Investigate the Influence of Phase Angle on the Performance of Propeller Boss Cap Fins Using CFD Method

Nguyen Thi Ngoc Hoa
Ho Chi Minh City University of Transport, Vietnam
Email: hoa_vt@hcmutrans.edu.vn

Mai Van Quan
Military Institute of Ship Design, Vietnam
Email: maiquan309@gmail.com

Vu Ngoc Bich
Binh Duong University, Vietnam
Email: vnbich@bdu.edu.vn

***Abstract*—This paper investigates the influence of phase angle of the Propeller Boss Cap Fins (PBCF) on the hydrodynamics performance of the propulsion system including the propeller, shaft, and PBCF by Computational Fluid Dynamic (CFD) method. The rotating reference frame approach was used to simulate propeller rotation. First, the well-known Potsdam Propeller in model scale was used as an initial propeller to verify and validate the propeller performance results obtained by CFD method with experimental data and then investigate the influence of phase angle on the performance of propeller. The best phase angle for PBCF is shown and the recommendations for choosing phase angle with respect to increase propeller performance are given. The commercial solver Star CCM+ is used to solve the flow around the PBCF and the propeller.**

***Index Terms*—PBCF, phase angle, propeller, CFD**

I. INTRODUCTION

Nowadays, there are increasing requirements for newly built ships in terms of energy saving and CO_2 emission reduction. In 2010, the International Maritime Organization (IMO) launched the Energy Efficiency Index (EEDI) to measure the amount of CO_2 a ship puts out in relation to the cargo being transported. Therefore, EEDI mitigation solutions must be taken care by the designers. One of the common approaches to reduce EEDI is using high performance propulsion system. The Boss Cap Fins Propeller (PBCF) is one of many energy-saving devices that have been widely used in the world. This is a device that replaces the traditional propeller cap, in order to improve the operating efficiency of the propeller and reduce the hub vortex behind the propeller. This device was used for more than 3000 ships on the world [1] (see Fig. 1).

The development of computational resources has enabled designers and researchers to solve many complex problems in ship hydrodynamics using Computational Fluid Dynamics (CFD). One of the most popular CFD approach in ship hydrodynamics is the Reynold Averaged Navier Stokes Equation (RANSE). It can give highly accurate results with reasonable computation time. This saves cost and time compared to a towing tank tested. Furthermore, the CFD method can perform calculations on the full scale, then directly output hydrodynamic results such as ship resistance, propeller thrust, etc. In the traditional method, the full-scale result is deduced from the experimental results with model scale.

Figure 1. Picture of PBCF [1]

Regarding the propeller calculation method and propeller design, many approaches are applied: such as lifting line, lifting surface, boundary element method (BEM) and CFD (including RANSE, LES, DNS) [7], [8], [9], [10]. However, as stated above, for flow analysis

Manuscript received January 5, 2022; revised May 31, 2022.
Corresponding author: Vu Ngoc Bich.

doi: 10.18178/ijmerr.11.7.513-519

around propellers, the RANSE method is the most popular and widely used [12], [13], [17].

Many authors have performed CFD calculation for PBCF and proved the effect of PBCF on the improvement of propeller performance. Lim et al. [14] conducted the parameters of hub with PBCF. The propeller test results show that the hub radius has a significant impact on the propeller performance. Nojiri et al. [15] carried out model tests and numerical analyses to improve the design of PBCF for achieving a high propulsive efficiency. Kawamura et al. [16] used CFD to investigate the scale effects of the performance of two sets of PBCF. However, there are few publications, which takes into account the influence of phase angle of the PBCF to the propeller performance. The phase angle is one of the most important parameters for designing PBCF [3]. Therefore, this paper investigates the influence of phase angle on the performance of propeller boss cap fins to find the best position, which gives highest efficiency. Besides the paper will highlight the characteristics of the PBCF and make recommendations for designers and researchers to design and optimize the PBCF.

II. Material and Method

A. Governing Equations

Mass conservation (continuity equation): mass is neither created nor destroyed:

$$\frac{d}{dt}\iiint_D \rho dv = 0 \tag{1}$$

Momentum conservation (Newton second's law): rate of change of momentum equal to force:

$$\frac{d}{dt}\iiint_D \rho\vec{U}dv = \iiint_D \rho\vec{f_v}dv + \iint_D \vec{T}dS \tag{2}$$

where: dv is arbitrary control volume; ρ is fluid density; D is fluid domain; $\vec{f}_v$ is a volume force (normally gravity force); $\vec{T}$ is constraints. $\vec{T} = \overline{\overline{\sigma}}.\vec{n}$, where: $\overline{\overline{\sigma}}$ - constraint tensor.

B. Turbulence Model

The turbulence model is applied in this study was realizable k–ε two layer. This model solves transport equations for turbulence kinetic energy (k) and the turbulence dissipation rate (ε) in order to calculate the eddy viscosity by following equation:

$$\mu_t = \rho C_\mu f_\mu kT \tag{3}$$

where ρ is the density of fluid, T is a turbulent time scale, f_μ is a damping function, C_μ is a model coefficient.

The turbulent time scales for realizable k–ε two layer is defined by Eqn. (4):

$$T = \max\left(T_e, C_t\sqrt{\frac{\upsilon}{\varepsilon}}\right) \tag{4}$$

where $T_e = k/\varepsilon$ is the large-eddy time scale, C_t is model coefficient, υ is the kinematic viscosity.

The transport equations for the turbulent kinetic energy k and the turbulent dissipation rate ε are determined as follow:

$$\frac{\partial}{\partial t}(\rho k) + \nabla\cdot(\rho k\overline{\mathrm{v}}) = \nabla\cdot\left[\left(\mu + \frac{\mu_t}{\sigma_k}\right)\nabla k\right] + P_k - \rho(\varepsilon - \varepsilon_0) + S_k \tag{5}$$

$$\begin{gathered}\frac{\partial}{\partial t}(\rho\varepsilon) + \nabla\cdot(\rho\varepsilon\overline{\mathrm{v}}) = \nabla\cdot\left[\left(\mu + \frac{\mu_t}{\sigma_\varepsilon}\right)\nabla\varepsilon\right] + \frac{1}{T_e}C_{\varepsilon 1}P_\varepsilon - \\ C_{\varepsilon 2} f_{2\rho}\left(\frac{\varepsilon}{T_e} - \frac{\varepsilon_0}{T_0}\right) + S_\varepsilon\end{gathered} \tag{6}$$

Where $\overline{\mathrm{v}}$ is the mean velocity, $\sigma_k, \sigma_\varepsilon, \sigma_{\varepsilon_1}$ and $C_{\varepsilon 2}$ are model coefficients, P_k and P_ε are production terms; f_2 is a damping function, S_k and S_ε are user-specified source terms, ε_0 is the ambient turbulence value in the source terms and T_0 is specific time-scale.

III. Numerical Simulation

A. Turbulence Model

As stated above, PBCF is a propeller cap with fins. It has some geometrical parameters similar to the propeller cap such as diameter (upward and downward), length. Besides, there are additional parameters that are unique for PBCF. They are fin height, fin length, Pitch angle, and Phase lag (Fig. 2). The influence of those parameters to the open water efficiency has been studied both in numerical and experimental method by Jeonghwa Seo et al. [3]. However, the impact of relative position of the PBCF to the propeller has not been studied. This paper will focus to study this parameter.

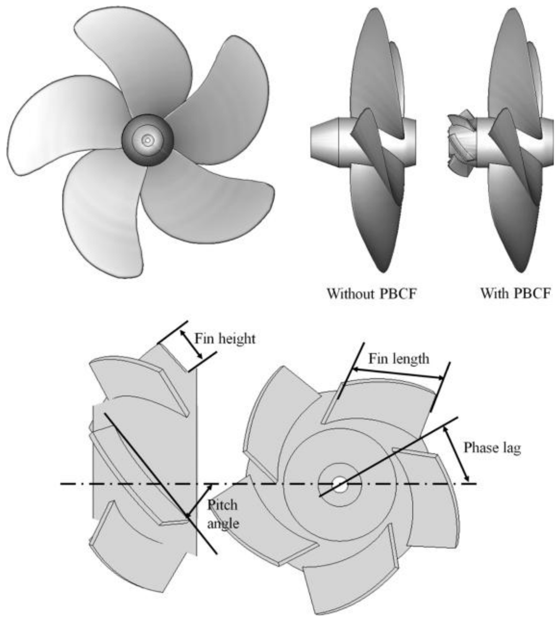


Figure 2. Geometrical features of PBCF [1]

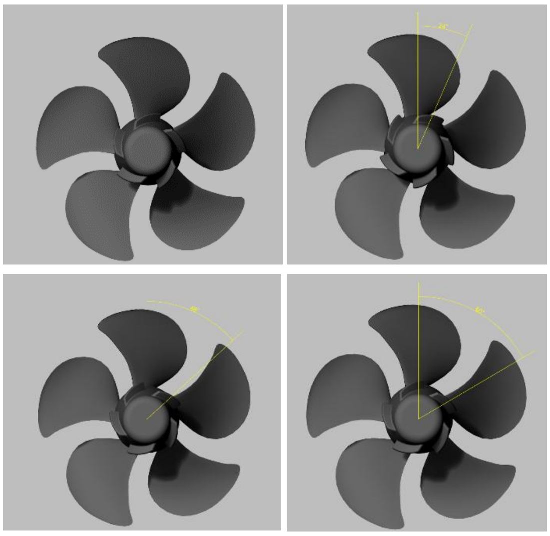


Figure 3. Relative position of PBCF to the propeller with phase angle 0, 24, 48 and 60 degrees

The relative position of the PBCF to the propeller can be called the phase angle. This is the relative angle between PBCF and the propeller when the PBCF is fitted with the propeller. This angle is 0 when the center of the fins is at the center line of the propeller blades. This angle can be varied from 0 degree to the 360 degrees divide to the number of blades. (Fig. 3).

B. Selection of Propeller Model

Potsdam Propeller is selected to perform CFD calculation validation. SVA Potsdam [2] provides the experimental results. Therefore, simulation result can compare to the experimental one, to validate the CFD setup and study the mesh convergence. Potsdam propeller parameters are shown in Table I and its shape is presented in Fig. 4.

TABLE I. POTSDAM PROPELLER PARAMETERS [2]

Propeller Parameters		Units	Values
Propeller Diameter	D	m	0.25
Expanded Area	A_E/A_0	-	0.778
Hub ratio	D_h/D	-	0.3
Blade number	Z	-	5
Pitch ratio	$P_{0.7}/D$		1.635
Direction of Rotation	-	-	Right

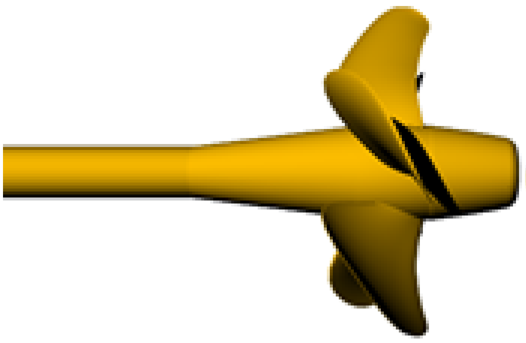


Figure 4. Potsdam propeller

C. Simulation Setup

Commercial solver Star CCM+ is chosen to simulate the propeller in open water. The fluid domain is created by a cylinder with a diameter of 2.5 meters (10 times the diameter of the propeller), 3.5 meters long. The outlet is 3 meters far away from the propeller (Fig. 5). To capture vortex and streamline behind the propeller, the wake area of the propeller is refined as in Fig. 6.

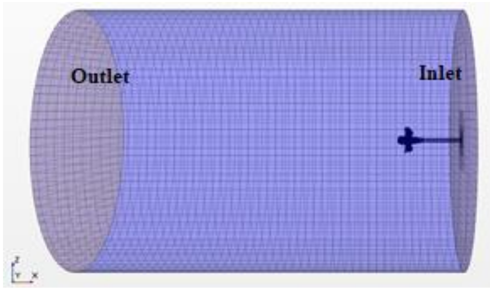


Figure 5. Fluid domain

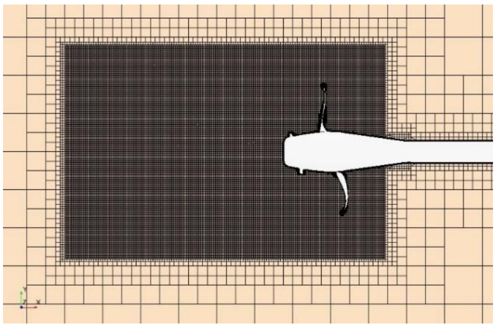


Figure 6. Local refinement around propeller

The Rotating Reference Frame method is applied to simulate the propeller rotation due to the fact that this approach is a steady one, leading to less computational time, but the level of accuracy is still maintained.

The velocity inlet boundary condition is set to the front boundary. The velocity at inlet is equal to the flow velocity coming to the propeller. The outlet boundary is set to pressure outlet boundary condition. All the surfaces of the propeller are used “wall function” approach. The turbulence model is k-omega SST with “all wall treatment” for Y+.

IV. NUMERICAL SIMULATION

A. Mesh Convergence Study

To make sure the result is mesh-independent; the first step of CFD simulation is performing mesh convergence study. Mesh convergence is studied with 5 different mesh size, ranging from 1.3 million cells to 11.88 million cells (Table II). The simulation is performed with advance coefficient J = 1.4. The calculation result is presented in Table II below. The numerical error is defined as:

$$\varepsilon_{i,i+1}(\%) = \frac{S_i - S_{i+1}}{S_i} \qquad (7)$$

Where S_i is the solution with mesh *i*

TABLE II. NUMERICAL RESULT OF MESH CONVERGENCE STUDY

Mesh	Mesh size [million cells]	K_T	ε(i,i+1)	$10K_Q$	ε(i,i+1)
Mesh 1	1.33	0.18590	-	0.552	-
Mesh 2	2.15	0.18611	0.11%	0.5534	0.25%
Mesh 3	4.28	0.18675	0.34%	0.5549	0.27%
Mesh 4	6.63	0.18701	0.14%	0.5557	0.14%
Mesh 5	11.88	0.18707	0.03%	0.5559	0.04%

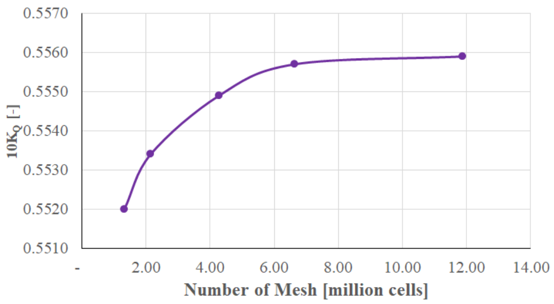


Figure 7. Mesh convergence study for K_Q

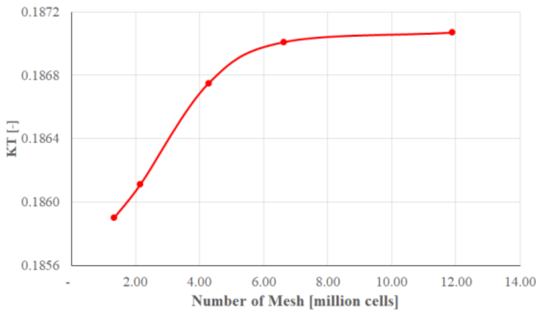


Figure 8. Mesh convergence study for K_T

Fig. 7 and Fig. 8 show that there is a significant convergence of mesh size. Besides, the deviation of result for different mesh are less than 1%. So, the largest mesh size (mesh 5 with 11.8 million cells) is selected for further computation to reduce discretization error as low as possible.

B. Validation of CFD Setup

As mentioned above, the mesh 5 is used to perform further computations with 5 different J and then compare to experimental results. The comparison is shown in Table III and the open water curve is presented in Fig. 9.

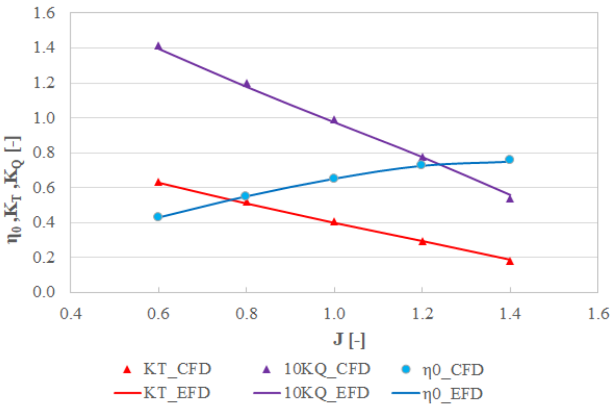


Figure 9. Open water curve (CFD and experimental result)

Both Table III and Fig. 9 show that the CFD results are very close the experimental one, with the deviations are just under 2%. It proves that this CFD setup is valid and it can be used for further computation with PBCF. The propeller attached with PBCF will be calculated in the next section by same CFD setup, just replacing the normal cap with PBCF in the simulation.

TABLE III. DETAILS OF CFD RESULT IN COMPARISON WITH EXPERIMENTAL ONE

j	K_T			$10K_Q$			η_0		
	CFD	**EFD**	**Diff**	**CFD**	**EFD**	**Diff**	**CFD**	**EFD**	**Diff**
0.6	0.632	0.629	0.4%	1.414	1.396	1.2%	0.428	0.430	-0.8%
0.8	0.517	0.510	1.6%	1.203	1.178	2.0%	0.548	0.551	-0.4%
1.0	0.406	0.399	1.5%	0.992	0.975	1.7%	0.651	0.652	-0.3%
1.2	0.295	0.295	-0.3%	0.774	0.776	-0.2%	0.726	0.726	-0.2%
1.4	0.185	0.188	-2.4%	0.542	0.559	-3.3%	0.757	0.749	0.9%

EFD = Experimental Fluid Dynamic

C. CFD Calculation of Propeller with PBCF

To access the influence of PBCF to the propeller performance, two major simulations are performed. The first simulation is the propeller with normal cap (without fins) and the second simulation is with PBCF. For the simulation with PBCF, the multiple phase angles are simulated to investigate the effect of these angles. Details of computational cases are shown in Table IV.

The Potsdam propeller has 5 blades and the PBCF has also 5 fins. Therefore, phase angle simulations are performed from 0 degree to 60 degree with step of 12 degree. The case with 72 degree is the same as 0 degree.

The setup of simulations is similar to the setup, which is validated above. The cap is replaced by the PBCF. Finest mesh with 11.88 million cells is used to calculate.

TABLE IV. DETAILS OF COMPUTATIONAL CASES

No.	Computational cases at J = 0.6
1.	Propeller with Cap only
2.	Propeller with PBCF at phase angle 0 degree
3.	Propeller with PBCF at phase angle 12 degree
4.	Propeller with PBCF at phase angle 24 degree
5.	Propeller with PBCF at phase angle 36 degree
6.	Propeller with PBCF at phase angle 48 degree
7.	Propeller with PBCF at phase angle 60 degree

V. RESULT AND DISCUSSION

Results of all computational cases are presented in Table V. The results of simulation with PBCF are compared with the case without PBCF.

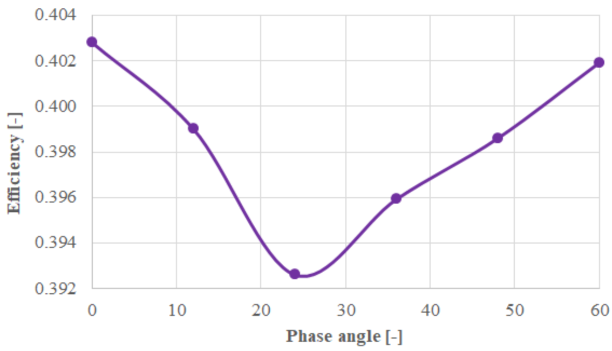
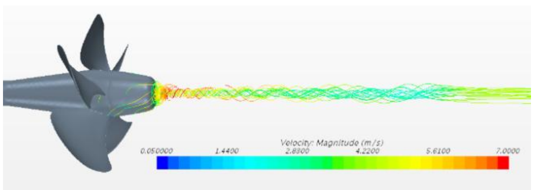


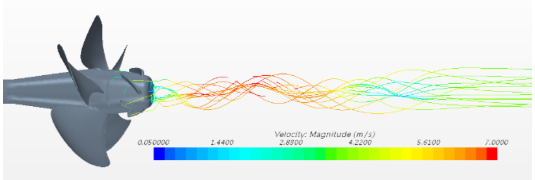
Figure 10. Influence of phase angle to open water efficiency

Overall, the replacement of traditional Cap by PBCF increase the open water efficiency up to 2%. At phase angle 0 degree, the open water efficiency reaches highest value. Fig. 10 shows the effect of phase angle to the open water efficiency. The open water efficiency decreases from phase angle of 0 degree to 36 degree and then increase again from 36 degree to 60 degree. Therefore, in this case, the best position for PBCF should be at phase angle 0 degree. The flow around propeller is visualized by post processing the result in the case with and without PBCF to see the differences.

By post-processing the result, the CFD method allows us to look into the details of the flow. Fig. 11 and Fig. 12 present the streamlines and vortex distribution in two cases: propeller with traditional cap and propeller with PBCF at phase angle 0 degree. The hub vortex decreases significantly for the case with PBCF, and the flow is more uniform. This leads to the increase of thrust coefficient and open water efficiency. Besides, the reduction of hub vortex can decrease the vibration and rudder erosion behind the ship.

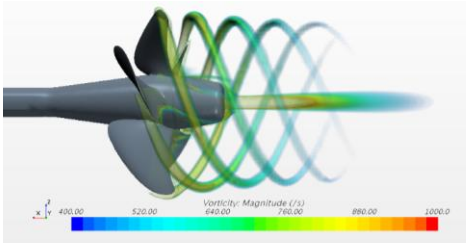


a) Stream lines for propeller without PBCF at J=0.6

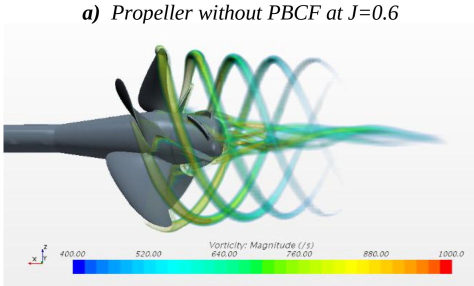


b) Stream lines for propeller with PBCF at J=0.6

Figure 11. Stream lines for propeller at J = 0.6 and phase angle 0 degree

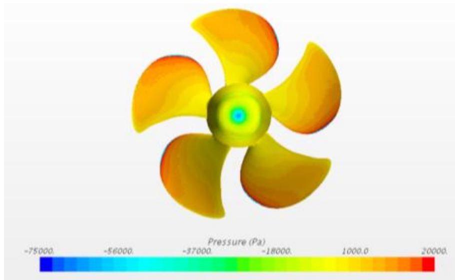


a) Propeller without PBCF at J=0.6

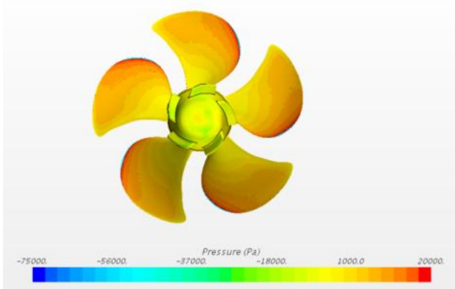


b) Propeller with PBCF at J=0.6

Figure 12. Vortex distribution at J = 0.6 and phase angle 0 degree



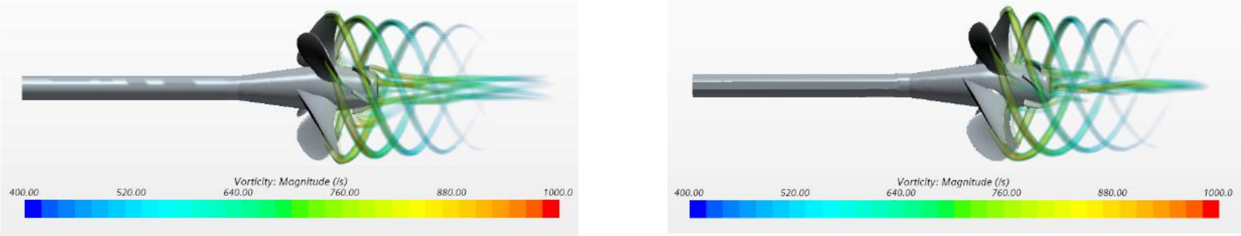
a) Propeller without PBCF at J=0.6



b) Propeller with PBCF at J=0.6 and phase angle 0 degree

Figure 13. Pressure distribution on propeller at J = 0.6 and phase angle 0 degree

Fig. 13 shows the pressure distribution on blades, there is a pressure drop at the center of the propeller cap. Meanwhile, with PBCF, the pressure distribution in this area is more uniform. This phenomenon is the reason why the hub vortex is significantly reduced for the case with PBCF.



a) *Propeller with PBCF, 0-degree phase angle*

b) *Propeller with PBCF, 24-degree phase angle*

Figure 14. Vortex distribution for the case with phase angle 0 and 24 degree

TABLE V. COMPUTATIONAL RESULT WITH DIFFERENT PHASE ANGLES

Case study	K_T	*Diff (%)*	$10K_Q$	*Diff (%)*	η_0	*Diff (%)*
Cap only	0.5886	0%	0.1429	0%	0.3931	0%
Phase angle 0	0.5933	0.80%	0.1406	-1.61%	0.4028	2.47%
Phase angle 12	0.5903	0.30%	0.1412	-1.19%	0.3990	1.50%
Phase angle 24	0.5960	1.26%	0.1403	-1.82%	0.3926	-0.13%
Phase angle 36	0.6027	2.40%	0.1453	1.68%	0.3959	0.72%
Phase angle 48	0.5969	1.41%	0.1418	-0.77%	0.4017	2.18%
Phase angle 60	0.5946	1.02%	0.1412	-1.18%	0.4019	2.24%

Fig. 14 shows the vortex distribution of the propeller with PBCF in case of phase angle 0 degree and 24 degrees. It can be seen Fig. 14 the considerable decrease of hub vortex of propeller with PBCF 24-degree phase angle in comparison with phase angle 0 degree. It is the reason for the higher open water efficiency of 24-degree-phase angle case than the 0-degree-phase angle case.

V. CONCLUSION

In this paper, unsteady RANSE method has been applied to investigate the effect phase angle on the performance of propeller boss cap fins. To assess this effect, six case studies with variation of phase angle are carried out. The following conclusions can be made:

- Generally, the replacement of traditional Cap by PBCF increase the open water efficiency. In case of analyzed propeller, the performance of propeller increases to 2%.
- The propeller performance changes with variation of phase angle. In case of propeller, the open water efficiency reaches highest value increase at phase angle 0 degree.
- Analyzing the change in flow field around the propeller with and without PBCF, and with variation of phase angle provides a fully explaining the physical phenomenon of changing performance of propeller.
- By considering the phase angle mentioned above, it might be worthwhile to extend this research by adding more design variables, such as diameter of boss cap, the shape of fin, the pitch angle.

CONFLICT OF INTEREST

The authors declare no conflict of interest.

AUTHOR CONTRIBUTIONS

Nguyen Thi Ngoc Hoa: Conceptualization, Methodology, Investigation, Writing-original draft, Writing-review & editing. Vu Ngoc Bich: Conceptualization, Methodology, Investigation, Visualization, Software, Resources, Supervision, Writing - revised draft, Writing - review & editing. Mai Van Quan: Conceptualization, Methodology, Investigation, Visualization, Software, Resources, Supervision, Writing – revised draft.

ACKNOWLEDGMENT

The author is grateful to Ho Chi Minh City University of Transport, the Binh Duong University, Military Institute of Ship Design and Hong Ha Shipbuilding One Member Limited Liability Company Vietnam for providing all research facilities during executing this work.

REFERENCES

[1] N. Takeo, I. Norio, and K. Hisashi, “Energy saving technology of PBCF (Propeller Boss Cap Fins) and its evolution,” *Journal of The Japan Institute of Marine Engineering*, vol. 46, pp. 350-358, 2011.

[2] SVA Potsdam Model Basin, Potsdam Propeller Test Case, Potsdam; 2011.

[3] S. Jeonghwa, L. Seung, H. Bunwoo, and R. Shin, “Influence of design parameter variations for propeller-boss-cap-fins on hub vortex reduction,” *Journal of Ship Research*, vol. 60, pp. 203-218, 2016.

[4] G. Kuiper, “Paint tests on three propellers for the leading edge project,” *Leading Edge Deliverable D4, CFD Validation for a Marine MARIN*, 2005.

[5] S. H. Rhee and S. Joshi, “Computational validation for flow around a marine propeller using unstructured mesh based navier-stokes solver,” in *Proc. 4th Joint Fluids Summer Engineering Conference*, ASME/JSME 2003

[6] D. Q. Li, “Validation of RANS predictions of open water performance of a highly skewed propeller with experiments,” *Journal of Hydrodynamics*, pp. 520–528, July 2006.

[7] M. Visonneau et al, “Computation of free-surface viscous flows around self-propelled ships with the help of sliding grids,” in *COMPIT 2012, 11th International Conference on Computer Applications and Information Technology in the Maritime Industries. 2012.*

[8] A. F. Molland and S. R. Turnock, *Marine Rudder and Control Surface*, 2006.

[9] S. Caja et al, “Simulation of the incompressible viscous flow around ducted propellers with rudders using a RANSE solver,” in *Proc. the 27th Symposium on Naval Hydrodynamics*, 2008, *US Office of Naval Research (ONR).*
[10] V. Krasilnikov et al, “A numerical study on the characteristics of the system propeller and rudder at low speed operation,” in *Proc. Second International Symposium on Marine Propulsors*, 2011.
[11] X. Wang and K. Walters, “Computational analysis of marine-propeller performance using transition-sensitive turbulence modeling,” *Journal of Fluids Engineering*, vol. 134, July 20, 2012.
[12] T. N. Tu and N. M. Chien, “Comparison of different approaches for calculation of propeller open water characteristic using RANSE method,” *Naval Eng. J.*, vol. 130, no. 1, pp. 105-111, vol. 7, 1 March 2018.
[13] T. N. Tu, “Numerical simulation of propeller open water characteristics using RANSE method,” *Alexandria Engineering Journal*, vol. 58, no. 2, pp. 531-537, 2019.
[14] S. S. Lim, T. W. Kim, D. M. Lee, C. G. Kang, and S. Y. Kim, “Parametric study of propeller boss cap fins for container ships,” *International Journal of Naval Architecture and Ocean Engineering*, vol. 6, no. 2, pp. 187–205, 2014.
[15] T. Nojiri, N. Ishii, and H. Kai, “Energy-saving technology of PBCF (propeller boss cap fins) and its evolution,” *Marine Engineering*, vol. 46, no. 3, pp. 63–71, 2011.
[16] T. Kawamura, K. Ouchi, and T. Nojiri, “Model and full-scale CFD analysis of propeller boss cap fins (PBCF),” *Journal of Marine Science and Technology*, vol. 17, no. 4, pp. 469–480, 2012.
[17] V. N. Bich and N. T. N. Hoa, "Numerical evaluation of roughness influences on open water propeller characteristics using RANSE method," *International Journal of Mechanical Engineering and Robotics Research*, vol. 11, no. 1, pp. 15-21, January 2022.

Nguyen Thi Ngoc Hoa received her Bachelor degree (2002) and M.Sc. degree (2013) from Ho Chi Minh city University of Transport.
She is a Lecturer, Institute of Mechanical Engineering, Ho Chi Minh city University of Transport. Her current interests include ship design, stability and optimization.

Vu Ngoc Bich was born in 1961. Graduated with a PhD (2007) in Russia. Associate professor in 2013.
He is currently a Senior Lecturer, Institute of Industrial Management and Logistics, Binh Duong University. His current interests include the design and optimization of ships and transportation equipment.

Mai Van Quan received his Engineer degree (2011) from Saint Petersburg State Maritime Technical University, received his Doctor of Philosophy degree (2014) from Nizhny Novgorod State Technical University n.a. R.E. Alekseev. He is a researcher, Military Institute of Ship Design. His current interests include ship design, construction and optimization