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

A numerical approach to the prediction of cavitation behavior and hull pressure fluctuation induced by marine propeller at full-scale

Jung-yun Won[1]*, Jaewook Hur[1], Jeong-yong Park[1] and Hyoungsuk Lee[1]

[1] Hyundai Maritime Research Institute, Hyundai Heavy Industries Co., Ltd., Republic of Korea

Abstract: In this study, numerical approach was conducted to evaluate cavitation behavior and pressure fluctuation using full-scale CFD simulation. Simulation results were compared with measured data and the applicability of full-scale CFD simulation was investigated. For the cavitation modelling, the S-S (Schnerr-Sauer) model, which is simplified form of the R-P (Rayleigh-Plesset) model, was used in the simulation. In order to verify the validity of full-scale CFD simulation result, full-scale measurement data of cavitation behavior and hull pressure fluctuation of two tanker and two container vessels were utilized. Cavitation pattern of simulation results and snap shots of videos taken by high-speed camera were compared at various propeller rotational positions, and FFT (Fast Fourier Transform) amplitudes of hull pressure fluctuation were compared with simulation result. For tanker ships, overall behavior of cavitation and pressure fluctuation were in good agreement with observation data. It is expected that numerical simulations at full-scale could give acceptable information on the design stage of tanker vessel propeller. However, for container ships, which have higher loading and ship speed than tanker vessels, relatively large difference were found in capturing tip vortex cavitation and pressure fluctuation.

Keywords: Marine Propeller; Full-scale measurement; CFD(Computational Fluid Dynamics); Schnerr-Sauer;

1. Introduction

Since marine propellers are rotating with high speed in the ship's complex three-dimensional wake field, improper propeller design may lead to unstable and excessive cavitation. Unstable propeller cavitation cause erosion on the blade surface, which can cause severe damage. So, it is necessary to precisely evaluate cavitation performance in the design stage to prevent cavitation related problems. In the design stage, this is mainly performed by model test or, numerical analysis in model scale. However, as the Reynolds number of full-scale ship is different from that of model scale ship, characteristics of cavitation and pressure fluctuation are different each other[1]. Therefore, it has been consistently raised that the necessity for the evaluation in full-scale to more accurately consider the effect of cavitation and related researches have been conducted. Ponkratov[2] predicted cavitation pattern and erosion area using full-scale CFD and the results were similar with observed data. In the research of Park[3], sound pressure level measurement data was compared with predicted data and the possibility for the practical use of full-scale CFD simulation was found.

In this study, cavitation pattern and pressure fluctuation of hull stern were predicted by numerical simulation in full-scale. For the cavitation pattern, numerical results were compared with snap shots at each rotational positions. FFT amplitudes of hull pressure fluctuation were also compared with simulation data for the 1st, 2nd and 3rd order of propeller BPF (Blade Passing Frequency). The objective of the present work is to investigate applicability of full-scale numerical simulation at the propeller design stage considering practical use of the result.

* Corresponding Author: Jung-yun Won, jungyunw@hhi.co.kr

2. Measurement condition for full-scale vessels

Full-scale measurement data of four vessels (Container ship A, B and Tanker A, B) were utilized for the comparison. Measurement conditions are listed in Table 1. Pressure sensors penetrating hull from the inside to the outside were installed for tanker A and B, while magnetic type sensors were attached to the outside of the hull for container A. The vertical upward position above the propeller was set as the datum point 'Center (C)' and sensors were installed at a distance of 25% of the propeller diameter in the forward (F), portside (P) and starboard side (S) direction from the Center (C). To observe the cavitation behavior, observation windows were installed one on the port and another on the starboard side.

Table 1. Conditions of full-scale measurement

Vessel Name	**Container A**	**Container B**	**Tanker A**	**Tanker B**
Diameter x Number of blades	8.7m x 5	9.9m x 5	9.1m x 4	10.7m x 4
Engine Power (% of MCR)	90%	85%	79.6%	97.7%
Position for pressure sensors	C, S, F	-	P, C, S, F	P, S

3. Numerical simulation set up

A commercial CFD tool, Star-CCM +12.04 was used for the full-scale CFD simulations. IDDES (Improved Delayed Detached Eddy Simulation), an intermediate model between RANS and LES, was applied considering prediction accuracy and practical use of the computational power. Cavitation was modelled by the S-S (Schnerr-Sauer) model, which is simplified form of the R-P (Reyleigh-Plesset) model. Even if S-S model was found to be limited in predicting the dynamic characteristics of cavitation [4], it is good enough at capturing the general cavitation patterns. The computational region with hull, rudder and ESD(Energy Saving Device) was discretized by trimmer mesh. The 6-DOF (6 Degree of Freedom) motion, trim and free surface effect were not considered for the effective use of simulation. The rotational motion of the propeller was simulated using sliding mesh by 0.5deg per time step. Details about simulation condition are shown in Table 2. For the advance ratio J and thrust coefficient K_T, assumed data from model test were used.

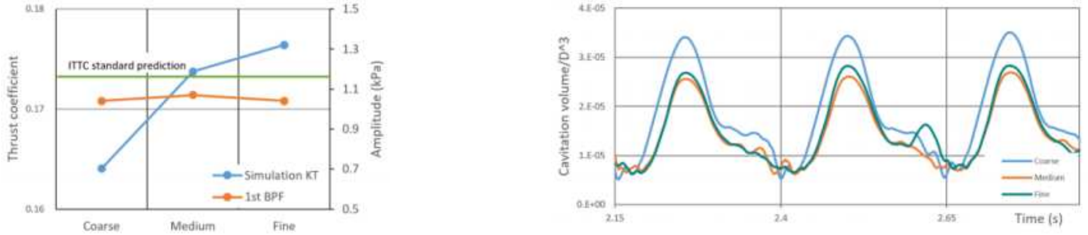
Table 2. Numerical simulation condition

Vessel Name		**Container A**	**Container B**	**Tanker A**	**Tanker B**
Cavitation Number ($\sigma_{0.7R} = \frac{P_{0.7R}-P_v}{0.5\rho n^2 D^2}$)		0.94	2.00	3.40	2.66
Reynolds number at 0.7R		10.57×10^7	6.41×10^7	4.71×10^7	4.11×10^7
Advance ratio ($J = \frac{Va}{nD}$)		0.7164	0.6479	0.5139	0.5015
Thrust coefficient ($K_T = \frac{T}{\rho n^2 D^4}$)		0.1709	0.1787	0.1542	0.1732
No. of cells	**Hull region**	About 8.0M cells by trimmer mesh			
	Prop. region	About 9.8M cells per blade by polyhedral mesh			
Wall Y+	**Hull**	~2000			
	Propeller blade	~500			

D: Propeller diameter, n: Propeller rotational speed, Va: Advance speed of propeller, T: Propeller thrust, Pv: Vapor pressure, $P_{0.7R}$: Pressure at 0.7radius above the propeller shaft center,

* Corresponding Author: Jung-yun Won, jungyunw@hhi.co.kr

The results of computational grid study are presented in Figure 1. (a) and (b). Thrust coefficient, K_T, is seemed to be converged at Fine mesh and the difference between Medium and Fine mesh was within 2%. The converged thrust coefficient was close to the predicted value by ITTC standard prediction from model test result. Pressure fluctuation amplitude of 1st BPF did not show significant difference depending on the number of grids. On the other hand, Cavitation volume seemed converged as the level of mesh increases from Coarse to Fine mesh and the volume of Fine mesh was slightly larger than Medium mesh because tip vortex cavitation volume was increased. Considering the result of grid dependency study, Fine mesh was applied to the simulation.



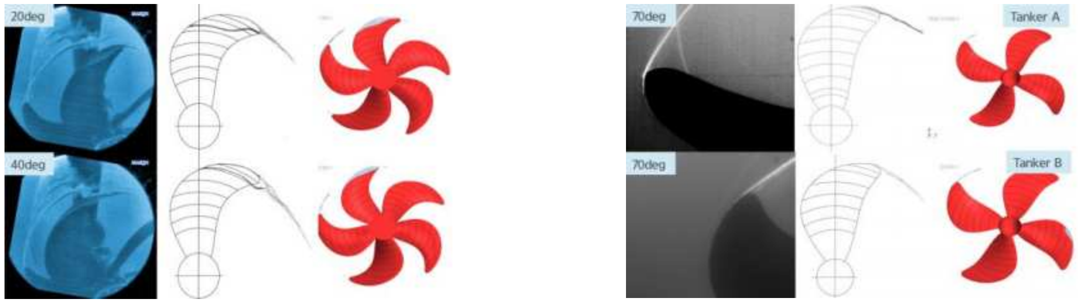
(a) (b)

Figure 1. (**a**) Thrust and hull pressure convergence study; (**b**) Convergence of cavitation volume

3. Results

3.1. Cavitation pattern observation

Cavitation pattern snap shots of videos taken by high-speed camera and numerical simulation results were compared at expected various propeller rotation angles. The snap shots, sketches and simulation results are presented in Figure 2. (a) for Container ship B and Figure 2. (b) for Tanker A, B. Container B shows large amount of sheet cavitation and strong tip vortex cavitation due to the high engine power and ship speed. The simulation result showed generally good agreement with observation data in shape and location of cavitation, but numerical simulation is considered to be short in capturing tip vortex cavitation. Unlike Container B, tanker vessels have small amount sheet cavitation at the tip of the blade, and similar behavior are shown in numerical results. However, even in the tanker vessels, tip vortex of numerical simulation is shorter and smaller than observation data. It is presumed that current numerical model is insufficient to reflect the complex behavior of tip vortex cavitation. The number of grids was set considering computational power, but it is expected that higher resolution near the tip is required to capture tip vortex more accurately.



(**a**) (**b**)

Figure 2. (**a**)Cavitation pattern comparison of Container B; (**b**)Cavitation pattern comparison of Tanker A, B

* Corresponding Author: Jung-yun Won, jungyunw@hhi.co.kr

3.2. Hull pressure fluctuation

Figure 3 shows the FFT amplitude of pressure pulse delivered to the hull. For the Tanker A, B, the numerical simulation predicted amplitude of pressure fluctuation with error of 0.4kPa for all position and the overall trend were also similar with measured data. The prediction error of container A was within 1kPa for the 1st order of pressure fluctuation, but larger error was found in the 2nd and 3rd order. As mentioned in the cavitation observation chapter, insufficient number of grid and simplified cavitation model can be the reason for the error of high order. In addition to these reasons, as pressure fluctuation of Container A was measured at lowest cavitation number and at highest ship speed and RPM in comparison with three other vessels, it is estimated that strong tip vortex cavitation is one of the reason for errors of high orders. Also, flow disturbance due to the pressure sensor of Container A is also considered as a reason as the magnetic type sensors were attached on the outside of the hull.

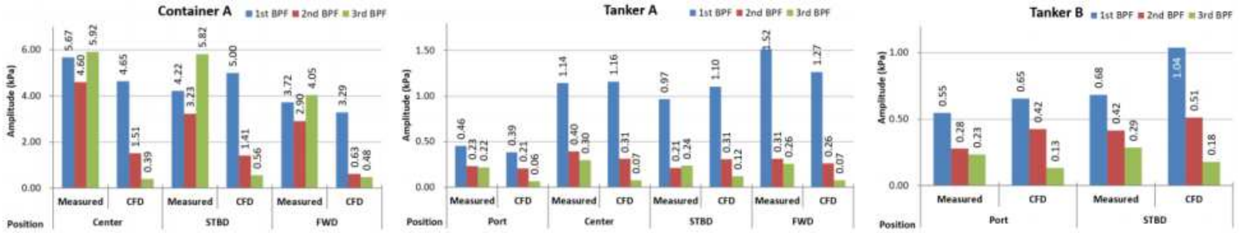


Figure 3. Hull pressure fluctuation for Container A, Tanker A and B

4. Conclusions

In this study, the cavitation behavior and hull pressure fluctuation were predicted by full-scale CFD simulation using IDDES and S-S cavitation model. For tanker ships, FFT amplitudes of pressure fluctuation were well predicted for the 1st, 2nd and 3rdorder of BPF within 0.4kPa difference and the cavitation behavior were similar to the observation data. In case of container ships, overall shape of sheet cavitation was similar with observation data and prediction error was within 1kPa for the 1st order BPF. However, very weak tip vortex cavitation is produced in the computation and relatively large errors were found in the 2nd and 3rd order BPF. Regarding the limitations of simulation, future work needs to be performed to improve prediction accuracy for tip vortex and high order BPF by using sufficient number of mesh near the tip and using higher order cavitation model. In addition, it is required to measure pressure fluctuation of high-speed vessels using plug type sensors considering possibility of flow disturbance.

From the study for numerical simulation at full-scale, it is expected that the full-scale CFD simulation can provide useful reference data for propeller design of tanker vessel.

References

1. Schuiling, B.; Lafeber, F. H.; Ploeg A.; Wijngarrden E. “The influence of the wake scale effect on the prediction of hull pressures due to cavitating propellers” Second International Symposium on Marine Propulsors SMP’11, Hamburg, Germany, June 2011.
2. Ponkratov D. DES prediction of cavitation erosion and its validation for a ship scale propeller. *Journal of physics: Conference series* **2015**, *Vol 656*, 012055.
3. Park, J.Y.; Peric, M.; Farrall, M.; Park, C. Numerical analysis for the prediction of hull pressure fluctuation and underwater radiated noise induced by marine propeller cavitation. Proceedings of the 10th international symposium on cavitation Baltimore, USA, May 2018.
4. Muzaferija, S.; Papoulias, D.; Peric. M. “VOF Simulations of hydrodynamic cavitation using the asymptotic and classical Rayleigh-Plesset models” Fifth International Symposium on Marine Propulsors SMP’17, Helsinki, Finland, June 2017.

* Corresponding Author: Jung-yun Won, jungyunw@hhi.co.kr