

CFD Calculation of Cavitation in Outboard Motors

Ayaka NANGO*

Yuka IGA**

ABSTRACT

CFD calculations accounting for cavitation in outboard motor propellers were performed to save calculation cost and calculation accuracy. The impact of the boundary layer mesh resolution on the main propeller performance values of thrust coefficient, torque coefficient, and propeller efficiency was clarified. The obtained results of CFD calculation were qualitatively consistent with the results of model tests, and its accuracy was within $\pm 7\%$. With the above, cavitation effects can be accounted for from the early stages of outboard motor development using CFD calculation.

1. Introduction

Cavitation is known to occur in the propellers and gear case of outboard motors. As depicted in Fig. 1, the phenomenon of cavitation is when drops in pressure cause liquid to phase change into gas.

With some exceptions⁽¹⁾, cavitation is known to decrease performance⁽²⁾. Propeller cavitation degrades thrust and gear case cavitation increases resistance values, in turn decreasing boat speed and increasing fuel consumption of outboard motors. Furthermore, cavitation can also cause erosion, leading to degradation of appearance quality. Thus, it would be preferable to account for cavitation effects from the early stages of development.

In outboard motor fuel economy and power performance projections for Honda outboard motor development to date⁽³⁾, performance has been calculated using calculations ignoring cavitation. However, as outboard motors have increased in output in recent years, they are approaching

speed ranges high enough to generate cavitation. This increases the need for calculations which account for cavitation.

While the cavitation phenomenon has become computable using commercially-available Computational Fluid Dynamics (CFD) software^{(4), (5)}, no method of modeling cavitation has yet been established⁽⁶⁾. Generally speaking, increasing mesh resolution in CFD calculations will allow for cavitation to be predicted more accurately at the expense of increased computational time, and thus computation cost. Due to this factor, a balance should be struck between calculational accuracy and cost if CFD is to be used in the initial development phase.

There have been several studies on predicting cavitation in outboard motors, including one study comparing two-dimensional blade tests and CFD calculations⁽⁷⁾, and another on predicting erosion based on CFD calculation results for cavitation⁽⁸⁾. However, in terms of three-dimensional cavitation prediction accuracy, neither of the above studies address the practical applications in outboard motors at any great length.

Thus, in this paper, the authors will clarify the connections between computational cost and accuracy in order to predict performance, accounting for the cavitation that occurs in outboard motor propellers.

2. Cavitation Prediction Calculation

2.1. Calculation Subject

The purpose of this paper was to reproduce the pressure around objects in a liquid and predict the areas where cavitation will be generated.

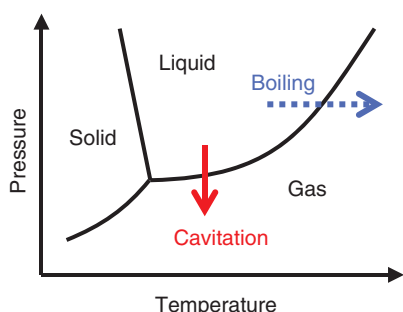


Fig. 1 Phase diagram of water

* Power Products R&D Center

** Institute of Fluid Science, Tohoku University

Given the intended application in outboard motors, as well as the multiple studies on external flow and in general, calculations were performed for a propeller shape in water.

2.2. Calculation Method

2.2.1. CFD solver

Commercial CFD software was used for calculations. The calculation settings are given in Table 1. Whereas actual cavitation is an unsteady phenomenon, it was treated as a steady flow using a Reynolds-averaged Navier-Stokes (RANS) equation in light of calculational costs. Assuming flow to be roughly equivalent in liquid and gas phases, an Euler mixture model with no speed difference between liquid and gas was used to model the gas phase. For the cavitation model, a Zwart-Gerber-Belamri bubble dynamics model⁽⁹⁾ was used.

2.2.2. Stabilization of the calculation procedure

When cavitation occurs, the density of a liquid will change suddenly, converting it to gas. Calculations are also destabilized by the sudden change in physical quantities; the calculation diverges at the point when the cavitation occurs, making the equation unsolvable in some cases.

As seen in Table 2, the calculation was stabilized by starting with the pressure environmental conditions set high, calculating for single-phase and multi-phase flow in separate steps, and then reducing the pressure gradually.

2.2.3. Calculational domain

The calculational domain is given in Fig. 2. The domain dimensions were aligned with conditions for the cavitation tunnel in the test to be described below, against which results were compared to verify computation accuracy for this study. The calculational domain used was a cylindrical shape of equivalent area to the flow path cross section of the tunnel test device. Propeller rotation was simulated by rotating the entire calculational domain with the outer periphery wall of the cylinder fixed to the absolute coordinate system. To reduce calculational costs, the calculational domain was reduced to a three-fold symmetrical domain centered on the propeller axis for just one propeller blade, using a periodic boundary surface.

2.2.4. Calculational mesh

As mentioned above, reproducing the flow near the

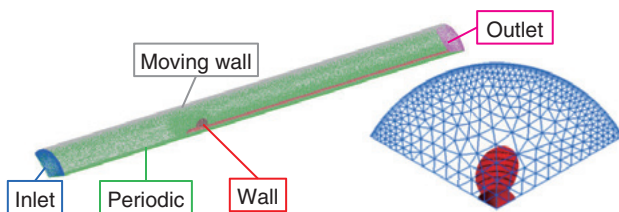


Fig. 2 Calculational domain

Table 1 Calculation settings

CFD software	ANSYS Fluent 16.0
Time	Steady
Viscous model	SST k- ω
Multiphase flow model	Mixture model
Cavitation model	Zwart-Gerber-Belamri

Table 2 Calculation procedures

Step 1	Single phase flow calculation with high pressure
Step 2	Multiphase flow calculation with high pressure
Step 3	Cavitation calculation with high pressure
Step 4	Cavitation calculation while reducing pressure

Table 3 Mesh generation parameter for boundary layer

	Mesh i	Mesh ii	Mesh iii
Number of boundary layers	25	15	5
First layer height [mm]	0.025	0.05	0.2
Growth rate	1.05		

object surface is important to simulating cavitation. As this study uses a low-Reynolds-number SST k- ω turbulence model, calculational accuracy should increase as the mesh resolution near the object surface is increased⁽¹⁰⁾. Conversely, increased mesh resolution will also increase the calculational cost. Thus, in order to verify the trade-off between calculation results and calculational cost due to mesh resolution, the mesh near the object surface was changed for the boundary layer only.

The mesh generation parameters for the calculated boundary layers are shown in Table 3. Note here that the mesh domain thickness was set to be about the same for each calculated boundary layer mesh.

3. Results and Discussion

3.1. Verification Tests for Calculational Accuracy

In order to verify their calculational accuracy, the CFD calculation results were compared to results of a model propeller test conducted in a cavitation tunnel. The cavitation tunnel test was conducted in the Marine Propeller Cavitation Tunnel at the University of Tokyo⁽¹¹⁾. A cavitation tunnel is a circulating water tank. Cavitation can be generated simulating actual usage by changing pressure and flow velocity in the tunnel. In the test, a propeller was rotated within a quadratic prismatic flow path, then a dynamometer was used to measure thrust T and torque Q while observing the cavitation patterns from the observation window.

The key propeller characteristics are given in Table 4. Three types of propeller were used in this paper. D in the table refers to the propeller diameter.

Here, the equations given below use the common, non-dimensional advance coefficient J , thrust coefficient K_T , torque coefficient K_Q , propeller efficiency η , and cavitation number σ_n as used in the field of ship propulsion.

$$J = \frac{V_A}{nD}$$

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

$$K_Q = \frac{Q}{\rho n^2 D^5}$$

$$\eta = \frac{J \cdot K_T}{2\pi \cdot K_Q}$$

$$\sigma_n = \frac{p_0 - p_v}{\frac{1}{2} \rho n^2 D^2}$$

V_A represents propeller inflow velocity, n the propeller speed, ρ the fluid density, p_0 the fluid standard pressure, and p_v the fluid saturated vapor pressure.

Advance coefficient J is a non-dimensional number for the angle of flow into the propeller. At the same propeller speed, J increases with boat speed. Cavitation number σ_n is a non-dimensional number for how easily propeller cavitation occurs. Cavitation is increasingly likely the smaller σ_n becomes.

In a cavitation tunnel test, advance coefficient J and cavitation number σ_n can be matched to actual usage conditions to observe similar cavitation.

In our test, an advance coefficient J of 0.99 and a cavitation number σ_n of 0.53 were set as the standard test conditions. These conditions were set to correspond to a flow speed of 70 km/h and propeller speed of 3000 rpm: the maximum speed range for a planing boat equipped with a 250-horsepower outboard motor under actual usage conditions. During the test, propeller speed was kept constant and the advance coefficient J was changed by controlling the flow velocity in the tunnel.

3.2. Calculation Mesh Verification Results

As mentioned in Clause 2.2.4., three calculation meshes with different boundary layers were used in CFD calculation on Propeller A from Table 4. Taking the calculation time for Mesh i as the baseline, the calculation times for one iteration under each calculation mesh condition are given in Table 5.

The cavitation patterns on the propeller blade suction side are given in Fig. 3, and a diagram of propeller performance is given in Fig. 4. The domain covered in cavitation from the propeller blade suction side grows as mesh resolution is increased. Given that thrust will increase the wider the domain covered by blade suction side

Table 4 Key propeller characteristics for cavitation tunnel test

Propeller name	A	B	C
Diameter [m]	0.220		
Pitch ratio (at 0.7D)	1.26	1.06	1.12
Expanded area ratio	0.481	0.481	0.460
Number of blades	3		
Rotational direction	Counter-clockwise		
Material	Corrosion resistant aluminum alloy		

Table 5 Relative calculation time

	Mesh i	Mesh ii	Mesh iii
Relative calculation time (Mesh i as 1.00)	1.00	0.83	0.58

cavitation, this is thought to account for the differences in thrust coefficient K_T in Fig. 4.

In the verification results, Mesh i and Mesh ii exhibited very similar cavitation patterns and propeller performance with a difference within 1%. Given this, Mesh ii was selected for subsequent calculations due to balance between calculational accuracy and cost.

3.3. Summary of Test Verification Results

The test verification results are separated into cavitation patterns for qualitative comparison and propeller performance for quantitative comparison. Note here that calculations took approximately 9 hours using a dual-CPU, 48-core parallel processor for each calculation condition.

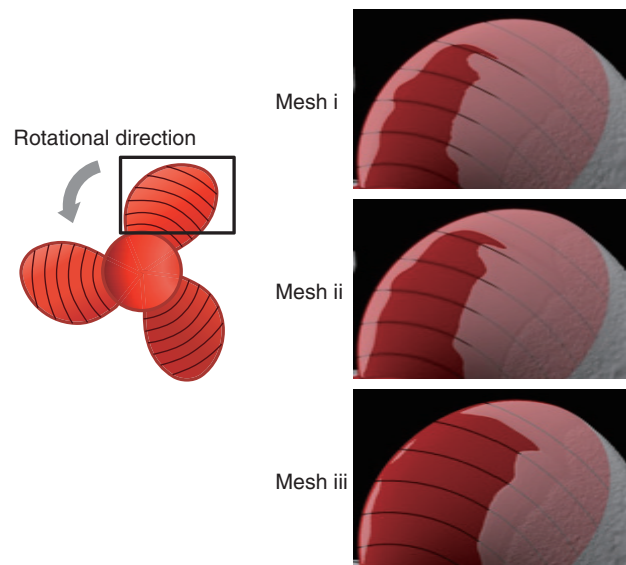


Fig. 3 Influence of boundary layer mesh resolution on cavitation on blade suction side

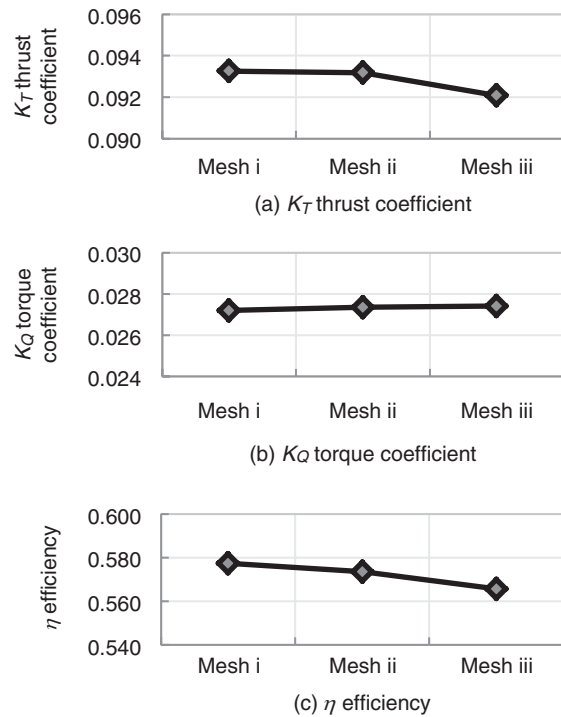


Fig. 4 Influence of boundary layer mesh on propeller performance

3.3.1. Cavitation patterns

The resulting cavitation patterns from the model propeller test and CFD calculations on the blade suction side of Propeller A are compared in Fig. 5. From left to right, the results are for an advance coefficient J of 0.80, 0.99, and 1.10, respectively.

- At J values of 0.80 and 0.99, the entire blade leading edge generates sheet cavitation; at a J value of 1.10, cavitation is only generated near the blade tip from positions of D values of 0.8 or higher.
- Cavitation generated by the preceding blade gets elongated when J is smaller.

These patterns were very similar.

While calculations did not replicate the bubble cavitation observed in the blade center at a J value of 1.10, bubble cavitation is highly unsteady and thus likely challenging to reproduce with the steady-state calculations done in this study. Furthermore, a J value of 1.10 exceeds peak propeller efficiency and thus is outside of actual usage ranges. Given that calculations successfully reproduced the sheet cavitation patterns generated at J values of 0.80 and 0.99 within the actual usage range, the test was deemed to be able to reproduce steady cavitation patterns.

3.3.2. Propeller performance

A comparison of propeller performance from the model

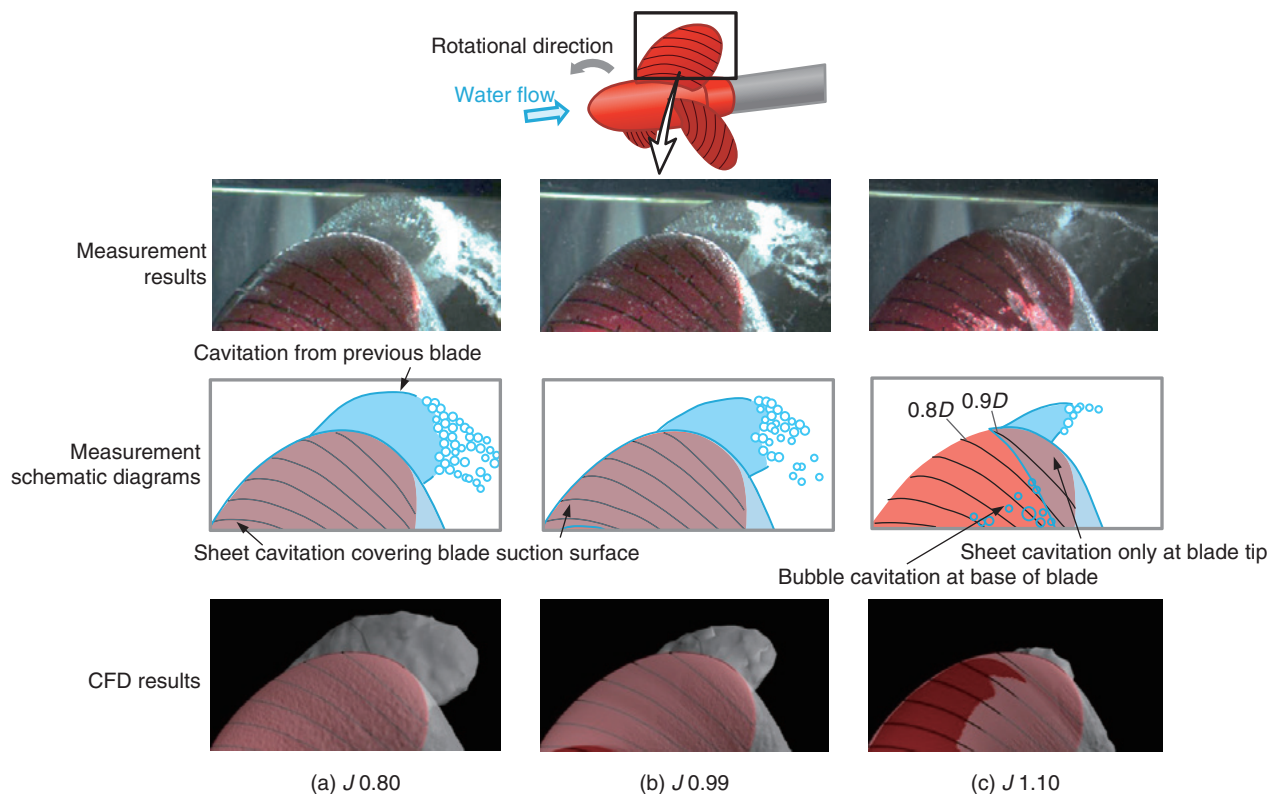


Fig. 5 Propeller blade cavitations, compared between test and calculation

propeller test and CFD calculations is given in Fig. 6. The high-to-low sequence of thrust coefficient K_T between the propellers and peak positions for propeller efficiency η

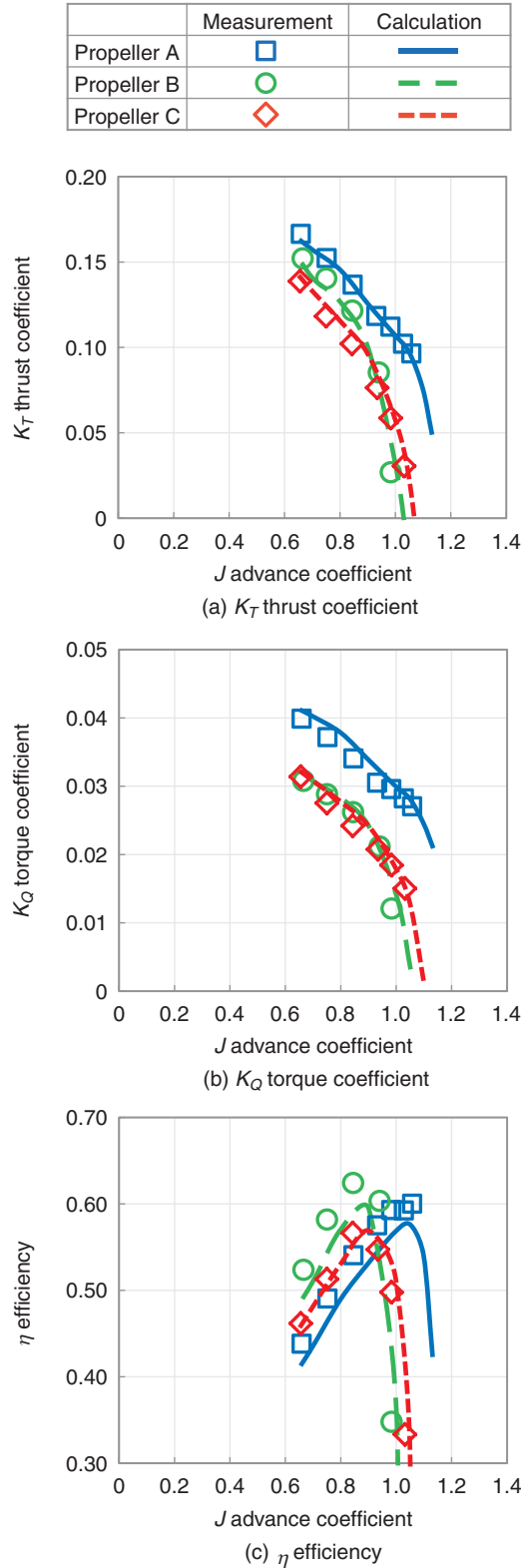


Fig. 6 Propeller performance curve compared with model test results

were consistent between the model propeller test results and CFD calculation. Thus, the CFD calculations are capable of reproducing the differences in performance from different propeller specifications.

Furthermore, within the actual usage range up to the efficiency peak for advance coefficient J , the model test and CFD calculation results were quantitatively consistent to within an error of $\pm 7\%$ for the performance values (thrust coefficient K_T , torque coefficient K_Q , and propeller efficiency η).

Thus, the results of the study illustrate that propeller performance in cavitation-generating conditions for actual outboard motor propeller usage can be quantitatively reproduced using CFD calculations.

4. Conclusion

In order to balance computational costs and accuracy in predicting the phenomenon of cavitation for the shape of outboard motor propellers, the following results were obtained:

- (1) The computational costs of changing the boundary layer mesh near the object surface was clarified, along with the impact of this change on the main propeller performance values of thrust coefficient K_T , torque coefficient K_Q , and propeller efficiency η .
- (2) The results of model tests and CFD calculation were compared, confirming that cavitation patterns were qualitatively consistent and that quantitatively, main propeller performance value accuracy is consistent to within $\pm 7\%$.

From the above results, cavitation effects can be accounted for from the early stages of outboard motor development as output and speed increases.

References

- (1) Ukon, Y., Fujisawa, J., Kudo T., Kurobe Y.: Application of a Theoretical Design Method to Full Scale Supercavitating Propellers for High-Speed Boats, Transactions of the West-Japan Society of Naval Architects, Vol. 100, p. 133-143, (2000) (in Japanese)
- (2) Suzuki, K., Sasaki, N., Kawamura, T.: Sentaiteiko to Suishin (Sempaku Kaiyo Kogaku Sirizu 2), SEIZANDO-SHOTEN PUBLISHING CO., LTD., p. 143-162, (2012) (in Japanese)
- (3) Kurioka, S.: Fuel Consumption and Performance Prediction in Outboard Motors for High-speed Planing Boats using CFD Simulation, Honda R&D Technical Review, Vol. 25, No. 1, p. 176-183
- (4) Fujiyama, K.: Numerical Simulation of Ship Hull Pressure Fluctuation Induced by Cavitation on Propeller with Capturing the Tip Vortex, International Symposium on Marine Propulsors smp'15, June, (2015)
- (5) Tomimatsu, S., Kado, K., Otani, R.: Cavitation CFD in Pump Development, Turbomachinery, Vol. 42, No. 11,

- p. 707-712, (2014) (in Japanese)
- (6) Nohmi, M.: Remaining Problems on Cavitation CFD, Turbomachinery, Vol. 42, No. 11, p. 694-700, (2014) (in Japanese)
 - (7) Kamiya, A.: Prediction of the Cavitation Effect on the Flow Around the Outboard Motor Propeller Blade Hydrofoil Section Using CFD, SAE Technical Paper, 2013-32-9157, (2013)
 - (8) Watanabe, T., Sakamoto, H.: Method for Predicting Erosion Due to Cavitation of Outboard-Motor, SAE Technical Paper, 2014-32-0054, (2014)
 - (9) Zwart, P. J., Gerber, A. G., Belamri T.: A Two-Phase Flow Model for Predicting Cavitation Dynamics, ICMF 2004 International Conference on Multiphase Flow, Japan, Paper No. 152, May 30-June 3, (2004)
 - (10) ANSYS, Inc. Fluent 16.0 Theory Guide, Chapter 4: Turbulence, (2014)
 - (11) Kawamura, T., Maeda, M.: Marine Propeller Cavitation Tunnel at The University of Tokyo, Journal of the Japan Institution of Marine Engineering, Vol. 43, No. 5, p. 720-725, (2008) (in Japanese)

■ Author ■



Ayaka NANGO



Yuka IGA