DEVELOPMENT OF A MULTIPHASE SOLVER FOR CAVITATION FLOW NEAR FREE SURFACE

HOUCUN ZHOU[1], MIN XIANG[2], SHIWEI ZHAO[3], WEIHUA ZHANG[4]

[1]*National University of Defense Technology, zhouhoucun09@nudt.edu.cn* [2]*National University of Defense Technology, xiangmin333@hotmail.com* [3]*National University of Defense Technology, 15211122522@163.com* [4]*National University of Defense Technology, zhangweihua@nudt.edu.cn*

Keywords: *Free Surface, Cavitation, Multipase, OpenFOAM*

1 Introduction

Cavitation is a common phenomenon in fluid machinery, which oftentimes negatively affects the performance of most fluid machinery applications, therefore leading to problems such as undue vibrations, noise and material erosion [1]. In more recent years, cavitation has attracted intensive attention due to its potentials in drag reduction for underwater vehicles [2]. When vehicles run near or across the free surface, ventilated cavitation happens, which is complicated issue and may provide new inspiration on high-speed surface cruising. Researchers have done a lot of investigation on this topic up to now[3][4][5]. In order to fully studies this problem, a multiphase cavitation solver is developed based on the OpenFOAM open source platform. And a fully investigation of the free surface flow will be carried out with the developed solver.

2 Mathematical Method

2.1 Governing Equation

In this paper, the flow described is treated as a homogeneous mixture, therefore only one set of equations is needed. The governing equations basically consist of the conservation of mass, momentum. The continuity equation of the mixture flow can be written as

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

Neglecting the gravity and surface tension term, the conservation of momentum for the mixture flow can be expressed as,

$$\frac{\partial (\rho U)}{\partial t} + \nabla \cdot (\rho U \otimes U) = -\nabla p + \nabla \cdot \tau \tag{2}$$

Where ρ is the density of the mixture, which is related to the volume of fractions of all phases as

$$\rho = \alpha_l \rho_l + \alpha_v \rho_v + \alpha_g \rho_g \tag{3}$$

For a three phases system, the basic form of transport equations for the volume fraction of each phase could write as:

$$\frac{\partial (\rho_l \alpha_l)}{\partial t} + \nabla \cdot (\rho_l \alpha_l U) = \dot{m} \tag{4}$$

$$\frac{\partial (\rho_v \alpha_v)}{\partial t} + \nabla \cdot (\rho_v \alpha_v U) = -\dot{m} \tag{5}$$

$$\frac{\partial (\rho_g \alpha_g)}{\partial t} + \nabla \cdot (\rho_g \alpha_g U) = 0 \tag{6}$$

Where the subscript l and v are for the liquid and vapour phases respectively. While g represents gas. Note that the velocity vector U in the above equations should be expressed as the averaged velocity, and $U = \alpha_l U_l + \alpha_v U_v + \alpha_g U_g$. And the $\dot{m}$ term on the RHS of the equations is donates the mass transfer rate caused by cavitation between the liquid and vapour phase, which is $\dot{m} = \dot{m}^+ + \dot{m}^-$.
Considering that the volume fraction of these phases obey the conservation law,

$$\alpha_l + \alpha_v + \alpha_g = 1 \tag{7}$$

The divergence of the velocity can be,

$$\nabla \cdot U = \left(\frac{1}{\rho_l} - \frac{1}{\rho_v} \right) \dot{m} \tag{8}$$

Neglecting the compressibility of the phases, and take the divergence term into consideration, the final form of the transport equation is:

$$\frac{\partial \alpha_l}{\partial t} + \nabla \cdot (\alpha_l U) = \alpha_l (\nabla \cdot U) + \left(\frac{1}{\rho_l} - \alpha_l \left(\frac{1}{\rho_l} - \frac{1}{\rho_v} \right) \right) \dot{m} \tag{9}$$

$$\frac{\partial \alpha_v}{\partial t} + \nabla \cdot (\alpha_v U) = \alpha_v (\nabla \cdot U) - \left(\frac{1}{\rho_v} + \alpha_v \left(\frac{1}{\rho_l} - \frac{1}{\rho_v} \right) \right) \dot{m} \tag{10}$$

$$\frac{\partial \alpha_g}{\partial t} + \nabla \cdot (\alpha_g U) = \alpha_g (\nabla \cdot U) - \alpha_g \left(\frac{1}{\rho_l} - \frac{1}{\rho_v} \right) \dot{m} \tag{11}$$

Given that the interface disperses due to numerical diffusion it might appear that a good approach to limit or reverse this effect would be to include a diffusion operator into the phase-fraction equation with a negative diffusion coefficient. While this approach would be conservative it would also be unbounded and unstable; negative diffusion is always problematic. An alternative to negative diffusion which is also conservative is to apply some kind of additional convection-based term which compresses the interface, maintains boundedness and reduces to zero (at least the integrated effect of it reduces to zero) as the mesh is refined. A "Counter-gradient" term, which is clearly conservative and maintains boundedness, has a general form of $\nabla \cdot (U_r \alpha \beta)$, $U_r$is the relative velocity between the two phases across the interface.

$$\frac{\partial \alpha_l}{\partial t} + \nabla \cdot (\alpha_l U) + + \nabla \cdot (\alpha_l \alpha_v (U_l - U_v) + \alpha_l \alpha_g (U_l - U_g)) = \alpha_l (\nabla \cdot U) + \left(\frac{1}{\rho_l} - \alpha_l \left(\frac{1}{\rho_l} - \frac{1}{\rho_v} \right) \right) \dot{m} \tag{12}$$

$$\frac{\partial \alpha_v}{\partial t} + \nabla \cdot (\alpha_v U) + \nabla \cdot (\alpha_v \alpha_l (U_v - U_l) + \alpha_v \alpha_g (U_v - U_g)) = \alpha_v (\nabla \cdot U) - \left(\frac{1}{\rho_v} + \alpha_v \left(\frac{1}{\rho_l} - \frac{1}{\rho_v} \right) \right) \dot{m} \tag{13}$$

$$\frac{\partial \alpha_g}{\partial t} + \nabla \cdot (\alpha_g U) + \nabla \cdot (\alpha_g \alpha_l (U_g - U_l) + \alpha_g \alpha_v (U_g - U_v)) = \alpha_g (\nabla \cdot U) - \alpha_g \left(\frac{1}{\rho_l} - \frac{1}{\rho_v} \right) \dot{m} \tag{14}$$

However, in this study, the homogeneous multiphase model which adopts an incompressible Navier-Stokes equations and the compressive volume of fluid (VOF) approach are adopted for flow simulation. Phases in the multiphase system share the velocity and pressure fields. So the relative velocity term should be reconstructed. Here we just adopt the following form which is also used by some other multiphase solver in OpenFOAM.

$$U_c = \min \left(\mathrm{c}_\alpha |\mathrm{U}|, \max \left(|\mathrm{U}| \right) \right) \frac{\nabla \alpha}{\|\nabla \alpha\|} \tag{15}$$

where c_α is a compressive factor, expression $\max(|\mathrm{U}|)$ returns the largest value of $|U|$ anywhere in the domain.

2.2 Flux Corrected Transport Theory

In OpenFOAM, the volume fraction transport equation of the multiphase system is solved by the MULES tool kit. For the given equation below

$$\frac{\partial (\rho \alpha)}{\partial t} + \nabla \cdot (\rho \alpha U) = \alpha \mathrm{Sp} + \mathrm{Su} \tag{16}$$

We have the following discretization equation

$$\rho \frac{\alpha^{n+1} - \alpha^n}{\Delta t} + F = \alpha^{n+1} \mathrm{Sp} + \mathrm{Su} \tag{17}$$

$$\alpha^{n+1} \left(1 - \frac{\Delta t \mathrm{Sp}}{\rho} \right) = \frac{\Delta t}{\rho} \left(\frac{\alpha^n \rho}{\Delta t} + \mathrm{Su} - \mathrm{F} \right) \tag{18}$$

$$\alpha^{n+1} = \frac{\alpha^n \rho / \Delta t + \mathrm{Su} - \mathrm{F}}{\rho / (\Delta t) - \mathrm{Sp}} \tag{19}$$

Usually, the density is set to unity in MULES correction. Therefore, the Sp term is an implicity volumetric source term due to cavitation, the Su is an explicit term in Eq.19. And F is the total volume fraction flux due to transportive effect of a velocity. The values of the flux depend on many variables but particularly on the values of α at faces. Boundedness of the temporal solution can be achieved via face value limiting, such as in TVD/NVD schemes, or by limiting the face fluxes. The values of F are obtained by a lower order and bounded method and a limited portion of the values obtained

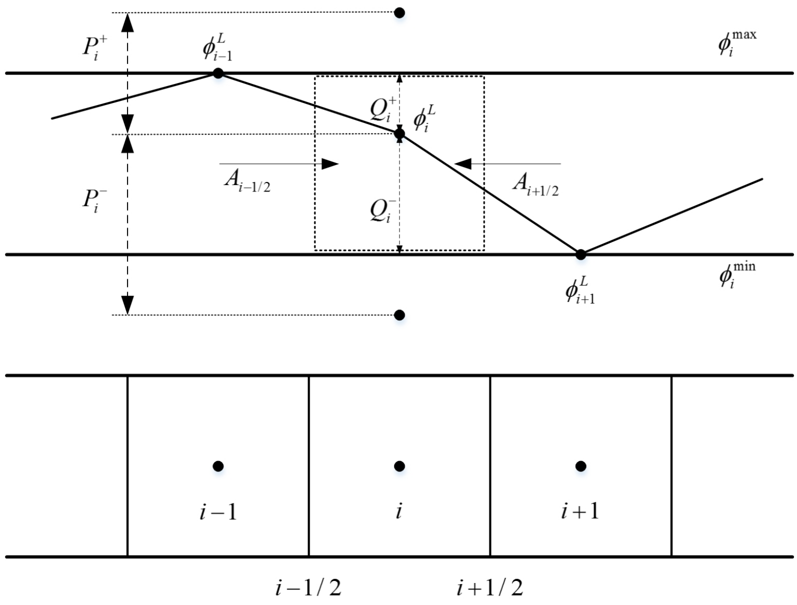

Figure 1: Schematic diagram of limiter for one dimensional geometry.

by a high order and possible unbounded method. This technique is Flux Corrected Transport (FCT). The sequence of this theory could be described as below[6]:

1. Compute F^L, the transportive flux given by some low order scheme which guarantees to give monotonic results.
2. Compute F^H, the transportive flux given by some high order scheme.
3. Define the anti-diffusive flux $A = F^H - F^L$.
4. Compute the corrected flux $F^C = F^L + \lambda A$, with $0 \leq \lambda \leq 1$.
5. Solve the equation by the given temporal scheme using corrected fluxes.

The critical step is clearly the fourth, where it is necessary to find the λ weighting factors.

The implementation of FCT theory is called MULES (Multidimensional Universal Limiter for Explicit Solution)[6] in OpenFOAM. Illustrated by Fig.1, the whole procedure is presented in Algorithm 1.

Algorithm 1 Procedure for MULES limiter

1: Calculate local extrema as

$$\alpha_i^a = max(\alpha_i^n, \alpha_{i,N}^n)$$
$$\alpha_i^b = min(\alpha_i^n, \alpha_{i,N}^n)$$

where $\alpha_{i,N}^n$ are all the neighbors by face for the i-th cell. In addition the inflows and outflows for each cell have to be calculated as $P^+ = -\sum_f A_f^-$ and $P^- = -\sum_f A_f^+$, where A_f^- are the inflows and A_f^+ the outflows;

2: Correct the local extrema by the limits imposed by user's defined global extrema α^{maxG} and α^{minG}

$$\alpha_i^a = max(\alpha^{maxG}, \alpha_i^a)$$
$$\alpha_i^b = min(\alpha^{minG}, \alpha_i^b)$$

3: Find $Q_i^{\pm}$ as

$$Q_i^+ = \frac{V}{\Delta t}(\alpha_i^a - \alpha_i^n) + \sum_f F_f^L$$
$$Q_i^- = \frac{V}{\Delta t}(\alpha_i^n - \alpha_i^b) - \sum_f F_f^L$$

4: Set $\lambda_f^{v=1} = 1$ for all faces. Do the following loop $nLimiterIter$ times to find the finalλ_f's

$$\lambda_i^{\mp,v+1} = max\left[min\left(\frac{\pm\sum_f \lambda_f^v A_f^{\pm} + Q_i^{\pm}}{P_i^{\pm}}, 1\right), 0\right]$$

$$\lambda_f^{v+1} = \begin{cases} min\{\lambda_P^{+,v+1}, \lambda_N^{-,v+1}\}, \mathrm{if A_{i+1/2}} \geq 0, \\ min\{\lambda_P^{-,v+1}, \lambda_N^{+,v+1}\}, \mathrm{if A_{i+1/2}} < 0 \end{cases}$$

where λ_P and λ_N represent the λ's for the owner and neighbor cell of a given face f.

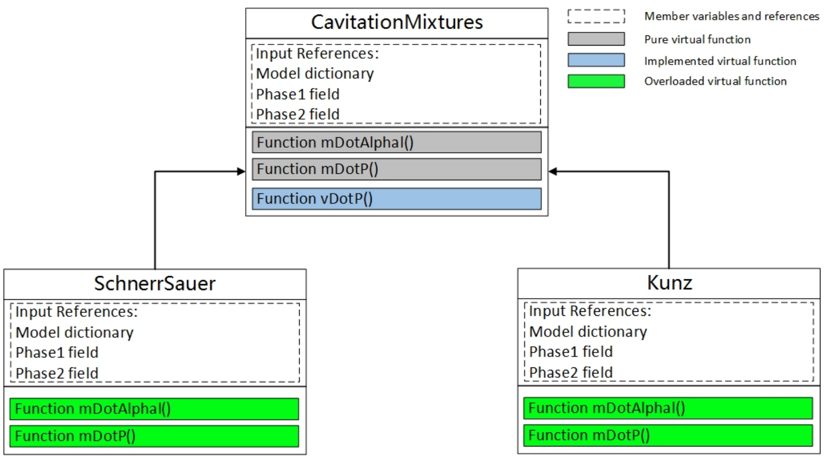

Figure 2: Block diagram of the mass transfer model framework and implemented models.

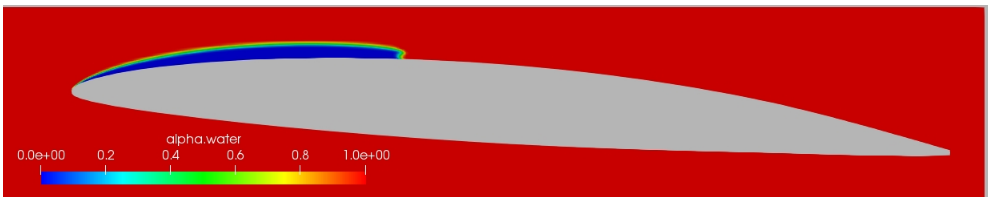

Figure 3: Sheet cavitation on the suction side of the hydrofoil

2.3 Mass Transfer Model

In order to calculate the mass transfer rate within multiphase systems, a framework of cavitation model is developed. All models inherit from a abstract class, which accepts a dictionary reference to look up parameters needed by the model and two reference of phase fraction field object. Additionally, the Kunz model[7] and Schnerr Sauer model[8] are also implemented to return the mass transfer rate. A UML block diagram of the mass transfer model framework is presented in Fig.2.

3 Validation

3.1 2D hydrofoil case

In this section, the two dimensional NACA66(MOD) hydrofoil is adopted. The main purpose of this case is validating the performance of two phase flow simulation. So only water and vapour phase are involved in this validation case. Results calculated by the new solver will compare with experimental results[9] and those calculated by the interPhaseChangeFoam solver. Some of the primary results are shown in Fig.3 and Fig.4.

3.2 2D throttle case

This 2D throttle case is copied form cavitatingFoam tutorials.In this case, three phases are involved, i.e. the water, vapour phase and gas. Two different flow condition are considered. Numerical simulation results are shown in Fig.5 and Fig.6, which validate the developed solver.

3.3 3D underwater projectile

The effect of the free surface has a great effect on high-speed surface vehicles. Wang et. al [3] carried out a typical launching experiment around an axisymmetric projectile to investigate the free surface effect. In this study, the established numerical approach is adopted to study this case. The numerical methods are validated by comparing results with underwater launching experiments. Results are shown in Fig.7 and Fig.8.

4 Conclusion

The new developed multiphase cavitation solver shows good performance in two phases and three phases cavitating flow simulation. It could also be applied to analyse the cavitating flow near free surface.

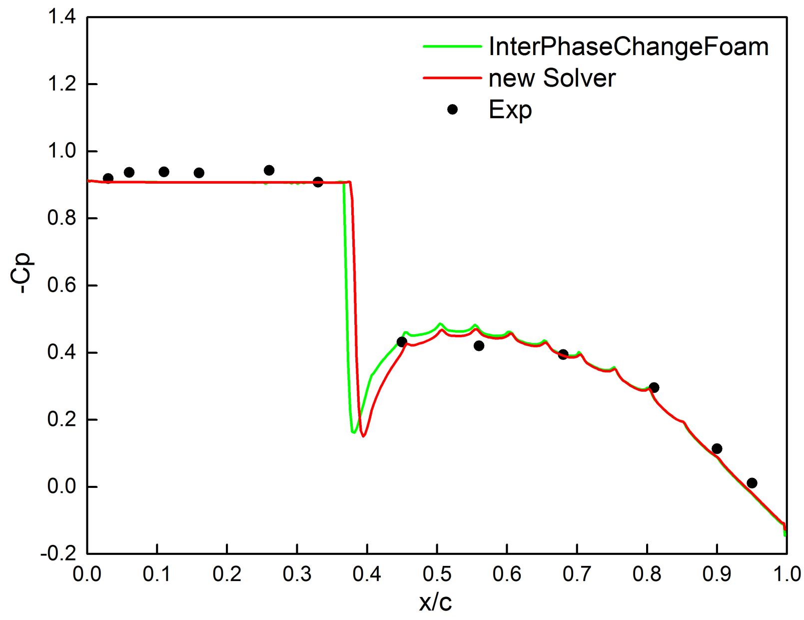

Figure 4: Pressure coefficient distribution on the suction side of the hydrofoil

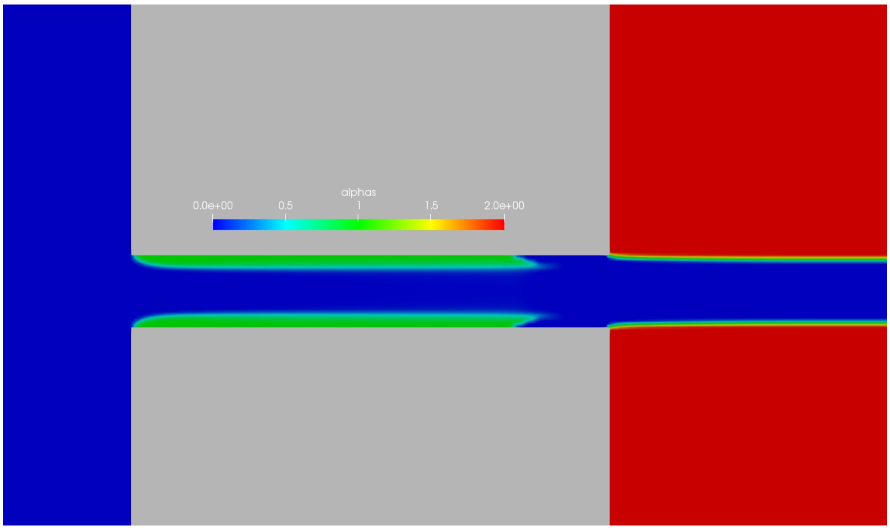

Figure 5: Volume fraction distribution in the throttle($P_{out} = 15\mathrm{atm}$)

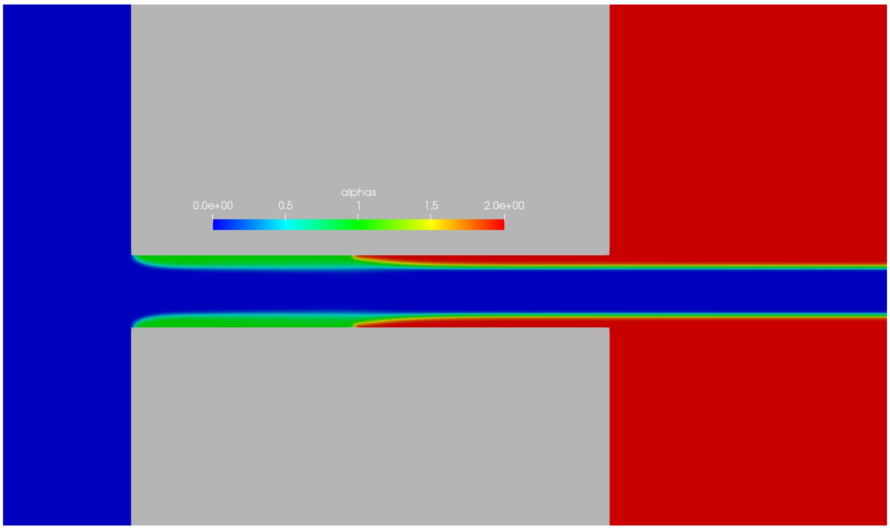

Figure 6: Volume fraction distribution in the throttle($P_{out} = 10\mathrm{atm}$)

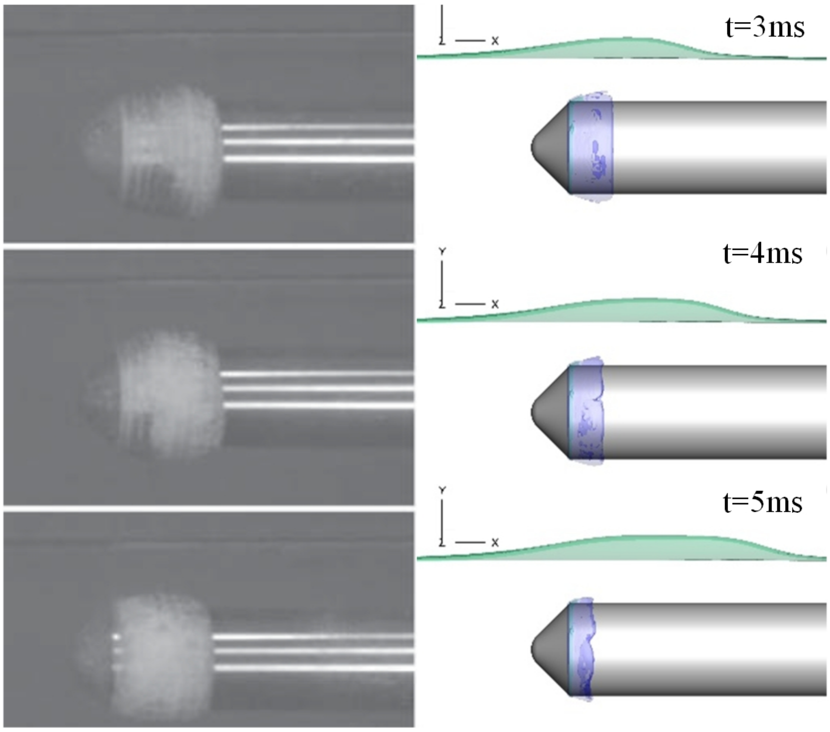

Figure 7: Evolutions of the cavity and free surface

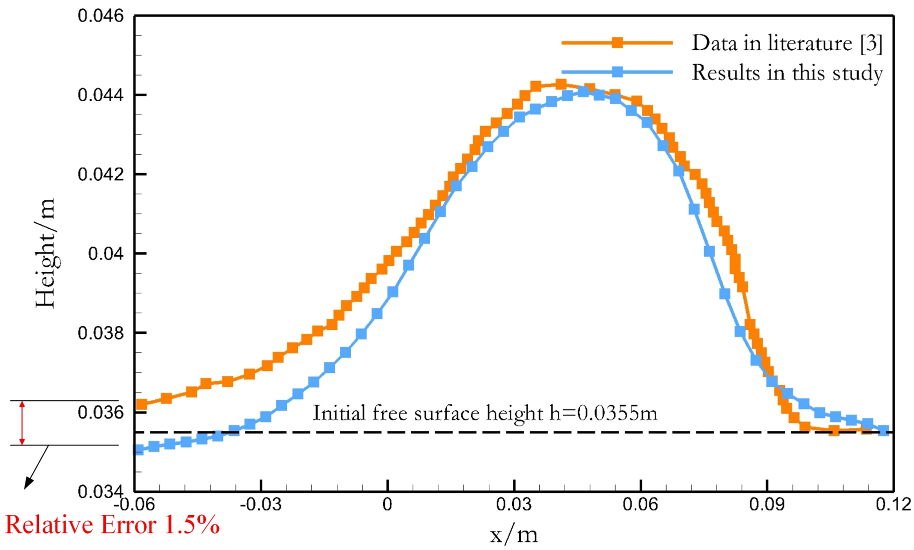

Figure 8: Wave profiles on the upper side of the projectile

Acknowledgments

The authors gratefully acknowledge support by the National Nature Science Foundation of China (NSFC, Grant NO: 51776221). And thank all those involved in the organisation of OFW13 and to all the contributors that will enrich this event.

References

[1] J.-P. Franc, J.-P. Franc, J.-M. Michel, and J.-M. Michel, *Fundamentals of cavitation*, 2004.

[2] B.-K. Ahn, C.-S. Lee, and H.-T. Kim, “Experimental and numerical studies on super-cavitating flow of axisymmetric cavitators,” *International Journal of Naval Architecture and Ocean Engineering*, vol. 2, no. 1, pp. 39–44, mar 2010. [Online]. Available: http://linkinghub.elsevier.com/retrieve/pii/S2092678216302308

[3] Y. Wang, X. Wu, C. Huang, and X. Wu, “Unsteady characteristics of cloud cavitating flow near the free surface around an axisymmetric projectile,” *International Journal of Multiphase Flow*, vol. 85, pp. 48 – 56, 2016. [Online]. Available: http://www.sciencedirect.com/science/article/pii/S0301932215300999

[4] M.-S. Jin, C.-T. Ha, and W.-G. Park, “Numerical study of ventilated cavitating flows with free surface effects,” *Journal of Mechanical Science and Technology*, vol. 27, no. 12, pp. 3683–3691, Dec 2013. [Online]. Available: https://doi.org/10.1007/s12206-013-0914-0

[5] C. Xu, J. Huang, Y. Wang, X. Wu, C. Huang, and X. Wu, “Supercavitating flow around high-speed underwater projectile near free surface induced by air entrainment,” *AIP Advances*, vol. 8, no. 3, p. 035016, 2018. [Online]. Available: https://doi.org/10.1063/1.5017182

[6] S. M. Damian, “An extended mixture model for the simultaneous treatment of short and long scale interfaces,” 2013.

[7] R. F. Kunz, D. A. Boger, T. S. Chyczewski, D. R. Stinebring, H. J. Gibeling, and T. R. Govindan, “Multi-Phase Cfd Analysis of Natural and Ventilated Cavitation About Submerged Bodies,” *FEDSM 99 1999 ASME/JSME Joint Fluids Engineering Conference*, pp. 1–9, 1999.

[8] G. H. Schnerr and J. Sauer, “Physical and numerical modeling of unsteady cavitation dynamics,” in *Proceedings of the 4th International Conference on Multiphase Flow*, New Orleans, La, USA, 2001.

[9] P. E. Dimotakis, H. F. Gaebler, and H. T. Hamaguchi, “Two Dimentional NACA 66(MOD) Hydrofoil High Speed Water Tunnel Tests,” GALCIT Report HSWT 1142, Tech. Rep., 1987.