Sixth International Symposium on Marine Propulsors
smp'19, Rome, Italy, May 2019

THE NUMERICAL SIMULATION OF PROPELLER CAVITATION BENCHMARK TESTS OF YUPENG SHIP MODEL

Zheng Chaosheng[1], Liu Dengcheng[1], Huang Hongbo[1]

[1] China Ship Scientific Research Center (CSSRC) national key laboratory on ship vibration & noise, Jiangsu Key laboratory of Green Ship technology, Wuxi, Jiangsu, China

ABSTRACT

The numerical simulation of propeller cavitation benchmark tests of YUPENG ship model is studied based on OpenFOAM, and the benchmark tests are introduced as well. The propeller cavitation shape and the hull pressure fluctuation are measured and predicted respectively. The uncertainty in hull pressure fluctuation measurement is also analyzed, and the analysis showed the uncertainty is below 10%. The cavitation shape and the hull pressure fluctuation predicted show good agreement with the experiment observations and measurement.

Keywords

Numerical simulation, propeller cavitation, benchmark tests, YUPENG

1 INTRODUCTION

As one important aspect of propeller performance, the propeller cavitation has been studied by numerical simulation method based on viscous RANS approach, and more and more research papers has been published, recently. Da-Qing Li (2012) has simulated the E779A propeller cavitation in non-uniform wake based on RANS approach and Zwart cavitation model using FLUENT. Kwang-Jun Paik (2013) has predicted the propeller cavitation pattern and the hull pressure fluctuation induced, using FLUENT and SchnerrSauer cavitation model. Keita Fujiyama (2017) has studied the unsteady cavitating flow of HSP-II and CP-II propeller at behind-hull condition using SC/Tetra v13, both in model and full scale.

In addition to commercial CFD software, OpenFOAM, the open-source CFD platform, has been increasingly popular in the numerical simulation of propeller cavitation. Abolfazl (2015) has predicted the PPTC propeller cavity extent within a 12°inclination of shaft using ILES method and SchnerrSauer cavitation model based on OpenFOAM. Rickard E Bensow (2015) has studied the cavity extent, flow field and forces on the propeller of a 7000 DWT chemical tanker, with ILES method and Kunz cavitation model adopted in OpenFOAM. Zheng Chaosheng (2016, 2017) has investigated the unsteady propeller cavitation in the ship stern using RANS method and OpenFOAM, and the cavitation shape change shows good agreement with the experiment observations.

The present work aims to predict the unsteady propeller cavitation and the hull pressure fluctuation in the stern region of YUPENG benchmark ship model, which will further verify the applicability of the numerical simulation method.

2 NUMERICAL METHODS AND MODELS

2.1 Governing equations

The unsteady RANS approach is adopted because of the significantly lower computational effort than LES.

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

$$\frac{\partial \rho U}{\partial t} + U \cdot \nabla(\rho U) = \nabla \cdot \left((\mu + \mu_t)\nabla U\right) - \nabla p - F_s \tag{2}$$

The turbulent viscosity μ_t is modeled by the SST $\kappa\omega$ turbulence model together with wall functions.

In the VOF approach, the physical properties of the fluid are scaled by the liquid volume fraction γ, with $\gamma = 1$ corresponding to pure water.

The density and dynamic viscosity of the fluid are scaled as

$$\rho = \gamma \rho_l + (1-\gamma)\rho_v \tag{3}$$

$$\mu = \gamma \mu_l + (1-\gamma)\mu_v \tag{4}$$

The mass transfer equation of the liquid volume fraction γ can be written as

$$\frac{D\gamma}{Dt} = \frac{\partial \gamma}{\partial t} + \nabla \cdot (\gamma U) = \frac{\dot{m}}{\rho_l} \tag{5}$$

where the mass transfer rate $\dot{m}$ is to be modeled by cavitation models.

The transport Equation (1) for mass (continuity equation) can be re-written as bellow in combination with Equation (1), Equation (3) and Equation (5).

$$\nabla \cdot U = \dot{m}\left(\frac{1}{\rho_l} - \frac{1}{\rho_v}\right) \tag{6}$$

It implies that the cavitation model is involved in the coupling procedure of velocity and pressure in Equation (6).

2.2 Cavitation model

Cavitation is the transition of liquid into vapour in the low pressure regions caused by the presence of small gas nuclei in the fluid. The SchnerrSauer cavitation mass transfer model is employed to mimic the phase change between vapour and liquid, and already implemented in OpenFOAM.

$$\dot{m} = sign(p_v - p)\frac{n_0}{1 + n_0 \frac{4}{3}\pi R^3} 4\pi R^2 \sqrt{\frac{2}{3}\frac{|p_v - p|}{\rho_l}} \tag{7}$$

where n_0 stands for the number density of micro bubbles per liquid volume and R is the initial nuclei radius. SchnerrSauer's model is based on bubble dynamics by considering the motion equation of a single bubble of radius R.

2.3 Discretization and solution procedure

The finite volume method is used for the discretization of the governing equations, and the unsolved flow variables are stored in the cell-centre positions in the computational grid. The Euler differencing scheme is used for the time discretization, and a second order differencing scheme is adopted for the components of the momentum equation.

The interPhaseChangeDyMFoam solver used in this study is a multiphase solver, taking two fluids into account using the VOF method.

To improve the convergency of the cavitation flow and reduce the computational time, the full wetted flow is simulated using MRF method at first, after obtaining a quasi-stable flow field, the sliding mesh is then applied to simulate the rotation of propeller. The three components of the momentum equation are solved sequentially in a loop within each time step. The PIMPLE algorithm is applied for the coupling between the velocity and the pressure fields, allowing for stable transient simulations with $\max Co \geq 1$. The PIMPLE algorithm is a combination of the SIMPLE and PISO algorithms, where the PISO loop is complemented by an outer iteration loop and the under-relaxation of the variables.

3 BOUNDARY CONDITIONS AND MESH GENERATION

The configuration investigated here is YUPENG benchmark ship, which is an 30000 DWT bulk carrier, a single screw vessel driven by a fixed pitch four-bladed propeller. The benchmark test of cavitation observation and measurement for the model propeller is performed in the large cavitation channel of China Ship Scientific Research Center. The principle parameters of model propeller is provided in Table 1, and the hull and propeller are shown in Figure 1.

Table 1 The principle parameters of model propeller

Propeller diameter, D	0.250m
Pitch-diameter ratio, P/D	0.8032
Skew angle	24.5°
Expanded area ratio, EAR	0.480
Number of blades, Z	4
Scale ratio, λ	26.8

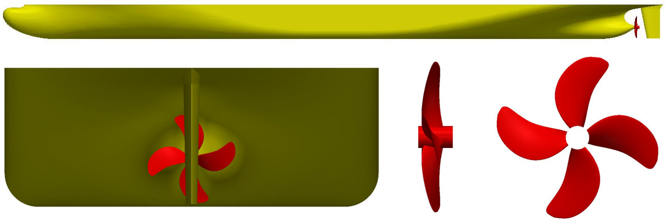


Figure 1 The YUPENG ship and propeller

The experiment setup inside the cavitation tunnel is shown in Figure 2. The initial rotation angle φ is defined as 0° at 12 o' clock.

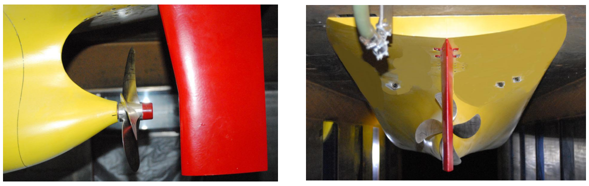


Figure 2 The experiment setup

The whole computational domain is divided into two sub-regions. The ship region contains the flow region that includes the inlet, outlet, ship and rudder, the propeller region is a small cylinder surrounding the propeller. The grid of the two sub-regions all consist of unstructured hexahedral cells, and a number of the boundary layer cells are inserted, the non-dimensional parameter of the hull, Y+ ≈ 80. The ship region and propeller region are consist of 6.79 million and 1.09 million cells, respectively. The overview of surface mesh is shown in Figure 3. The interpolation between the non-conforming interfaces of the two sub-regions is implemented in OpenFOAM, named as AMI (Arbitrary Mesh Interface).

The inlet boundary is defined as velocity inlet condition with a fixed value of velocity U. The outlet boundary defined as pressure outlet condition, and the pressure value is constant based on the cavitation number $\sigma_{n0.8R}$. The hull, propeller, rudder and hub boundary are defined as no-slip wall conditions, respectively.

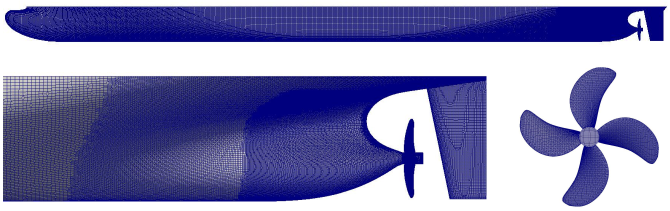


Figure 3 The overview of surface mesh

4 RESULTS AND DISCUSSIONS

The test conditions of the cavitation observation and measurement in this study is summarized in Table 2.

Table 2 The test conditions

	Design draft
n	28rps
$\sigma_{n0.8R}$	0.3154
KT	0.1371

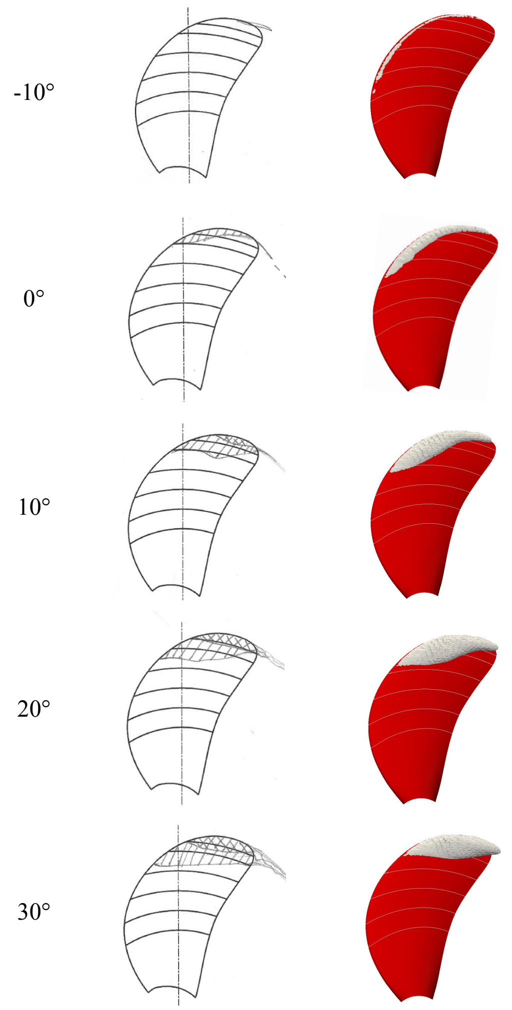
where n is the rotational speed of the propeller, $\sigma_{n0.8R}$ is the cavitation number, and K_T is the thrust coefficient. The definitions are as follows:

$$\sigma_{n0.8R} = \frac{p - p_v}{0.5\rho(0.8\pi nD)^2} \tag{8}$$

$$K_T = \frac{T}{\rho n^2 D^4} \tag{9}$$

Firstly, the steady full wetted flow is simulated using MRF method to obtain a quasi-stable flow field, then the unsteady computation is started to simulate the rotation of propeller using sliding mesh method, and the cavitation model is activated to predict the cavitation, finally.

The predicted cavitation in the stern region at design draft is compared with the experiment sketches side-by-side in Figure 4.



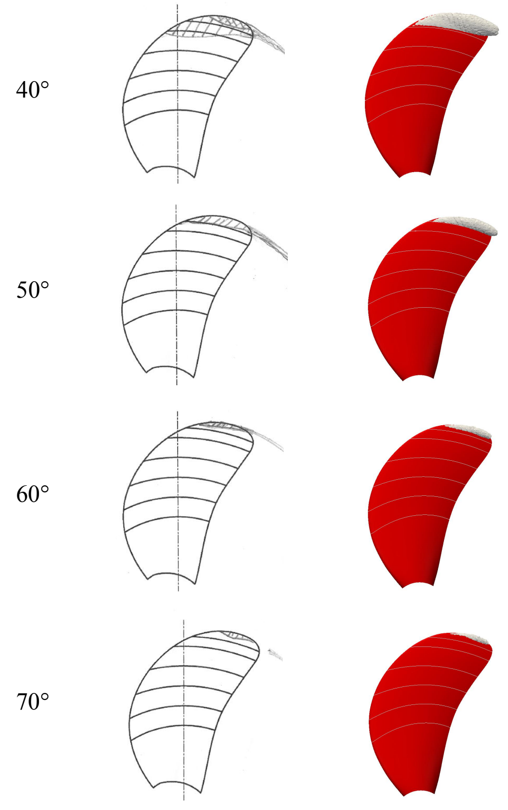


Figure 4 The experiment sketches vs. prediction at design draft condition

In Figure 4, the predicted cavity, represented by vapour iso-surface of 0.1 shows the same behaviour as the experiment observations. The key feature, the extent change of the attached cavity with the rotation angles and the collapse at the tail of the main cavity, correlates well with the experiment, e.g. the cavity begins at about the same location $\varphi \approx$-10°, reaches the maximum area at $\varphi \approx$30°. Nevertheless, the tip vortex cavity cannot be predicted on account of the insufficient mesh resolution around the blade tip.

In order to investigate the hull pressure fluctuation induced by cavitation, the monitor points are arranged on the stern surface shown in Figure 5.

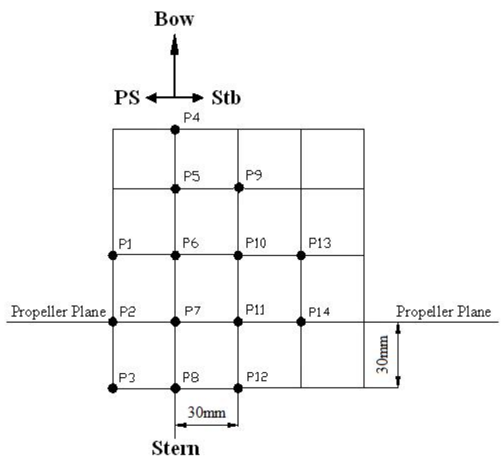


Figure 5 The arrangement of monitor points

The pressure fluctuation predicted at model scale is analysed by FFT, and converted to the pressure fluctuation at full scale by the formula.

For the uncertainty analysis, the hull pressure fluctuation measurement has been repeated three times, and the maximum value of the 1BF pressure fluctuation is at P10 each time. The uncertainty contribution of the measurement, the combined uncertainty and the expanded uncertainty are counted in Table 3, respectively. The analysis shows that the expanded uncertainty is below 10%.

Table 3 The expanded uncertainty in measurement for hull pressure fluctuation (at P10)

$\Delta p'(\sigma_{n(0.8R)}=0.3154)$	Type	Uncertainty	Remark
Propeller diameter	B	0.04%	negligible
Operating pressure	B	0.11%	negligible
Saturated vapor pressure	B	0.29%	negligible
Water density	B	0.001%	negligible
Propeller rotate speed	B	0.02%	negligible
Dynamic pressure transducer	B	3.33%	dominant
Repeat test (N=3)	A	0.70%	minor
Combined uncertainty		3.42%	
Expanded uncertainty		6.83%	

Regarding the numerical simulation results, the amplitudes of the first blade frequency (1BF) of the hull pressure fluctuation predicted at design draft condition is compared with the experiment in Figure 6. The prediction shows good agreement with the test measurement, and the spatial laws of the pressure fluctuation is predicted well, and the maximum value of the 1BF pressure fluctuation predicted is also at P10.

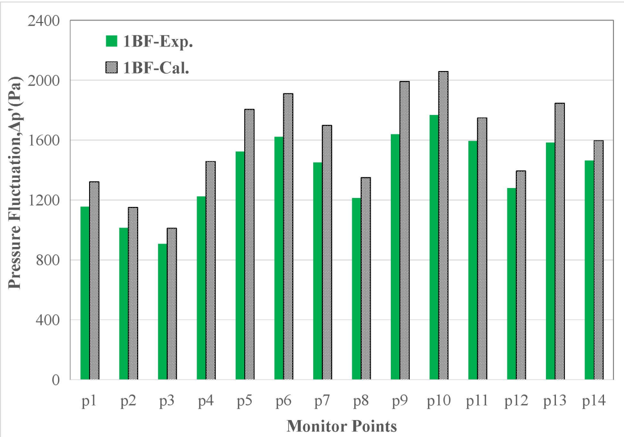


Figure 6 The 1BF of hull pressure fluctuation predicted vs. the experiment at design draft condition

Furthermore, the second and third blade frequency are compared with the experiment in Figure 7 and Figure 8, respectively. The obvious differences indicate that the instability of the propeller cavitation at the same angle in different period, i.e. the cavity shedding at high frequency is not well captured, and needed to be further studied in the future.

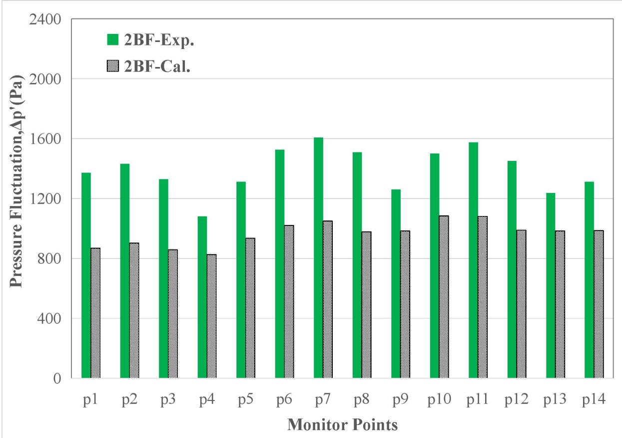


Figure 7 The 2BF of hull pressure fluctuation predicted vs. the experiment at design draft condition

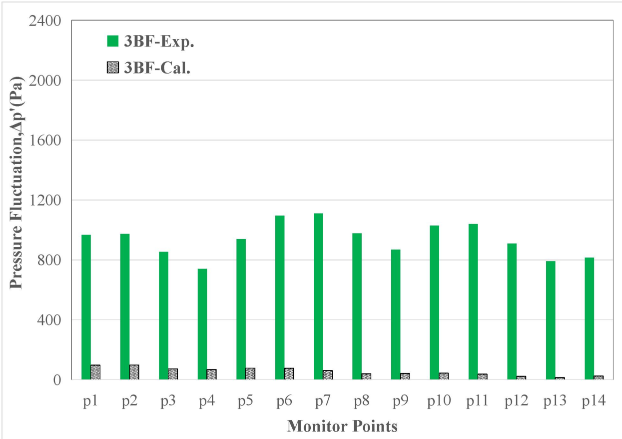


Figure 8 The 3BF of hull pressure fluctuation predicted vs. the experiment at design draft condition

5 CONCLUSIONS

In the stern region of YUPENG benchmark ship model, the unsteady propeller cavitation shape and the amplitudes of the 1BF of the hull pressure fluctuation predicted resemble well with observations and measurements in benchmark test, which further validate the reliability and applicability of the numerical simulation method.

ACKNOWLEDGMENTS

This work is financially supported by National Natural Science Foundation of China (Project NO. 11332009).

REFERENCES

Da-Qing Li. (2012) . 'Prediction of Cavitation for the INSEAN Propeller E779A Operating in Uniform Flow and Non-Uniform Wakes'. 8th International Symposium on Cavitation, Singapore, pp.368-373

Kwang-Jun Paik. (2013) . 'URANS Simulations of Cavitation and Hull Pressure Fluctuation for Marine Propeller with Hull Interaction'. 3nd International Symposium on Marine Propulsors, Tasmania, pp.389-396

Abolfazl Asnaghi. (2015) . 'Computational Analysis of Cavitating Marine Propeller Performance using OpenFOAM'. 4th International Symposium on Marine Propulsors, Tesas, pp.148-155

Rickard E Bensow. (2015) . 'Large Eddy Simulation of a Cavitating Propeller Operating in Behind Conditions with and without Pre-Swirl Stators'. 4th International Symposium on Marine Propulsors, Tesas, pp.458-477

Zheng Chaosheng. (2016) . 'The unsteady numerical simulation of propeller cavitation behind a single-screw transport ship using OpenFOAM'. 2nd Conference of Global Chinese Scholars on Hydrodynamics, Wuxi, pp.117-123

Zheng Chaosheng. (2017) . 'The numerical prediction of the propeller cavitation and hull pressure fluctuation in the ship stern using OpenFOAM'. 5th Symposium on Marine Propulsors Proceedings, SMP'17, Espoo, Finland, pp.745-749

Keita Fujiyama. (2017) . 'Numerical Prediction of Acoustic Noise Level Induced by Cavitation on Ship Propeller at Behind-Hull Condition'. 5th Symposium on Marine Propulsors Proceedings, SMP'17, Espoo, Finland, pp.739-744