Research Article

A Study on the Cavitation Model for the Cavitating Flow Analysis around the Marine Propeller

Insu Lee,¹ Sunho Park^{ID},² Woochan Seok,³ and Shin Hyung Rhee^{ID}^{1,3}

¹Department of Naval Architecture and Ocean Engineering, Seoul National University, Seoul, Republic of Korea

²Department of Ocean Engineering, Korea Maritime and Ocean University, Busan, Republic of Korea

³Research Institute of Marine Systems Engineering, Seoul National University, Seoul, Republic of Korea

Correspondence should be addressed to Shin Hyung Rhee; shr@snu.ac.kr

Received 13 April 2021; Revised 1 June 2021; Accepted 5 June 2021; Published 19 June 2021

Academic Editor: Hao Zhang

Copyright © 2021 Insu Lee et al. This is an open access article distributed under the Creative Commons Attribution License, which permits unrestricted use, distribution, and reproduction in any medium, provided the original work is properly cited.

In this study, a cavitation model for propeller analysis was selected using computational fluid dynamics (CFD), and the model was applied to the cavitating flow around the Potsdam Propeller Test Case (PPTC) propeller. The cavitating flow around the NACA 66 hydrofoil was analyzed to select a cavitation model suitable for propeller analysis among various cavitation models. The present and the experimental results were compared to select a cavitation model that would be applied to propeller cavitation analysis. Although the CFD results using the selected cavitation model showed limitations in estimating some of the foam cavitation and bubble cavitation identified in the experimental results, it was identified that foam cavitation and sheet cavitation around the tip were well simulated.

1. Introduction

Cavitation is a phenomenon for liquids to evaporate when the pressure around hydrodynamic machinery becomes lower than the vapor pressure. In particular, the thrust of the marine propeller is generated by the pressure difference between the pressure side and the suction side, and cavitation mainly occurs at the suction side due to the low pressure at the suction side. When the cavitation occurs excessively, noise, vibration, erosion, and performance degradation occur. Therefore, highly precisely predicting propeller cavitation is very important when designing marine propellers.

From around 20 years ago, studies related to the development of cavitation models for estimation of cavitation using computational fluid dynamics (CFD) have been conducted. Merkle [1] took notice of the macroscopic movement of cavitation, modeled cavitation using dynamic relations between the inside and outside of the cavity, and verified the cavitation model by applying it to the prediction of the cavitating flow around the NACA 66 hydrofoil. Kunz et al. [2] created a cavitation model by simplifying the

Merkle [1] model using the potential theory and verified the model with the surface pressure of the hemispheric head cylindrical body where cavitation occurs. Schnerr and Sauer [3] created a cavitation model by introducing the relation between nuclei and volume fraction to the Rayleigh–Plesset equation, an equation for a single bubble, and verified the model by applying it to the analysis of the cavitation around the nozzle neck. Zwart et al. [4] created a cavitation model considering the space occupied by nucleation in the Rayleigh–Plesset equation and verified the model by applying it to the cavitating flow around the NACA 66 hydrofoil.

Recently, studies on the cavitation phenomenon of propellers have been conducted using various cavitation models. Ahn and Kwon [5] analyzed the cavitating flow around the P4381 propeller developed by the Naval Ship Research and Development Center (NSRDC) using the Merkle cavitation model. From the CFD results, it was identified that the thrust, torque, and head changes matched well when compared with the experimental results. Paik et al. [6] performed cavitation analysis with a propeller mounted behind the hull using the Schnerr and Sauer cavitation model. It was identified that the results were similar to the

experimental results performed in the cavitation tunnel in terms of the cavity shape and pressure fluctuations. Wang et al. [7] used the Schnerr and Sauer cavitation model to analyze the cavitating flow during the E779A propeller heave motion. To verify the cavitation model, propeller cavitation analysis was performed in the absence of the heave motion, and the results of the analysis were compared with the experimental results. Yilmaz et al. [8] applied the Schnerr and Sauer cavitation model and adaptive mesh refinement to analyze the cavitating flow around the hull, rudder, and propeller. In the cavitation tunnel experiment, the CFD results similar to the results of the cavitation observation test were derived.

As with previous studies, various propeller geometries were used to estimate the cavitating flow around the propeller. Since the propeller geometry and experimental data for the Potsdam Propeller Test Case (PPTC) propeller among the propellers have been open to the public, many studies using the PPTC propeller have been conducted. Heinke [9] conducted cavitation tests for a PPTC propeller in a cavitation tunnel. She explained the cavity shape around the propeller according to the advance ratio and cavitation number and shared the propeller geometry and experimental results. Lübke [10] carried out cavitation tests for PPTC propellers in oblique flow conditions and summarized the experimental results and CFD results. Gaggero and Villa [11] analyzed the cavitating flow around a PPTC propeller using the boundary element method (BEM). They used the $k-\omega$ SST model as a turbulence model and applied the model to the NACA66 hydrofoil to verify the numerical model. They applied the verified numerical model to the PPTC propeller and identified that the experimental results and the CFD results were consistent. Usta and Korkut [12] analyzed the cavitating flow analysis around hydrofoil and PPTC propellers using the Detached Eddy Simulation (DES) model. For turbulence termination, they used the $k-\omega$ SST model and identified that the cavity shape in the analysis result simulated the cavitation observation tests well.

In this study, CFD simulations on the cavitating flow around a PPTC propeller was carried out. Before propeller cavitation analysis, CFD simulations were carried out to select a cavitation model for highly precise cavitation analysis. To select the cavitation model, the Merkle model and the Schnerr and Sauer model, which were mainly used in previous studies, were used. The NACA 66 hydrofoil, which is mainly used as the propeller section, was selected as a target model, and the present results of the cavitation flow analysis around the NACA 66 hydrofoil were compared with the model test results to select a cavitation model for marine propeller cavitation analysis. The selected cavitation model was applied to the target propeller to carry out CFD simulations, and the results were also compared with the model test results. In this study, OpenFOAM, an open source CFD toolkit, was used to simulate the cavitating flows around the NACA 66 hydrofoil and marine propeller. The simulation for cavitating flows around the NACA 66 hydrofoil was performed using interPhaseChangeFoam, a cavitating flow solver of OpenFOAM. The simulation for cavitating flows around the marine propeller was performed

using modified interPhaseChangeFoam in a moving reference frame. This paper is composed as follows. First, the computational fluid analysis method is mentioned and the analysis target, analysis conditions, and analysis grid are presented. Thereafter, the results of the NACA 66 hydrofoil analysis performed to select a cavitation model are analyzed, and a cavitation model is selected. The selected model is applied to propeller cavitation analysis, and the present results are discussed. Finally, a summary and conclusions are provided.

2. Computational Method

2.1. Governing Equations. The continuity equation for incompressible flow and the Reynolds-averaged Navier-Stokes (RANS) equation were used as governing equations to simulate the cavitating flow around the NACA66 hydrofoil, and the equations are expressed, as shown in the following equations:

$$\rho \frac{\partial \bar{u}_i}{\partial x_i} = \left(\frac{1}{\rho_v} - \frac{1}{\rho_l} \right) S_\alpha, \quad (1)$$

$$\rho \frac{\partial \bar{u}_i}{\partial t} + \rho \bar{u}_j \frac{\partial \bar{u}_i}{\partial x_j} = -\frac{\partial \bar{p}}{\partial x_i} + \frac{\partial}{\partial x_j} \left(\mu \frac{\partial \bar{u}_i}{\partial x_j} - \tau'_{ij} \right) - S_{M,i}, \quad (2)$$

where ρ is the fluid density, $\bar{p}$ is the static pressure, $\bar{u}_i$ and $\bar{u}'_i$ represent the mean and fluctuating components of fluid velocity, respectively, μ is the dynamic viscosity, $\tau'_{ij} = \rho \bar{u}'_i \bar{u}'_j$ is the Reynolds' stress tensor, $S_{M,i}$ is the source term, and S_α is the source term of the volume fraction transport equation.

The moving reference frame (MRF) method was used to efficiently simulate the propeller under the condition that the inlet is uniform flow. The relation between the velocity in the moving reference frame $u_{rel,i}$ and the velocity in the absolute frame u_i is shown in the following equation:

$$u_{rel,i} = u_i - \epsilon_{ijk} \Omega_j r_k, \quad (3)$$

where Ω_i is the rotational velocity of the reference frame and r_i is the position vector. By substituting equation (3) into equations (1) and (2), which are the governing equations of the absolute frame, and organizing it, the governing equations of the reference frame can be obtained. The governing equations used for the simulation of a propeller with a constant rotational velocity in this study are shown in the following equations:

$$\rho \frac{\partial u_{rel,i}}{\partial x_i} = 0, \quad (4)$$

$$\begin{aligned} \rho \frac{\partial u_{rel,i}}{\partial t} + \rho u_{rel,j} \frac{\partial u_{rel,i}}{\partial x_j} = & -\frac{\partial \bar{p}}{\partial x_i} + \frac{\partial}{\partial x_j} \left(\mu \frac{\partial u_{rel,i}}{\partial x_j} - \tau'_{ij} \right) \\ & - \left\{ \epsilon_{ijk} \Omega_j \epsilon_{klm} \Omega_l r_m + 2\epsilon_{ijk} \Omega_j u_{rel,k} \right\}. \end{aligned} \quad (5)$$

For the turbulence closure, the $k-\omega$ shear stress transport (SST) model, which is a turbulence model mainly used in propeller simulation in previous studies, was applied.

For the time derivative terms, the first-order accurate implicit scheme was used. For the spatial difference terms, the second-order accurate scheme using linear interpolation was used. For the pressure-velocity coupling, the PIMPLE algorithm, which combines the semi-implicit method for pressure linked equations' (SIMPLE) algorithm and the pressure implicit with the splitting of the operator (PISO) algorithm, was adopted.

2.2. Cavitation Simulation Method. The volume fraction was introduced to simulate cavitation. The volume fraction is expressed as 1 for the liquid phase and 0 for the vapor phase and exists between 0 and 1 in the free surface. The transport equation of the volume fraction is shown in the following equation:

$$\frac{\partial \alpha}{\partial t} + \frac{\partial \alpha u_i}{\partial x_i} = \frac{S_\alpha}{\rho}. \quad (6)$$

Here, α is the volume fraction and S_α is the source term for simulation of cavitation.

The source term S_α varies according to cavitation models. In this study, the Merkle model and Schnerr and

Sauer model, which are cavitation models mainly used for the simulation of the cavitating flow around propellers, are used. Taking notice of the macroscopic behavior of the cavity, Merkle [1] created a cavitation model that considers the dynamic equilibrium between the inside and outside of the cavity. In the cavitation model proposed by Merkle [1], the source term is shown in the following equation:

$$S_\alpha = \begin{cases} \frac{C_c}{(1/2)U_\infty^2 t_\infty} (1 - \alpha)(p - p_v), & p > p_v, \\ -\frac{C_v \rho_l}{(1/2)\rho_v U_\infty^2 t_\infty} \alpha(p_v - p), & p < p_v, \end{cases} \quad (7)$$

where C_c is the condensation coefficient, C_v is the evaporation coefficient, p is the pressure, p_v is the vapor pressure, ρ_l is the density of the liquid, U_∞ is the velocity of the inflow, and t_∞ is the time scale of the cavitation.

Schnerr and Sauer [3] proposed a cavitation model based on the Rayleigh-Plesset equation, which is an equation for a single bubble. The following equation is the source term in the cavitation model proposed by Schnerr and Sauer [3]:

$$S_\alpha = \begin{cases} C_c \frac{\rho_v \rho_l}{\rho} \alpha (1 - \alpha) \frac{3}{R} \sqrt{\frac{2}{3} \frac{p - p_v}{\rho_l}}, & p > p_v, \\ -C_v \frac{\rho_v \rho_l}{\rho} \alpha (1 - \alpha) \frac{3}{R} \sqrt{\frac{2}{3} \frac{p_v - p}{\rho_l}}, & p < p_v, \end{cases} \quad \text{where } R = \left(\frac{1 - \alpha}{\alpha} \frac{3}{4\pi n_0} \right)^{(1/3)}, \quad (8)$$

where R is the bubble radius and n_0 is the number of nuclei per unit volume.

2.3. Numerical Setup for Selection of Cavitation Models. Before propeller cavitation simulation, the NACA 66 hydrofoil was used to select a cavitation model. This geometry is often used as a blade section of a propeller and is mainly used for verification of cavitation models. For comparison with the experimental results of Shen and Dimotakis [13], the NACA 66 hydrofoil proposed by Brockett [14] was used, and values of $f/C = 0.020$ and $t/C = 0.09$ were used, where f is the maximum camber, t is the maximum thickness, and C is the chord length. The working conditions for simulation were the same as the experimental conditions of Shen and Dimotakis [13] and are shown in Table 1, where U_∞ is the freestream velocity and p_{ref} is the reference pressure. $\sigma = (p_{\text{ref}} - p_v)/(0.5\rho U_\infty^2)$ is the cavitation number, and $\text{Re} = (\rho C U_\infty/\mu)$ is the chord length-based Reynolds number. To select the cavitation model, the simulation was performed using the Merkle model and Schnerr and Sauer model.

The computational domain and boundary conditions for selection of a cavitation model are shown in Figure 1. The size of the computational domain is 5C in the direction toward the inlet, 8C in the direction toward the outlet in the

hydrofoil, and 5C in normal directions to freestream velocity. In the hydrofoil shown in Figure 1, 2D simulation was performed by selecting the outlet in the hydrofoil as the x -axis and the vertical upward direction as the z -axis. At the inlet, the Dirichlet boundary condition was applied to the velocity and the Neumann boundary condition was applied to the pressure. At the outlet, the Neumann boundary condition was applied to the velocity and the Dirichlet boundary condition was applied to the pressure. In the hydrofoil, the no-slip boundary condition was applied to the velocity and the Neumann boundary condition was applied to the pressure. At the side, the slip boundary condition was applied to the velocity and the Neumann boundary condition was applied to the pressure.

As shown in Figure 2, the entire grids for NACA 66 hydrofoil simulation and the grids around the hydrofoil were enlarged. A total of 27,000 grids of C-type grids were used with 170 grids on the hydrofoil and 50 grids on the normal direction of the hydrofoil.

2.4. Numerical Setup of Propeller Simulation. As the object propeller for propeller cavitation simulation, VP1304, which is the PPTC propeller geometry, was selected. The main dimensions of VP 1304 are summarized in Table 2, and the

TABLE 1: NACA 66 hydrofoil working conditions.

AoA (deg.)	U_{∞} (m/s)	P_{ref} (Pa)	P_v (Pa)	σ	C (m)	Re	Remark
4	2.01	4035 5892	2340	0.84 1.76	1.0	2.0×10^6	Leading edge cavitation Noncavitating condition

FIGURE 1: Computational domain for the NACA 66 hydrofoil.

FIGURE 2: Computational grid for the NACA 66 hydrofoil.

TABLE 2: Main dimensions of VP1304.

VP 1304	
Diameter (m)	0.250
Pitch at $r/R = 0.7$ (m)	0.409
Pitch ratio at $r/R = 0.7$	1.635
Skew ($^{\circ}$)	18.837
Number of blades	5
Direction of rotation	Right-handed

geometry of the propeller is shown in Figure 3. The test conditions used were the same as the experimental condition in the report of Heinke [9] and are shown in Table 3. $K_T = (T/(pn^2D^4))$ is the thrust coefficients, and $\sigma_n = (p_{ref} - p_v)/(0.5\rho(nD)^2)$ and $\sigma_v = (p_{ref} - p_v)/(0.5\rho V_A^2)$ are the rate of the revolution-based cavitation number and the inflow velocity-based cavitation number, respectively, where T is the thrust of the propeller, n is the rate of revolution of the propeller, and D is the diameter of the propeller.

The computational domain and boundary conditions for propeller cavitation simulation are shown in Figure 4. In order to efficiently simulate the propeller under uniform inflow conditions, the calculation area was set as large as 1/5

FIGURE 3: VP1304 propeller geometry.

TABLE 3: VP1304 cavitation test conditions.

J	n (rps)	σ_n	σ_v	K_T (Noncavitating)
1.019	25	2.024	1.949	0.387
1.269		1.424	0.885	0.245
1.408		2.000	1.008	0.167

of the tunnel. The size of the computational domain is 6D in the direction toward the inlet and 8D in the direction toward the outlet in the blade. The radius in the direction perpendicular to the inflow was set to 1.34D so that the cross-sectional area would be the same as that of the cavitation tunnel in the experiment. The x -axis was in the direction toward the outlet in the inlet, and the z -axis was in the direction of the propeller reference line. At the inlet, the Dirichlet boundary condition was applied to the velocity and the Neumann boundary condition was applied to the pressure. At the outlet, the Neumann boundary condition was applied to the velocity and the Dirichlet boundary condition was applied to the pressure. In the blade, the no-slip boundary condition was applied to the velocity and the Neumann boundary condition was applied to the pressure. In the tunnel and shaft, the Dirichlet boundary condition was applied to the velocity and the Neumann boundary condition to the pressure. At the side, the periodic boundary condition was applied to both the velocity and pressure.

The propeller simulation grids were created as unstructured grids using Star-CCM+, and the surface grids of the propeller are shown in Figure 5. In order to perform efficient propeller simulation with a small number of grids, the areas around the blade were filled with polyhedral grids, and the inside of the domain was filled with trimmer grids. There are a total of 407,657 grids with 100,964 grids around the blade and 306,693 grids inside of the domain.

3. Cavitation Model Selection

In order to select a cavitation model that would be applied to the simulation of the cavitating flow around the propeller,

FIGURE 4: Computational domain and boundary conditions for propeller cavitation simulation.

FIGURE 5: Surface grids for propeller simulation.

the cavitating flow around the NACA 66 hydrofoil was simulated. Merkle and Schnerr and Sauer cavitation models were used for the cavitating flow simulation. The time-averaged surface pressure distribution under $\sigma = 1.76$ condition where cavitation was not identified in the experiment conducted by Shen and Dimotakis [13] is shown in Figure 6. The surface pressure was nondimensionalized, as shown in equation (9). In the analysis results, cavitation was not identified in neither of the cavitation models and the models showed the same surface pressure distribution. The present results were in good agreement with the results of the surface pressure distribution experiment conducted by Shen and Dimotakis [13], indicating that the computational method and grids used were appropriate:

FIGURE 6: NACA 66 hydrofoil surface pressure distribution under $\sigma = 1.76$.

$$C_p = \frac{(p - p_{\text{ref}})}{0.5\rho U_{\infty}^2}. \quad (9)$$

Figure 7 shows the time-averaged volume fraction results under the $\sigma = 0.84$ condition where leading edge cavitation was identified in the experiment. In the present results, cavitation inception appeared around the leading edge and cavitation closure was identified around the midchord in both cavitation models.

FIGURE 7: Results of analysis of the volume fraction of the NACA 66 hydrofoil under $\sigma = 0.84$. (a) Schnerr and Sauer model. (b) Merkle model.

In order to compare the results of simulation in the two models to figure out the differences, the time-averaged surface pressure distributions under the $\sigma = 0.84$ condition were nondimensionalized by equation (9) and shown in Figure 8. In the pressure distribution results, cavities were identified in the areas where $C_p = -\sigma$ in both cavitation models. From the present results with both cavitation models, it was identified that the experimental results were well simulated in the areas of $x/C = 0 \sim 5.5$ and $0.7 \sim 1.0$. However, it was identified that the Schnerr and Sauer model estimated the experimental results more similarly than the Merkle model in the surface pressure distribution in the area of $x/C = 5.5 \sim 0.7$, which is around the cavitation closure.

To analyze the difference in the surface pressure distributions, the velocity distributions in the freestream direction were compared. Figure 9 shows the velocity distributions in the freestream direction at the time when the re-entrant jet appeared most strongly under the $\sigma = 0.84$ condition. The velocity is shown after nondimensionalizing with freestream velocity, and the $\alpha \leq 0.5$ area is shown in gray. In the results of velocity distribution around the hydrofoil, the area around the cavitation closure is shown after being enlarged. Figure 9(a) shows the results of the Schnerr and Sauer model, and Figure 9(b) shows the results of the Merkle model. From the present results, the overall velocity distributions of the two models were estimated to be similar. In the velocity distributions around the cavitation closure, it can be identified that the re-entrant jet is generated more strongly in the results of the Schnerr and Sauer model than in the results of the Merkle model. This seems to be the reason why the point where the cavitation closure is located upstream in the results of the Schnerr and Sauer model than in the results of the Merkle model. Through the results of surface pressure distributions and the flow velocity distributions of hydrofoil, it was judged appropriate to use the Schnerr & Sauer cavitation model in propeller simulations.

4. Application

To accurately estimate propeller cavitation phenomena, propeller loading under noncavitating conditions must be accurately estimated. To that end, propeller analysis was performed under noncavitating conditions and the results were compared with the results of model tests. The propeller loading in this case means thrust. The rate of revolution and advance ratio of the propeller are shown in Tables 3, and 4 shows the experimental result of Heinke [9] and the present results. The difference between the experimental and present

FIGURE 8: NACA 66 hydrofoil surface pressure distribution under $\sigma = 0.84$.

results was divided by the experimental result, and it was indicated as Diff. (%). When the present and experimental results were compared for thrust coefficients, errors of $-0.60 \sim 2.07\%$ were shown. Through the foregoing, it was identified that the computational methods and grids are reasonable for the estimation of propeller loading.

Cavitation analysis was performed using the same computational methods and grids as those of the analysis under noncavitating conditions. The Schnerr and Sauer model selected using the results of NACA 66 hydrofoil cavitation analysis was used as a cavitation model, and $C_c = C_v = 1$ was used. Figure 10 shows the comparison between the cavitation sketch in the experiment and the cavitation development in the numerical prediction. Figure 10(a) shows the cavitation sketches of Heinke [9]; and, Figure 10(b) shows the numerical prediction. In the present results, the iso-surface of $\alpha = 0.5$ is shown in gray to indicate the cavity. The cavity shape at the suction side under the condition of $J = 1.019$ and 1.269 was shown, and the cavity shape at the pressure side under the condition of $J = 1.408$ was shown. The transition from the sheet to the foam cavitation around the hub in the sketch under the condition of $J = 1.019$ could not be estimated from the present results. However, the foam cavitation that appeared in the $r/R > 0.95$ in the sketch and the sheet cavitation around the hub were well estimated from the present results. The transition of the sheet around the hub to the foam cavitation in the sketch was

FIGURE 9: Velocity distributions in the freestream direction. (a) Schnerr and Sauer model. (b) Merkle model.

TABLE 4: Result of analysis of thrust coefficients under noncavitating conditions.

J	n (rps)	$K_{T, \text{no cav.}}$ [9]	$K_{T, \text{no cav.}}$ (Present results)	Diff. (%)
1.019	25	0.387	0.395	2.07
1.269		0.245	0.249	1.63
1.408		0.167	0.166	-0.60

FIGURE 10: Comparison of cavitation development around PPTC propellers. (a) Cavitation sketch [9]. (b) CFD analysis results (present results).

under the condition of $J = 1.269$, and the bubble cavitation that appeared in the $r/R = 0.5 \sim 0.9$ could not be estimated from the present results. However, the cavitation around the tip and the sheet cavitation around the hub were predicted well from the present results. The foam cavitation around the hub that appeared in the sketch under the condition of $J = 1.408$, and the transition of the cavitation detached from the sheet cavitation in the $r/R = 0.9 \sim 0.95$ to the streak cavitation were not properly estimated in the analysis but the

sheet cavitation around the leading edge and hub was well estimated. In summary, the transition from sheet cavitation to foam cavitation or streak cavitation in the sketch and bubble cavitation were not well estimated from the present results. Gnanaskandan and Mahesh [15] argued that the phenomenon of transition from sheet cavitation identified in the large eddy simulation (LES) and experiment to other types of cavitation was not properly captured in the RANS simulation and the transition of cavitation was not properly

captured in the results of the present results that used RANS. The reason why the bubble cavitation was not properly estimated from the analysis results was judged to be the fact that the grids were not dense enough to enable the computational grids to capture the bubble cavitation. However, in the simulation results, sheet cavitation and foam cavitation in the $r/R > 0.95$ were well estimated. It was identified that the present computational model and grids showed reasonable results in propeller cavitation analysis.

5. Summary and Conclusions

In this study, analyses of the cavitating flow around a PPTC propeller were carried out. In order to select the cavitation model, the cavitating flow around the NACA 66 hydrofoil was analyzed. For the analysis of the NACA 66 hydrofoil, the Schnerr and Sauer and Merkle cavitation models were used, and the Schnerr and Sauer model, which better simulated the experimental results around the cavitation closure, was selected as a cavitation model for propeller cavitation simulation. Before propeller cavitation analysis, PPTC propeller simulations were carried out under noncavitating conditions. The present results showed differences of $-0.60 \sim 2.07\%$ from the thrust coefficients in the experimental results, indicating that the computational method and grids in this study predict the propeller loading well. PPTC propeller cavitation simulations were performed using the same method and grids used under noncavitating conditions and the cavitation model selected in the NACA 66 hydrofoil analysis. It was identified that the present results well estimated the sheet cavitation identified in the sketch of the experimental results and the foam cavitation around the tip.

From the propeller cavitation analysis results, it was identified that the cavitation observed in the experimental sketches was generally well estimated, but some foam cavitation and bubble cavitation were not resolved. In the future, studies on the computational methods, grids, and cavitation models that can well estimate the phenomenon of transition from sheet cavitation to streak cavitation, foam cavitation, and bubble cavitation through studies of the cavitating flow around the propeller are necessary. In addition, further studies on turbulence models with advanced turbulence models, such as PANS (partially averaged Navier–Stokes), are needed to simulate the turbulence flows around the marine propeller with high fidelity.

Data Availability

The data used to support the findings of the study are included within this manuscript and are available from the corresponding author upon reasonable request.

Conflicts of Interest

The authors declare that they have no conflicts of interest.

Acknowledgments

This research was supported by the National Research Foundation of Korea through a grant funded by the Korean Government (NRF-2017K1A3A1A19071629 and NRF-2020R1I1A2074369) and Institute of Engineering Research

at Seoul National University. In addition, this was supported by Daewoo Shipbuilding & Marine Engineering-Seoul National University Future Ocean Cluster (0690-20190014).

References

- [1] C. L. Merkle, "Computational modelling of the dynamics of sheet cavitation," in *Proceedings of the 3rd International Symposium on Cavitation*, Grenoble, France, April 1998.
- [2] R. F. Kunz, D. A. Boger, T. S. Chyczewski, D. Stinebring, H. Gibeling, and T. Govindan, "Multi-phase CFD analysis of natural and ventilated cavitation about submerged bodies," in *Proceedings of the 3rd ASME-JSME Joint Fluids Engineering Conference*, San Francisco, CA, USA, July 1999.
- [3] G. H. Schnerr and J. Sauer, "Physical and numerical modeling of unsteady cavitation dynamics," in *Proceedings of Fourth International Conference on Multiphase Flow*, ICMF, New Orleans, LA, USA, June 2001.
- [4] P. J. Zwart, A. G. Gerber, and T. Belamri, "A two-phase flow model for predicting cavitation dynamics," in *Proceedings of Fifth International Conference on Multiphase Flow*, Yokohama, Japan, May 2004.
- [5] S. J. Ahn and O. J. Kwon, "Numerical investigation of cavitating flows for marine propulsors using an unstructured mesh technique," *International Journal of Heat and Fluid Flow*, vol. 43, pp. 259–267, 2013.
- [6] K.-J. Paik, H.-G. Park, and J. Seo, "RANS simulation of cavitation and hull pressure fluctuation for marine propeller operating behind-hull condition," *International Journal of Naval Architecture and Ocean Engineering*, vol. 5, no. 4, pp. 502–512, 2013.
- [7] L. Wang, C. Guo, Y. Su, P. Xu, and T. Wu, "Numerical analysis of a propeller during heave motion in cavitating flow," *Applied Ocean Research*, vol. 66, pp. 131–145, 2017.
- [8] N. Yilmaz, B. Aktas, M. Atlar, P. A. Fitzsimmons, and M. Felli, "An experimental and numerical investigation of propeller-rudder-hull interaction in the presence of tip vortex cavitation (TVC)," *Ocean Engineering*, vol. 216, Article ID 108024, 2020.
- [9] H. J. Heinke, "Potsdam propeller test case (PPTC), cavitation tests with the model propeller VP1304," Report 3753, Schiffbau Versuchsanstalt Potsdam, Potsdam, Germany, 2011.
- [10] L. Lübke, "Cavitation test in oblique flow case 2," in *Proceedings of Fourth International Symposium on Marine Propulsors Smp'15*, pp. 1–58, Austin, TX, USA, June 2015.
- [11] S. Gaggero and D. Villa, "Steady cavitating propeller performance by using OpenFOAM, StarCCM+ and a boundary element method," *Proceedings of the Institution of Mechanical Engineers, Part M: Journal of Engineering for the Maritime Environment*, vol. 231, no. 2, pp. 411–440, 2017.
- [12] O. Usta and E. Korkut, "A study for cavitating flow analysis using DES model," *Ocean Engineering*, vol. 160, pp. 397–411, 2018.
- [13] Y. Shen and P. E. Dimotakis, "The influence of surface cavitation on hydrodynamic forces," in *Proceedings of 22nd American Towing Tank Conference*, St. John's, Canada, August 1989.
- [14] T. Brockett, "Minimum pressure envelopes for modified naca-66 sections with naca A = 0.8 camber and buships type 1 and type 2 sections," Report 1780, David Taylor Model Basin, Washington, DC, USA, 1966.
- [15] A. Gnanaskandan and K. Mahesh, "Large eddy simulation of the transition from sheet to cloud cavitation over a wedge," *International Journal of Multiphase Flow*, vol. 83, pp. 86–102, 2016.