Keywords: Coupling Level-Set and VOF; PNU cavitation model; OpenFOAM.

1. Introduction

Cavitation is a common phenomenon in fluid mechanics, it happens when the pressure in some region of liquid flow drops below the saturation pressure. The liquid is then vaporized and replaced by a cavity consequently. This phenomenon has negative effects on the performance of many fluid machinery application such as marine propeller, turbomachinery's impeller, hydrofoils, nozzles, or torpedoes, etc., Existence of a cavity when devices such as propellers, impellers, hydrofoils are in operating condition can cause severe noise, vibration, and erosion which have a negative influence their performance. However, in some other applications such as high-speed ships (trimaran, catamaran), torpedoes, cavitating flow can be utilized as an air buffer to minimize resistance caused by fluid flow acting on the body's surface. The cavitating flow is a complex phenomenon and has not been completely modeled. Accurate simulation of cavitating flows is essential for fully understanding its mechanism. For accurately simulating the cavitating flow, it is crucial to establish a proper mathematical model that describes well the phase change process between liquid and vapor. Various cavitation models have been proposed in the literature [1-5]. These models can be categorized into two groups, i.e., the barotropic law model and the transport equation-based model, in which the transport equation-based model is widely employed since the flow can be considered as a single fluid. The constituted phases can be modeled as an incompressible flow, and the mass transfer can happen between phases. Merkle et al., [2] presented a cavitation model in which the mass transfer rate is proportional to the dynamic pressure of the liquid flow. Empirical constants were employed to evaluate the condensed and evaporated rates corresponding to the evaporation and condensation processes. Kunz et al., [1] improved the condensation term source terms based on Merkle's research work. Schnerr and Sauer [4] produced a cavitation model based on the simplified Rayleigh equation which does not depend on the empirical constants. Because the empirical coefficients have a strong impact on the solution of flow fields, optimization on phase change model in such a way that dependency of the cavity shape and

* Corresponding Author: Warn-Gyu Park, wgpark@pusan.ac.kr

flow field distribution on empirical constant is reduced is essential. An enhancement of Merkle's model has been made at Pusan National University (PNU) in which the dependency on the empirical constant of the phase change rate is significantly reduced [3, 6, 7]. The objective of the present work is to evaluate the capability of a newly developed solver based on PNU cavitation model for incompressible fluid flow using an open-source platform of OpenFOAM through an examination of the cavitating flow past a cylinder with different cavitator configurations. To effectively capture the cavity shape, a coupling method that combines the advantages of mass conservation of VOF approach; smoothed and sharpen interface of Level-Set (LS) method is applied. The remainder of this paper is organized as follows: the governing Navier-Stokes equation system together with PNU cavitation model and a brief description of a coupling technique is presented in section 2. The simulated results with discussion on cavitating flow surrounding cylinders are shown in section 3. Section 4 shows some remarkable conclusions.

2. Numerical Method

2.1. Governing equations

The cavitation flow is considered as a single-fluid model of a homogeneous vapor-liquid mixture in this paper. The governing equations consist of the conservation of mass, momentum, and liquid phase fraction implemented in OpenFOAM platform are as follows [8-10]:

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \mathbf{U}) = 0 \tag{1}$$

$$\frac{\partial (\rho \mathbf{U})}{\partial t} + \nabla \cdot (\rho \mathbf{U}\mathbf{U}) = -\nabla p + \nabla \cdot \mathbf{T} + f_b \tag{2}$$

$$\frac{\partial \alpha}{\partial t} + \mathbf{U} \cdot \nabla \alpha = S_\alpha \tag{3}$$

where $\mathbf{U}$ is the velocity shared by the constituted fluids throughout the domain. $\mathbf{T}$ is the viscous tensor derived from the velocity field; f_b is the body force per mass unit which includes the effects of gravity and surface tension at the interface, and S_α stands for the phase change rate between vapor and liquid phases.

In this study, the volume transport equation of mixture flow is solved in which the location of the interface is defined based on the volume fraction function. Mixture properties (ρ, μ) of the fluid flow in the interface cells are then computed based on the phase fraction as follows:

$$\phi_m = \alpha_l \phi_l + (1 - \alpha_l)\phi_g \tag{4}$$

Due to the phase change phenomena, the density of mixture flow varies in interface cells causing velocity divergence is non-zero there. To stable solution method and precisely define the interface location, the relation between the velocity divergence and phase change rate is assumed as follow:

$$S_a = \frac{1}{\rho_l}(\dot{m}^+ + \dot{m}^-) \tag{5}$$

$$S_m = \frac{\partial u_j}{\partial x_j} = \frac{-1}{\rho_m}\frac{D\rho_m}{D_t} = \frac{\rho_l - \rho_g}{\rho_l} S_\alpha \tag{6}$$

2.2. Coupling Method

To solve the phase fraction equation (3), an algebraic approach of the VOF model with a MULES solver based on an interFoam solver of OpenFOAM platform is applied. Details of the numerical schemes and implementation of the MULES algorithm can be found in the literature [9, 11]. It is widely acknowledged that though the mass conservation property is well preserved with the VOF method, the simulated interface is not smoothed owing to the discontinuity of the phase fraction value. To improve the interface sharpness, a coupling approach that utilizes the continuous characteristic of a Conservative LS (CLS) function is applied in this study. The reason explained why the CLS function is selected, is that its variation is in the same range as the scalar function of the VOF approach. Its gradient used to estimate interface normal vector, therefore, is in the same range as with VOF approach with a

* Corresponding Author: Warn-Gyu Park, wgpark@pusan.ac.kr

complement of the smoothness. The simulated interface is then expected to more sharpen, and mass conservation is well preserved. This coupling method has been proposed and well examined in an our previous study [8]. Details of the numerical method and its implementation have been also described in that study.

2.3. Cavitation Model

In this study, an enhancement of the Merkle model [2] to decrease the dependency of the empirical coefficient is used. The rate of phase change used in equation (5) is estimated as follows [3, 6, 7]:

$$\dot{m}^{-} = -k_g \frac{\rho_g \alpha_l}{t_\infty} min\left[1, max\left(\frac{p_{Sat} - p}{k_p p_{Sat}}, 0\right)\right] \quad (7)$$

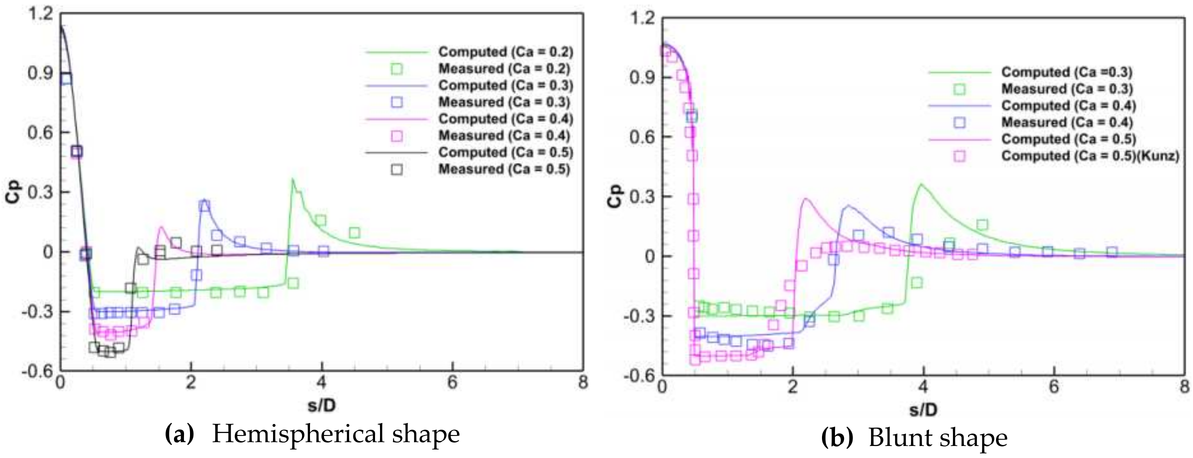
$$\dot{m}^{+} = -k_l \frac{\rho_g \alpha_g}{t_\infty} min\left[1, max\left(\frac{p - p_{Sat}}{k_p p_{Sat}}, 0\right)\right] \quad (8)$$

in which the ramping function defined by equation (9) is used to ensure the stability of the solution scheme. The factor k_p is defined as small as possible to ensure that the scaling constant is the only main parameters to control phase changes. In this work, $k_p = 0.01$ is adopted.

$$f = min\left[1, max\left(\frac{p - p_{Sat}}{k_p p_{Sat}}, 0\right)\right] \quad (9)$$

3. Results and Discussion

In this section, the simulated results of axisymmetric flow over a cylindrical body with different cavitator shapes will be presented to validate the capability of the present solver. A cylinder with hemispherical and blunt nose shape are applied in computations with different cavitation number (Ca). Figure 1 compares the computed time-averaged pressure coefficient distributions on bodies surface with available experimental/benchmark data [1]. The computed results show a good agreement regarding the measured data, especially in the case of the hemispherical shape cavitator. The difference of the computed results of pressure peak on body's surface in case of the blunt shape (Figure 1(b)) may come from the fact that estimation of the turbulent viscosity in that region is not accurate owing to the default setup of $k - \varepsilon$ model in OpenFOAM was applied in this work. In addition, the fluid compressibility and the cavitation induced turbulence effects resulting in the inaccurate physical phenomenon in the highly compressible mixture region have not considered in this study. However, it could be generally concluded that the proposed method can yield reasonably good results of pressure distribution on bodies surfaces.



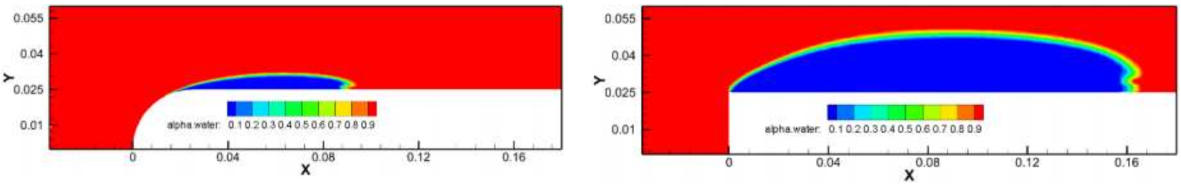
(a) Hemispherical shape **(b)** Blunt shape

Figure 1. Comparisons of time-averaged surface pressure with different cavitation number

The simulated cavity surrounding the body's surface with $Ca = 0.3$ is shown in Figure 2. The simulated results visibly depicted that the cavity shape is relatively smooth and sharp. The formation of the cavity could be explained as follow: The fluid flow after attacking the head of the cylinder will

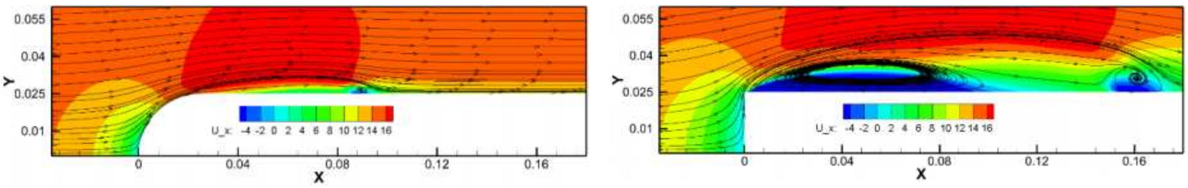
* Corresponding Author: Warn-Gyu Park, wgpark@pusan.ac.kr

pass downstream following the body's surface direction. However, owing to the change of body shape at the corner, the fluid flow could not adhere to the body surface there making a large deficit of pressure. The pressure of the fluid flow drops under the saturated point making formation of a cavity in that region. Moreover, the simulated results also indicate that the cavity shape strongly depends on the body shape. With the same cavitation number, the cavity shape and length of the blunt-body are wider and longer than those of the hemispherical cavitator. These are caused by a larger deficit of pressure in the case of the blunt shape. The sudden change of body shape at the sharp corner makes the fluid flow more divergent there. The pressure of the fluid flow, therefore, drops deeper than the saturation point in a wider range in comparison to the hemispherical ones as visibly depicted in Figure 3.



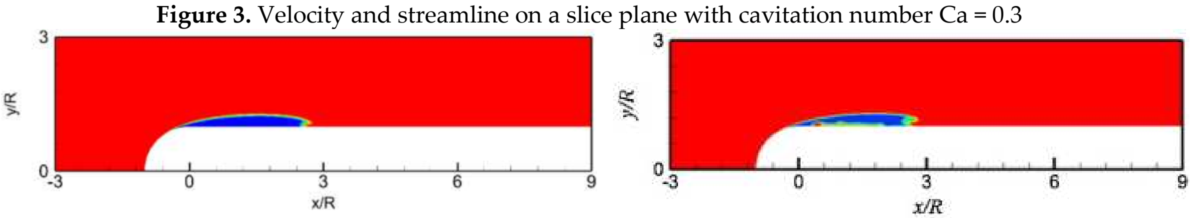
(a) Hemispherical shape **(b)** Blunt shape

Figure 2. Cavity shape over the cylinder with different cavitator shape with Ca = 0.3



(a) Hemispherical shape **(b)** Blunt shape

Figure 3. Velocity and streamline on a slice plane with cavitation number Ca = 0.3



(a) Present **(b)** Park and Ree [12]

Figure 4. Velocity and streamline on a slice plane with cavitation number Ca = 0.3

Figure 4 compares the cavity shape surrounding a hemispherical cavitator shape computed by the present method and the published data using another OpenFOAM solver conducted by Park and Ree [12]. It is visibly shown that the present solver yields a smoother and more shapen cavity shape than those of Park and Ree. This comes from the fact that the interface is defined using the coupling method in which the smoothed and bounded CLS function is employed in the solution method of interface solver as explained before.

4. Conclusions

In this paper, a cavitating flow solver based on an efficient coupling method for interface computation and an enhancement cavitating Merkle model has been developed. The developed solver inherits the advantages of the sharpened, smoothed interface of a CLS method for interface simulation while maintaining the mass conservation property of VOF approach. The shortcoming of dependency on empirical constants of the Merkle cavitation model has been also reduced by an improved one in which the condensed rate is proportional to vapor density. The proposed method is validated with benchmark cases of cavitating flow past a cylinder with hemispherical/blunt cavitator shape. Good agreement with the measured/computed data has demonstrated the capability of this solver.

* Corresponding Author: Warn-Gyu Park, wgpark@pusan.ac.kr

Acknowledgments: This research was supported by the Korea Research Fellowship program funded by the Ministry of Science and ICT through the National Research Foundation of Korea (No. 2019H1D3A1A01103008).

References

1. Kunz, R., et al. Multi-phase CFD analysis of natural and ventilated cavitation about submerged bodies. in *Proceedings of the 1999 3rd ASME/JSME Joint Fluids Engineering Conference, FEDSM'99*. **1999**. San Francisco, California, USA.
2. Merkle, C., J. Feng, and P.E.O. Buelow. Computational modeling of the dynamics of sheet cavitation. in *Proceedings of 3rd International Symposium on Cavitation*. **1998**. Grenoble, USA.
3. Park, W., C. Ha, and C. Merkle. Multiphase flow analysis of cylinder using a new cavitation model. in *7th International Symposium on Cavitation*. **2009**. Ann Arbor, MI, USA.
4. Schnerr, G. and J. Sauer. Physical and numerical modeling of unsteady cavitation dynamics. in *ICMF-2001, 4th International Conference on Multiphase Flow*. **2001**. New Orleans, USA.
5. Senocak, I. and W. Shyy, Interfacial dynamics-based modelling of turbulent cavitating flows, Part-1: Model development and steady-state computations. *International Journal for Numerical Methods in Fluids*, **2004**. *44(9)*: p. 975-995.
6. D-H., K., S.S. Paramanantham, and W.-G. Park, Numerical Analysis of Multi-Phase Flow around Supercavitating Body at Various Cavitator Angle of Attack and Ventilation Mass Flux. *Applied Sciences*, **2020.** *10(12)*: p. 4228.
7. Phan, T.-H., et al., Numerical analysis of an unsteady natural cavitating flow around an axisymmetric projectile under various free-stream temperature conditions. *International Journal of Heat and Mass Transfer*, **2021**. *164*: p. 120484.
8. Duy, T.-N., et al., An enhancement of coupling method for interface computations in incompressible two-phase flows. *Computers & Fluids*, **2021**. *214*: p. 104763.
9. Nguyen Duy, T. and T. Hino, An improvement of interface computation of incompressible two-phase flows based on coupling volume of fluid with level-set methods. *International Journal of Computational Fluid Dynamics*, **2020**. *34(1)*: p. 1-15.
10. Nguyen, T.T., V.P.T. Luu, and D.T. Nguyen, Improved Interface Capturing for Ship Hydrodynamics and Multiphase Flow Simulation. In: Randolph M., Doan D., Tang A., Bui M., Dinh V. (eds) *Proceedings of the 1st Vietnam Symposium on Advances in Offshore Engineering. VSOE 2018*. Lecture Notes in Civil Engineering, vol 18. Springer, Singapore., **2019**: p. 298-304.
11. Deshpande, S.S., L. Anumolu, and M.F. Trujillo, Evaluating the performance of the two-phase flow solver interFoam. *Computational Science & Discovery*, **2012**. *5(1)*: p. 014016.
12. Park, S. and S.H. Rhee, Computational analysis of turbulent super-cavitating flow around a two-dimensional wedge-shaped cavitator geometry. *Computers & Fluids*, **2012**. *70*: p. 73-85.

* Corresponding Author: Warn-Gyu Park, wgpark@pusan.ac.kr