Proceedings of the 7th International Symposium on Cavitation
CAV2009 – Paper No. 68
August 17-22, 2009, Ann Arbor, Michigan, USA

Study on Unsteady Cavitating Flow Simulation around Marine Propeller using a RANS CFD code

Koyu Kimura
Akishima Laboratories (Mitsui Zosen) Inc.
Tokyo, Japan

Takafumi Kawamura
The University of Tokyo.
Tokyo, Japan

Akihiko Fujii
Mitsui Engineering & Shipbuilding Co., Ltd.
Tokyo, Japan

Tadashi Taketani
Akishima Laboratories (Mitsui Zosen) Inc.
Tokyo, Japan

Zhenchuan Huang
The University of Tokyo.
Tokyo, Japan

ABSTRACT

Reduction of greenhouse gas (CO2 etc) emission is an important issue to address global warming. In recent ship design, it is essential to improve propulsive performance and fuel oil consumption, and the demand for high-efficiency propeller is growing rapidly.

The authors have been investigating the possibility of the application of CFD to the propeller performance evaluation and optimization. In these previous papers[1,2], the authors presented CFD simulation of non-cavitating and cavitating flow around a marine propeller using a commercial CFD code. A good agreement with the experiment was confirmed for the non-cavitating flow.

Various validations were also carried out for the cavitating flow, and the followings were described. First, we confirmed that the cavity shape in a uniform flow was qualitatively well estimated, but the difference between two propellers, of which the blade sections were somewhat different, were not reproduced. Secondary, the cavity shape in the non-uniform flow was also qualitatively well estimated, but the resulting pressure fluctuation was not validated.

In this paper, the systematic experiment was carried out using two marine propellers, which dimensions were very similar, to study the above issues, and simulation was carried out for the same cases. In the uniform cavitating flow simulation, the difference of cavity shape around these propellers was reproduced, and the quantitative validation of the fluid force such as thrust and torque was done. In the non-uniform cavitating flow simulation, the comparison of the cavity shape with the experiment and the quantitative validation of the fluctuating pressure on the wall of the cavitation tunnel were done.

INTRODUCTION

Propeller design requires careful consideration of the cavitation generated around propellers, since this phenomenon can lead to various problems, including performance decrease, noise, vibration, and erosion. Currently, propeller design is based on design charts and theoretical analysis, with estimates of cavitation properties based on model testing in cavitation tunnel. Given the growing demand for higher-performance propeller, we anticipate the development of CFD-based simulations capable of precisely predicting propeller efficiency and cavitation behavior in the propeller design stage.

Former studies[3,4,5] describe CFD simulations are as accurate as model testing in predicting marine propeller performance. Nevertheless, CFD simulations encounter difficulties reproducing differences in the cavitation behavior of propellers having slight differences in blade-sectional shapes et al [5]. Development of CFD simulation capable of reproducing such minute differences is crucial subject for propeller design.

In this study, we carried out a series of model tests on two types of practically high-skew marine propellers having similar dimensions in uniform and non-uniform flow. Based on the experimental data obtained, we performed CFD simulations and comparison between the results with respect to usefulness for propeller design.

EXPERIMENTAL SET UP AND MODEL PROPPELERS

The several experiment was carried out in Akishima Laboratories (Mitsui Zosen) Inc. The propeller open tests (POT) in uniform flow were performed at the large towing tank. Figure 1 shows the experimental set up for cavitation tests, eight pressure sensors were situated above the propeller and the fluctuating pressure was measured.

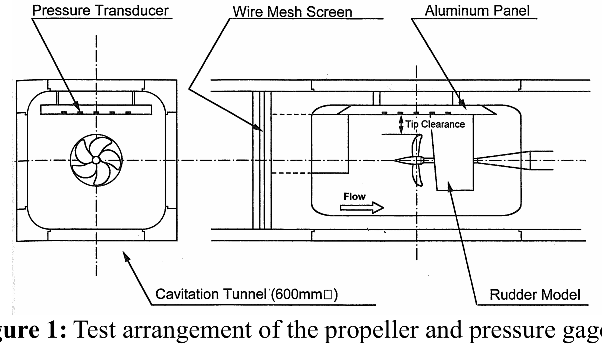


Figure 1: Test arrangement of the propeller and pressure gages

The cavitation number and the inflow velocity were systematically varied in the uniform and the non-uniform flow. The wake in the non-uniform flow was simulated by a wire mesh screen. The propeller Reynolds number Rn defined by equation(1), was $Rn(D)$=6 x 10^5 in POT; approx. $Rn(D)$=1.5 x 10^6 in uniform flow; and approx. $Rn(D)$=1.7 x 10^6 in non-uniform flow in cavitation testing (with n, D, ν, respectively, representing propeller revolution, propeller diameter, and the kinematic viscosity coefficient).

$$R_n = \frac{nD^2}{\nu} \tag{1}$$

Table 1 and Figure 2 show principal particular and photos of the two types of high-skew marine propellers used in several model tests. The two propellers are quite similar, with slight differences in expanded area ratio. Figure 3 shows non-dimensional pitch distribution normalized by maximum pitch of each propeller. As tip-loaded type propeller, P406R has a higher non-dimensional pitch distribution than P407R around propeller tip.

Table 1: Principal particular of the model propellers

MP No.	P406R	P407R
Number of blades	5	5
Diameter [m]	0.250	0.250
Boss ratio	0.1814	0.1814
Extension area ratio	0.7650	0.7300
Skew angle	36°	36°

(a) P406R (b)P407R

Figure 2: Pictures of two model propellers

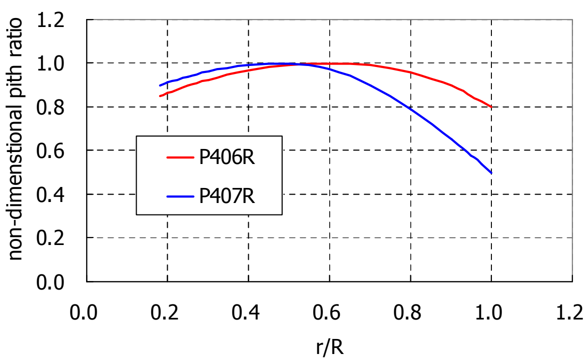


Figure 3: Comparison of the pitch distribution between two model propellers

CALCULATION METHOD AND CONDITIONS

We used Fluent ver.6.3, a commercially thermal flow analysis software based on non-structured mesh and finite volume methods, to perform calculation of non-cavitating and cavitating flow around the marine propellers. Fluent is considered useful in marine propeller analysis[1,2,3,4,5,6] because it installed a low-Reynolds 2-equation turbulent model, a sliding mesh method, and cavitation models.

We used the SIMPLE method for steady simulation and the PISO method for unsteady state simulation. For cavitation calculations, we used the unsteady method even for uniform flows to stabilize calculations. We used the QUICK scheme to estimate convection term and the second center differential scheme to estimate the characteristics of other phenomena. The k-w SST model was used as the turbulent model.

For the calculation of propeller open test(POT) and steady cavitating in uniform flow, single propeller blade simulation with periodic boundary condition was applied. Figure 4 shows the computational domain around the single propeller blade with periodic boundary condition.

For the calculation of non-uniform cavitating flow, full propeller simulation with sliding-mesh technique was applied. Figure 5 shows the computational domain around full propeller with similar dimension of test section of cavitation tunnel.

The mesh generation was applied method of Kawamura et al.[6], generating square and triangle meshes on blade surfaces and using prism meshes to resolve boundaries. The tetrahedral meshes were generated outside the prism area. The minimum mesh interval was set to about 1.0 in viscous length to discern the boundary layer. In particular, we partitioned the area close to the blade surface profile into square meshes to increase the resolution for the blade leading and trailing edges. Figure 6 shows the calculation meshes. The number of meshes used was approx. 1.3 million for each of the two types of propellers.

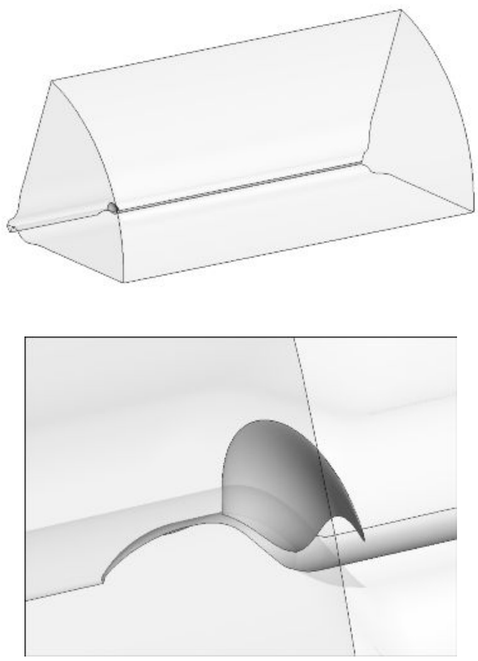


Figure 4: Computational domain of the single propeller blade simulation with periodic boundary condition

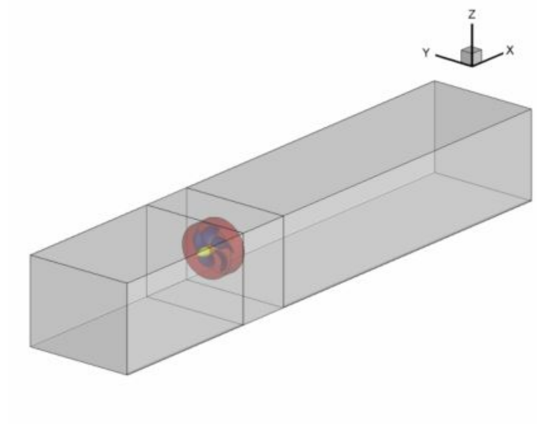


Figure 5: Computational domain of the full propeller simulation with sliding-mesh technique

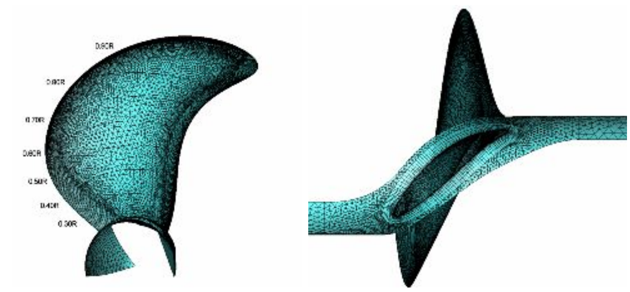


Figure 6: Mesh on the propeller blade

The cavitation model in FLUENT was applied in the present study. It is based on the so-called "full cavitation model" by Singhal et al.(2002)[7]. This model accounts for actual physical phenomena such as phase change, turbulence, and non-condensable gas (NCG). The model expresses phase change by a formula based on bubble dynamics and accounts for turbulence pressure fluctuations and the existence of non-condensable gases. The partial mass ratio, a model parameter, was set to 1.0 x 10^{-6}.

SIMULATION OF PROPELLER OPEN TEST (POT)

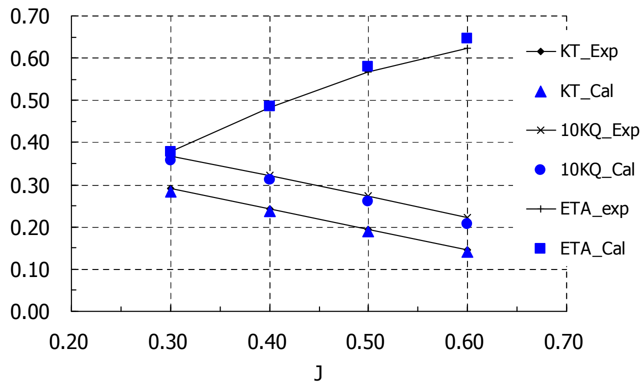
The simulation of propeller open test (POT) was performed to confirm the accuracy of steady flow calculations by present method. Figure 7 shows the calculation results. The calculation and experimental results agree well for thrust and torque, indicating that the simulations have sufficiently accurate to estimate the performance differences between the practically propellers. The parameters are shown below. V_a is propeller advance velocity, ρ fluid density, T thrust, and Q torque.

advance velocity coefficient
$$J = \frac{V_a}{nD} \tag{2}$$

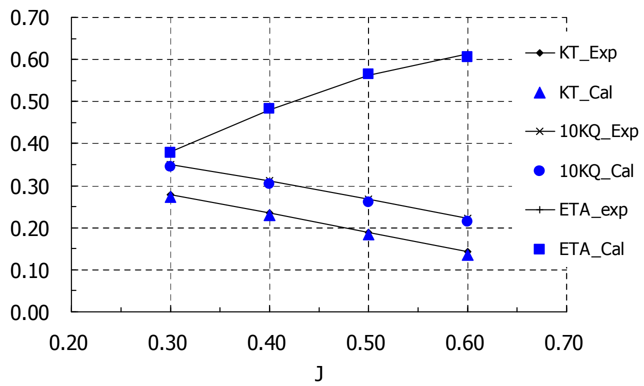
Thrust coefficient
$$K_T = \frac{T}{\rho n^2 D^4} \tag{3}$$

Torque coefficient
$$K_Q = \frac{Q}{\rho n^2 D^5} \tag{4}$$

propeller efficiency
$$\eta_o = \frac{J}{2\pi}\frac{K_T}{K_Q} \tag{5}$$



(a) P406R



(b) P407R

Figure 7: Results of POT simulation

UNIFORM CAVITATING FLOW SIMULATION

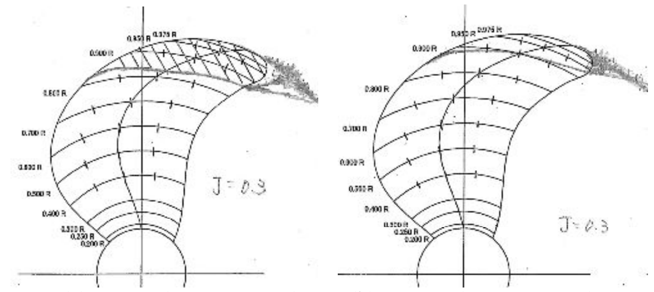
The uniform cavitation test was carried out in cavitation number σ (defined below) to 1.8, 1.2 and performed a test while changing advace velocity coefficient J, where P_∞ and P_v are the freestream and vapor pressures.

$$\sigma = \frac{P_\infty - P_v}{1/2 \cdot \rho n^2 D^2} \tag{6}$$

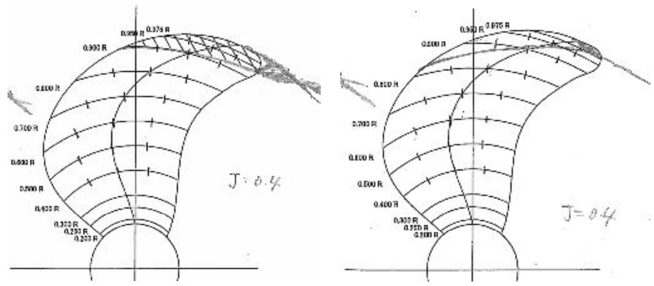
The cavitation sketches were shown in Figures 8. P406R has a larger cavity area around propeller tip; P407R is more likely to generate cavitation around leading edge of blade. Main cause of these behaviors is attributed to the different distribution of the pitch shown in Fig.2. Since P406R has a higher pitch around tip, it was assumed that tip loading is high, that the negative pressure peak is high near the tip of propeller, that cavitation becomes relatively significant near the tip, and that cavity area and volume increase. However P407R has a relatively lower pitch around tip and high loading on the blade central region, the pressure peak increases close to the central region. The area of cavitation generation extends toward the propeller boss, while cavitation generation around the tip of propeller is relatively weak.

Figure 9 shows comparison of the pressure distribution on the blade surface in non-cavitating flow. The negative pressure occurred stronger in the P406R at r=0.9R. P407R showed a higher peak at the leading edge of blade at r=0.7R.

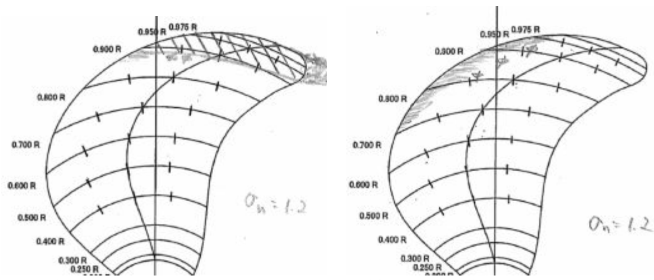
Figure 10 shows simulation results for steady cavitation flow in uniform flow. These figures show the iso-surface of vapor volume fraction of 10% and pressure distributions at the blade surface and boss surface. Poor mesh resolution for the tip vortex resulted in underestimates for tip vortex cavitation, but the cavity shape on the blade surface is well-reproduced. In particular, the calculation results and model test results agree quite closely with respect to the two characteristics above, and cavitation volume is also well-reproduced. These results confirm that present simulation is capable of reproducing with sufficient accuracy the cavitation behavior due to differences in the geometry of practically high-skew propellers.



(a)P406R(J=0.3,σ=1.8) (b) P407R(J=0.3,σ=1.8)

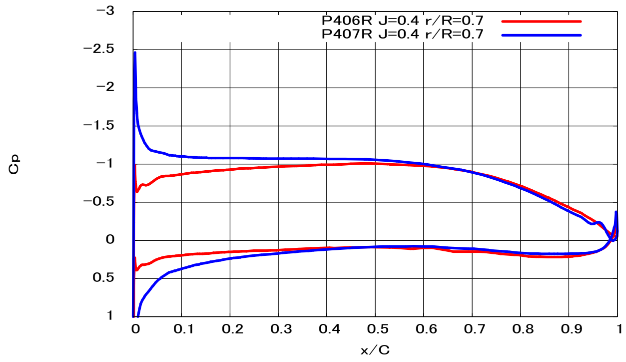


(c)P406R(J=0.4,σ=1.8) (d) P407R(J=0.4,σ=1.8)

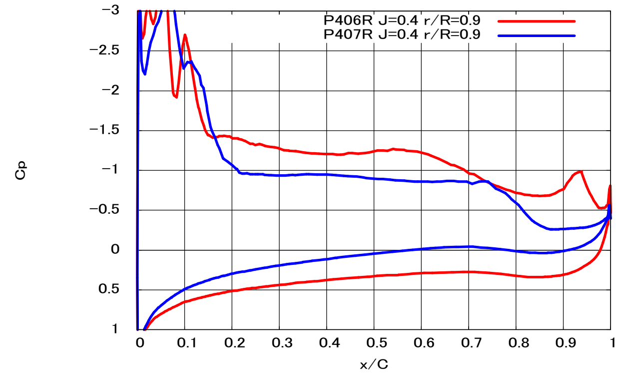


(e)P406R(J=0.5,σ=1.2) (f) P407R(J=0.5,σ=1.2)

Figure 8: Comparison of cavity sketch by experiment in uniform flow

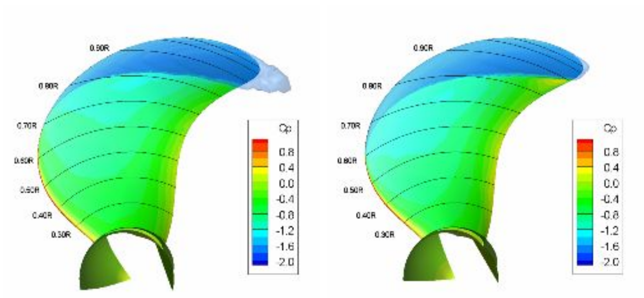


(a) J=0.4, r=0.7R

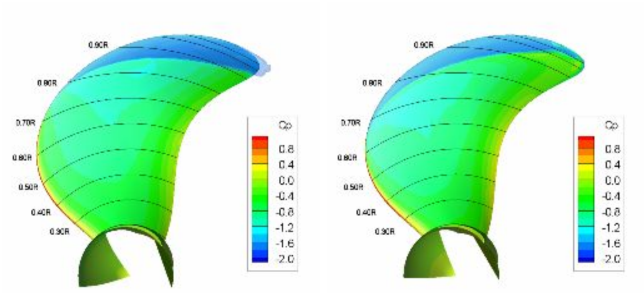


(b) J=0.4, r=0.9R

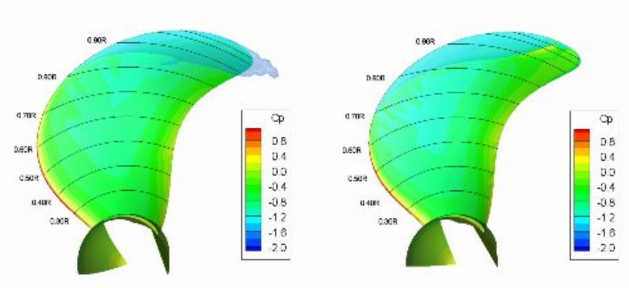
Figure 9: Comparison of the pressure distribution on the blade surface in non-cavitating flow



(a)P406R(J=0.3,σ=1.8) (b) P407R(J=0.3,σ=1.8)



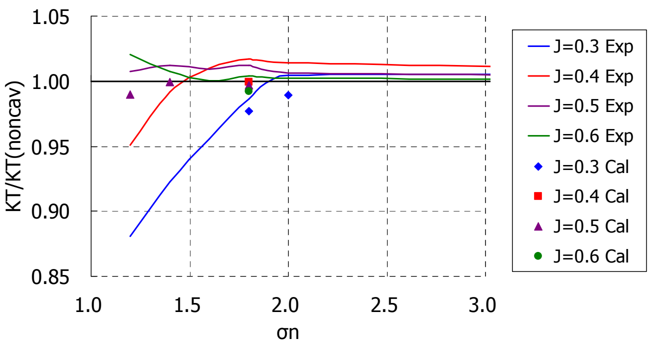
(b)P406R(J=0.4,σ=1.8) (d) P407R(J=0.4,σ=1.8)



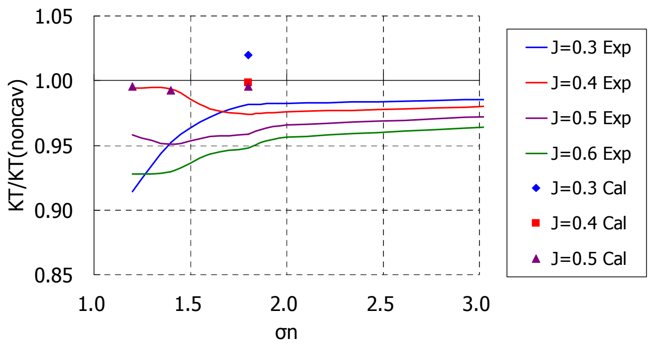
(e)P406R(J=0.5,σ=1.2) (f) P407R(J=0.5,σ=1.2)

Figure 10: Comparison of cavity sketch by calculation in uniform flow

Furthermore, keeping advanced velocity coefficient J constant, the test was carried out in which the cavitation number was continuously changed. Figure 11 shows the test results. The lateral axis represents cavitation number, while the vertical axis represents the ratio of thrust coefficients of the cavitating condition and non-cavitating condition. The lines are test results, while the plots are calculation results. They show that thrust decreases with lower cavitation numbers. This trend is especially marked at J=0.3 with the P406R, probably because a thrust breakdown (cavitation-induced thrust reduction) occurs where cavitation is significant. Present calculations were able to reproduce this phenomenon qualitatively. Generally, thrust breakdown for the purposes of propeller design has been estimated by empirical method, including charts and model tests. Consequently, CFD simulation was also useful for propeller design.



(a) P406R



(b) P407R

Figure 11: Comparison of thrust breakdown test

NON-UNIFORM CAVITATING FLOW SIMULATION

The non-uniform cavitating flow simulation was carried out using a wake distribution as shown in figure 12.

Figure 13 shows the comparison of pressure distribution on the blade surface in non-cavitating condition. Figure 14 shows the comparison of pressure fluctuation in non-cavitating condition. In non-cavitating condition, the characteristic difference of pressure distribution on blade surface and pressure fluctuation was reproduced due to the difference between two model propellers. However, the quantitative evaluation of pressure fluctuation was not sufficient to ensure accurate. The reason for this is that the accuracy of simulated wake distribution in computational domain was not sufficient compared with experiment.

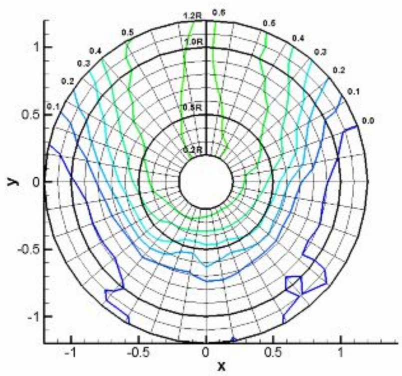
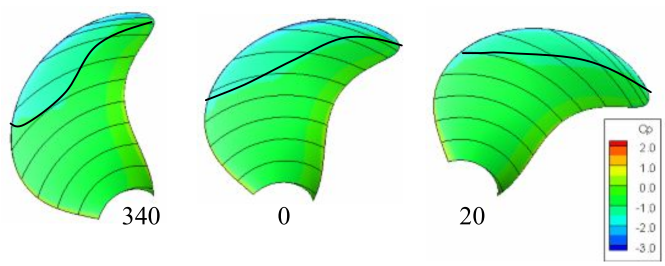
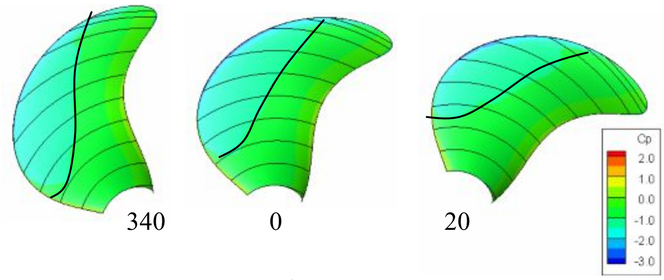


Figure 12: Simulated wake distribution

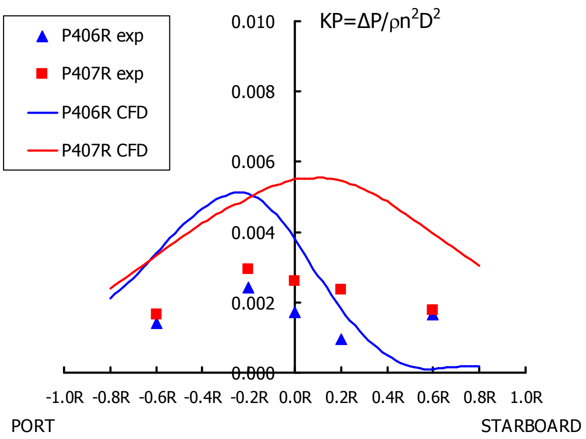


(a) P406R

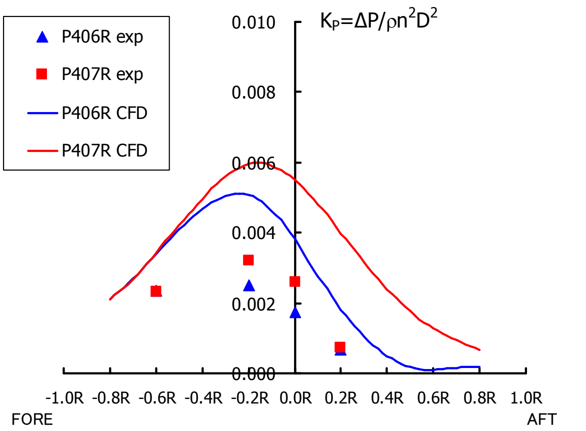


(b) P407R

Figure 13: Comparison of the pressure distribution on the blade surface in non-cavitating flow（kt=0.17）

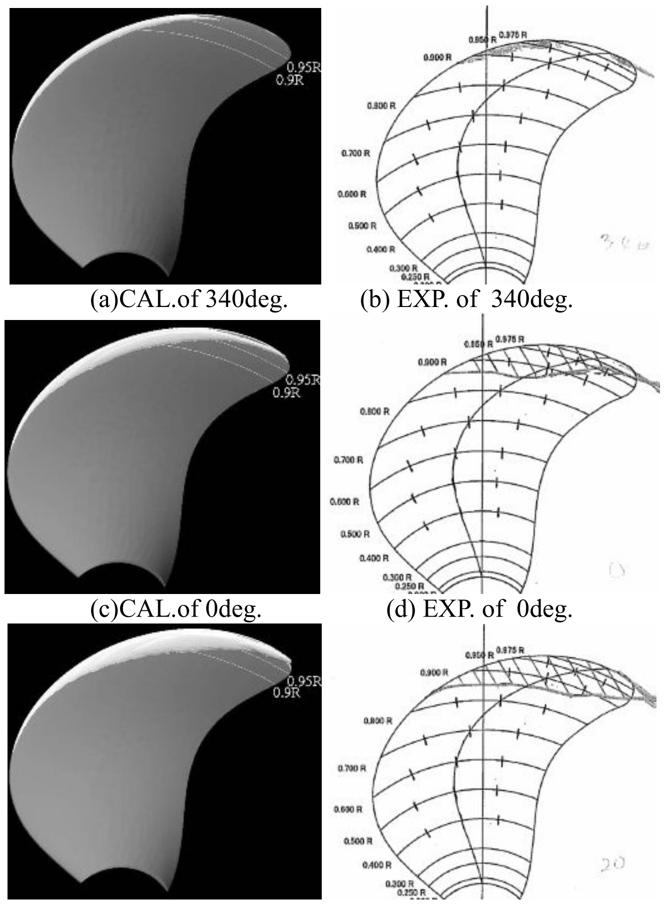


(a) port-starboard



(b) fore-aft

Figure 14: Comparison of the pressure fluctuation in non-cavitation flow(kt=0.17)



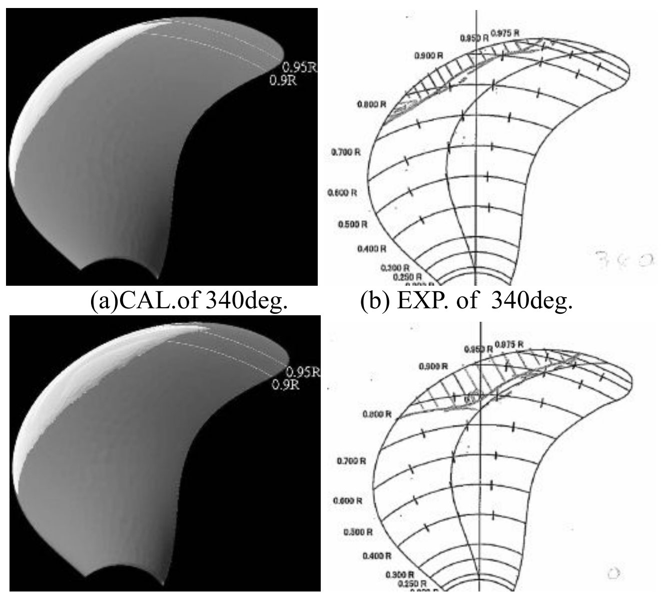
(a)CAL.of 340deg. (b) EXP. of 340deg.
(c)CAL.of 0deg. (d) EXP. of 0deg.
(e)CAL.of 20deg. (f) EXP. of 20deg.

Figure 15: Comparison of the cavity shape of P406R(kt=0.17, σ=1.8)

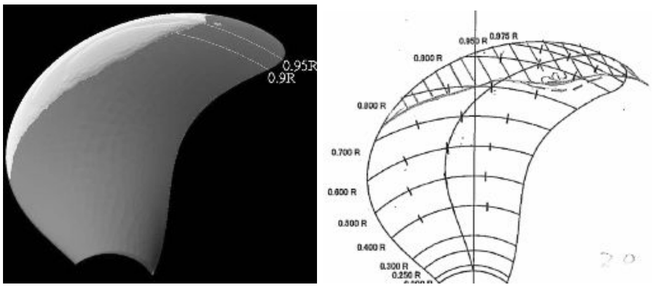
The comparison of cavity shape between experiments and calculations of two model propellers were shown in figure 15 and figure 16. The difference of cavity shape between two model propellers was reproduced similar as uniform cavitating flow simulation. However, the qualitatively difference of cavity shape (i.e. area, volume) between experiments and calculations was insufficient accuracy compared with uniform cavitating flow simulation. Therefore, the prediction of pressure fluctuation between two model propellers was not also reproduced compared with non-cavitating condition.

For this reason, the effect of the different wake distribution between the computational domain and cavitation tunnel wase mentioned above, furthermore, the current flow analysis method and the cavitation model would be need improvement about the non-uniform cavitating flow in wake field.

However, it was verified that the cavitation characteristics due to the difference geometry of practical high–skew propeller was qualitatively well estimated in unsteady flow. Finally, it was confirmed that this simulation is a useful tool in the practically propeller design.



(a)CAL.of 340deg. (b) EXP. of 340deg.
(c)CAL.of 0deg. (d) EXP. of 0deg.



(e)CAL.of 20deg. (f) EXP. of 20deg.

Figure 16: Comparison of the cavity shape of P407R(kt=0.17, σ=1.8)

CONCLUSIONS

We compared CFD calculations and a series of experiments with two similar types of practically high-skew propellers, obtaining the following results:

- The POT results can be precisely predicted, confirming the adequacy of present simulation meshing and method.
- In cavitation flow simulations, we successfully reproduced in qualitative cavity shape such as cavitation area, volume, and cavity leading-edge position based on differences in propeller geometry.
- CFD simulations are capable of qualitatively reproducing thrust breakdown (cavitation-induced thrust reduction).
- With accuracy sufficient for practical purposes, present CFD simulations can predict the details of efficiency and cavitation behaviors. And CFD simulations represent a useful tool for estimating propeller performance and optimization.

As future research, it is necessary to investigate the improvement of cvitation model to calculate the non-uniform cavitating flow and the qualitatively estimation of cavitation behavior in the design stage of propeller.

REFERENCES

[1] T. Watanabe et al. 2003, "Simulation of Steady and Unsteady Cavitation on a Marine Propeller using a RANS CFD code, " *CAV2003*, Osaka, Japan.

[2] Kawamura et al. 2006, "Simulation of Unsteady Cavitating Flow around Marine Propeller using a RANS CFD code," *CAV2006*, Wagenigen, Netherlands.

[3] Rhee, S.H. et al. 2003, "A Study of Propeller Cavitation using a RANS CFD Method," *The 8th International Conference on Numerical Ship Hydrodynamics*, Busan.

[4] Takafumi Kawamura et al. 2004, "Numerical Simulation of Cavitating Flow around a Propeller," *Journal of the Society of Naval Architects of Japan,* No.195, 211-219.

[5] Takafumi Kawamura et al. 2006, "RANS Simulation of Unsteady Cavitation around Marine Propellers," *The 13th Symposium on Cavitation, Sapporo*.

[6] Takafumi Kawamura et al. May 2008, "Numerical Prediction of Hull Surface Pressure Fluctuation due to Propeller Cavitation," *Proceedings of Japan Society of Naval Architects and Ocean Engineers*, Vol.6, 213-216.

[7] Singhal A.K. et al. 2002, "Mathematical Basis and Validation of the Full Cavitation Model," *Journal of Fluid engineering*, Vol. 124, No. 3, 617-624