11th International Conference on Marine Technology
MARTEC 2018

PREDICTION OF PROPELLER PERFORMANCE USING COMPUTATIONAL FLUID DYNAMICS (CFD) APPROACH

N. AmiraAdam[1], A. Fitriadhy[1*], CJ Quah[2], Tri Haryanto,[3] Jaswar Koto[4]

[1]School of Ocean Engineering,
Universiti Malaysia Terengganu,
21030 Kuala Terengganu, Terengganu.

[2]CEO Numit Enterprise,
Seri Kembangan, Selangor, Malaysia.

[3]PT. Tarafulk Megantara Design, Jalan Gayunsari Barat VII,
No 20, Gayungan, Kota Surabaya, Jawa Timur 60233 Indonesia

[4]Department of Aeronautical, Automotive and Ocean Engineering,
Universiti Teknologi Malaysia, Johor Bahru, Malaysia

ABSTRACT

A reliable prediction approach to obtain a sufficient thrust and torque to propel ship at desired forward speed is obviously required. To achieve this objective, the authors propose to predict the thrust coefficient (K_T), torque coefficient (K_Q) and efficiency (η) of the propeller in open-water model test condition using Computational Fluid Dynamics (CFD) simulation approach. The computational simulation presented in various number of propeller blades(Z) within the range of advance ratio J=0.1 up to 1.05. The higher value of J lead to decrease K_T and K_Q but the η increased steadily at lower value of J and decreased at higher value of J. The results also showed that the three number of propeller blades, Z=3 give the best efficiency. The computation result is very useful as preliminary data for propeller performance characteristics.

Keywords: *Propeller, Thrust coefficient, Efficiency, CFD, Number of Blades*

*Corresponding author: naoe.afit@gmail.com

1.0 INTRODUCTION

The hydrodynamics analysis of propeller performance is an important element of the ship propulsion system to ensure the propeller will generate maximum thrust and minimum torque for the optimum propeller rotational speed and propel the ship at desired forward speed [1]. A great propeller design and specification have directly influence on fuel efficiency of the ship propulsion system and to reduce the rate of propeller cavitation [2] and [3]. As a common propeller design, a hydrodynamics characteristic of the propeller is a very prominent aspect to be analyzed in the early design stage using several methods in the form of open-water propeller model test. Several researcher, has been investigate the hydrodynamics characteristics of propeller using numerical and experimental approaches. According to [4], [5], [4] and [6] mathematical methods can used to predict the hydrodynamics characteristics of the propeller based on circulation or lifting line theory. Meanwhile, hydrodynamics characteristics of the propeller also predicted by using an experimental model test with towing tank [7], [8] and [9]. This experimental method is very expensive, time-consuming, and have a complex procedure for various hydrodynamics analysis test configuration. Following the works of [10], [11], [12], [13], [14] and [15] the numerical methods is adopted to solve and analyze the fluid problem. The computational fluid dynamics (CFD) simulation are the best alternative with several advantages such as allow to simulate using actual and model geometry scale in extreme condition of the fluid flow and the CFD simulation also have a good agreement with experimental data [16], [17], [18] and [19].

This paper presents a CFD simulation approach for the extension work from [20] and [21] to accessing the performance of propeller by K_T, K_Q, and Ƞ of the propeller. Here, a commercial CFD software, namely NUMECA Fine^{TM}/Turbo v12.2 is utilize by grid generation, flow solver and post-processing capabilities. Fine^{TM}/Turbo is specialized to simulate internal, rotating and turbomachinery flows for all types of fluids. The package has a fully hexahedral and highly automated grid generation module AutoGrid^{TM}. The package uses a 3D Reynolds Averaged Euler and Navier Stokes flow solver EURANUS. CFView^{TM} is a post-processing module which is also part of the package. Basically, this is solved by means of a grid independent study to estimate the optimal domain discretization. In this computation simulation, several numbers of blades are considered. The result is then comprehensively discussed to analyze their effect on K_T, K_Q, and η with the purpose of quantifying propeller performance quality.

2.0 THEORETICAL BACKGROUND

2.1 The Turbulence Model

In this turbulence model, the author proposes the Spalart-Allmaras model, which is an alternative formulation of a one-equation model that determines the turbulent viscosity directly from a single transport equation. The one-equation model is given by the following equation:

$$\frac{\partial \hat{\nu}}{\partial t}+u_j\frac{\partial \hat{\nu}}{\partial x_j}=c_{b1}(1-f_{t2})\hat{S}\hat{\nu}-\left[c_{w1}f_w-\frac{c_{b1}}{\kappa^2}f_{t2}\right]\left(\frac{\hat{\nu}}{d}\right)^2+\frac{1}{\sigma}\left[\frac{\partial}{\partial x_j}\left((\nu+\hat{\nu})\frac{\partial \hat{\nu}}{\partial x_j}\right)+c_{b2}\frac{\partial \hat{\nu}}{\partial x_i}\frac{\partial \hat{\nu}}{\partial x_i}\right] \quad (1)$$

2.2 Hydrodynamics Theory of Propeller

According to [2], the propeller performance characteristics in open water test that indicates the behavior of the propeller in uniform flow with steady load. The performance data are given the form of dimensionless propeller thrust and torque, K_T and K_Q, plotted against the dimensionless ship speed called advance coefficient, *J*. the dimensionless quantities are defined as:

$$J=\frac{V_a}{n.D}, \qquad K_T=\frac{T}{\rho n^2 D^4}, \qquad K_Q=\frac{Q}{\rho n^2 D^5}, \qquad \eta=\frac{J}{2\pi}\frac{K_T}{K_Q} \quad (2)$$

where ρ is the water density, n the number of propeller rotations per second and D the propeller diameter. The dimensional quantities are propeller thrust, T, propeller torque, Q and ship speed, V.

3.0 SIMULATION CONDITION

3.1 Principal Data of Propeller

In this paper, two different number of propeller blades models were used to investigates. The principal dimension of the propeller, which composes with three or four number of blades is clearly presented in Table 1.

Table 1: Principle dimensions of propeller

Geometrical parameters	Full Scale	Model Scale
Diameter (mm)	3650	119.25
AE/AO	0.695	0.695
P/D	1.013	1.013
Pitch (mm)	3697.45	120.83
Scale	1:30.6	
Propeller Orientation	Right-hand rotation	

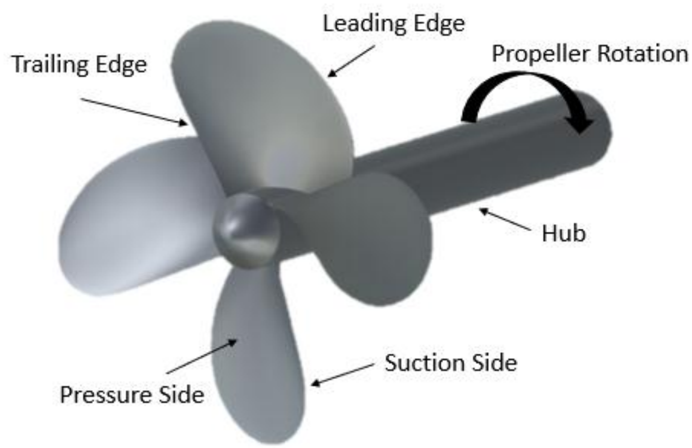


Figure 1: 3D view of propeller geometries.

3.2 *Simulation Condition*

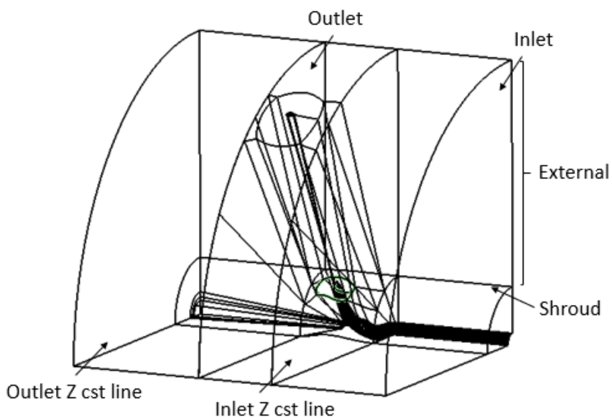
In this study, there are the parametric studies to be focused to achieve the objectives. The parametric studies were based on Table 2 below:

Table 2. The simulation conditions.

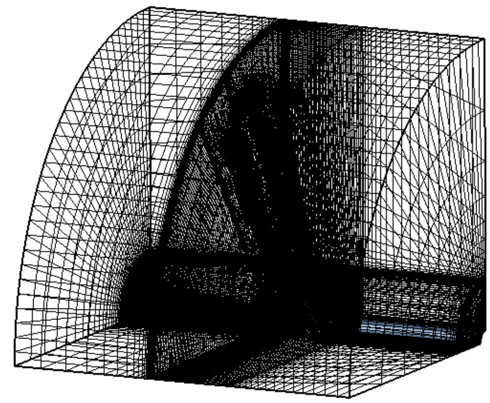
Number of Blades (Z)	Number of RPM
3	1200
4	

4.0 COMPUTATIONAL DOMAIN AND GRID GENERATION

The automatic hexahedral structured-grid generator AutoGrid5^{TM} software from NUMECA has been used for the meshing process.



(a) Bondary condition



(b) Mesh generation

Figure 2: (a) Bondary condition ; and (b) the computation domain associated with 4 number of propeller blades mesh model.

The automatic grid generator needs to define the geometry such as leading edge, trailing edge, pressure side, suction side and hub of the propeller. The meshing procedure continue to defines a fictitious shroud line as showed at Figure 3, which splits the upstream and downstream domains into a main channel. The main channel extends from the hub to the blades tip with no artificial tip gap is needed.

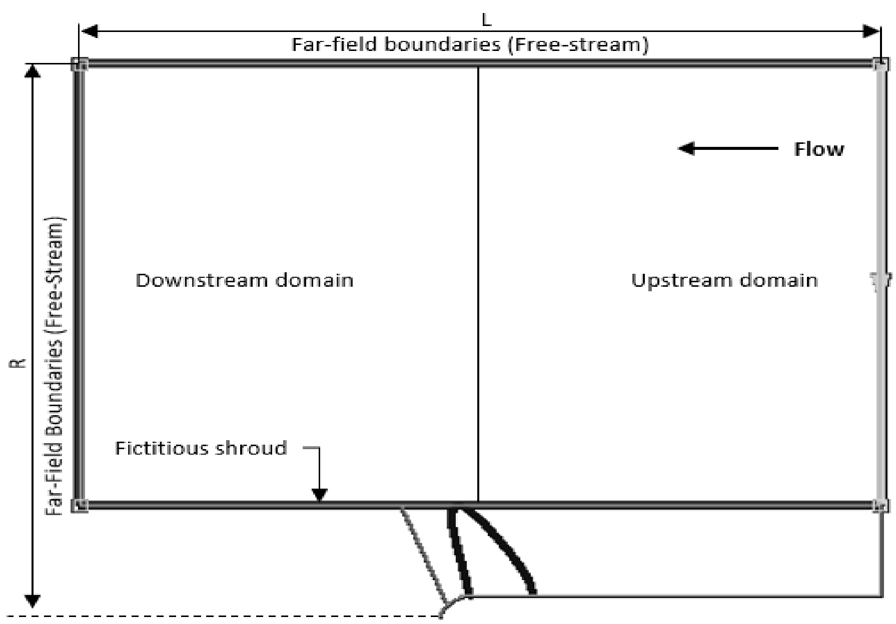


Figure 3: Upstream and downstream location in domain.

The blades and the hub, including the infinite cylinders, are considered as rotating viscous walls. It has to be noticed that this assumption has been taken in the two calculations. The mesh presents therefore a clustering in the vicinity of the hub. The domain extends radially to a radius R corresponding to five times the rear rotor radius, while the axial length L corresponds to six times this same radius.

The mesh point distribution is guided by two main criteria. A high-density distribution must be kept in the near-field of the blades for acoustic requirements while the first cell size on solid surfaces must be consistent with the low Reynolds turbulence models used here. The nodes number of this scale-model reaches 2.8 million. The first cell size is set at 1.0µm for blades and hub surfaces, keeping y+ values below 1.14 on all solid surfaces. The O4H blade-to-blade (B2B) grid topology uses 97 grid points in the pitchwise direction. Therefore, Figure 4 present the periodic matching patches of the upstream and downstream domain in B2B view.

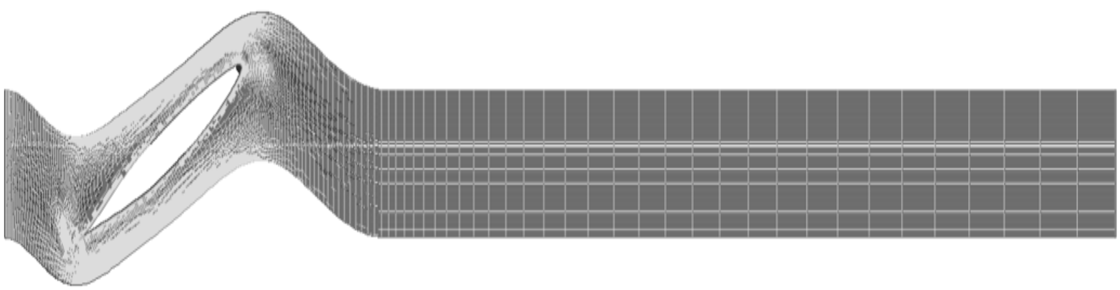


Figure 4: Mesh B2B view for propeller.

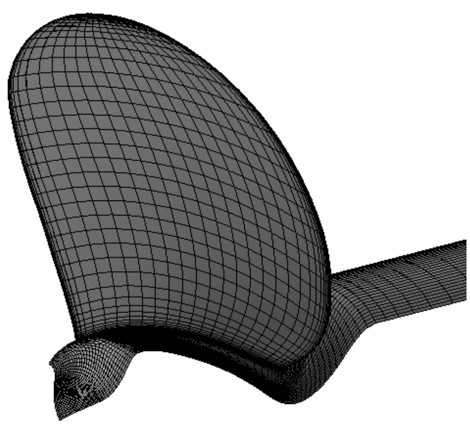


Figure 5: Surface mesh for single blade of propeller.

Referring to the meshing procedure, the B2B and 3D mesh as showed at Figure 4 and 5 will have generated without any negative cells in a quality report. A time-marching algorithm is used to converge to the steady state solutions of the time-mean equation. Multigrid method and local time-stepping are also used to speed-up the convergence. Turbulence is modeled by the one-equation Spalart-Allmaras model. The boundary condition is set the velocity of the fluid flow and the rotational speed for rotating machinery. This solution serves using medium grid and full multigrid start-up procedure. The convergence criteria considered here require a fully stabilization of the global quantities, axial thrust and torque on blades surface. In accordance with the best practice guidelines for turbo-machinery application of NUMECA Fine^{TM}/ Turbo. The average duration of every simulation was about 7 hours (double solver precision) on a HP Z820 Workstation PC with processor Intel® Xeon® CPU ES-2690 v2 @ 3.0GHz (2 processor) associated with the installed memory RAM of 32.0 GB and 64-bit operating system.

Table 3: Mesh Independent study on propeller geometry

Case	Grid Level	Total Number of Cell Meshing	Thrust Coefficient, K_T	Torque Coefficient, K_Q	Efficiency, η
A	Medium	1,789,042	0.3679	0.0554	0.1057
B		2,559,546	0.3719	0.0556	0.1064
C		2,817,090	0.3727	0.0557	0.1065
D		4,000,666	0.3744	0.0555	0.1073

The meshing generation of the propeller was created in AutoGrid5 v12.2 software. It should note that an adequate number of mesh is very important for proper and accurate simulation. Hence, a mesh independent study may need to be performed for four different total number of cell meshing. Referring to mesh independent study result, the case C with 2.8 million total number of cell meshing was selected in all computed simulation of the propeller model accuracy of the CFD solution. This can be explained by the fact the 4.0 million total number of cell meshing were unnecessary due to its insignificant influence into the computational result of the thrust coefficient, torque coefficient and efficiency. In the final stage of the CFD simulation, a package software in CFView was used to visualize the scalar torque for pressure and suction side for all various configuration of the propeller as displayed in Figure 6.

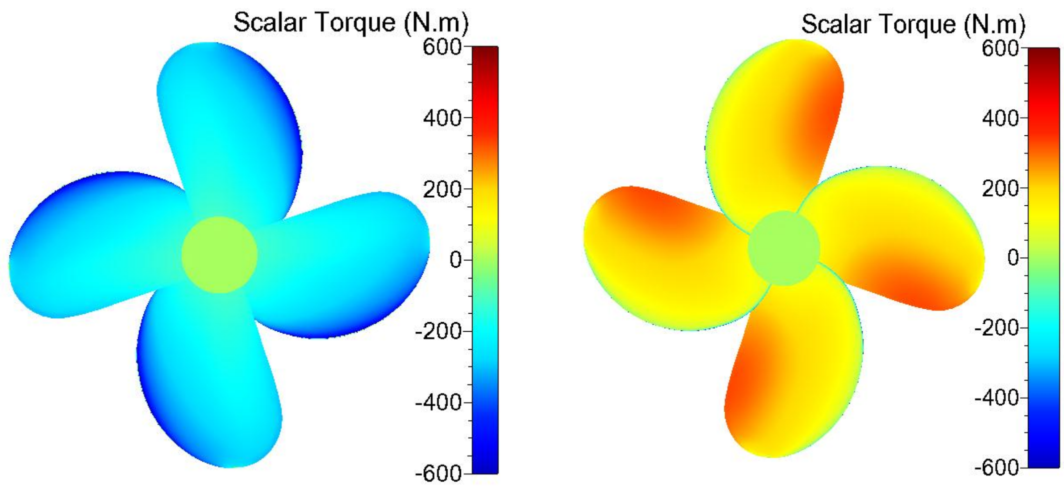


Figure 6: Pressure and suction side of four number of propeller blades, J=0.1, RPM=1200.

5.0 RESULT AND DISCUSSION

The analysis for thrust coefficient, torque coefficient and efficiency of the propeller in various number of propeller blades was analyze in the steady state have been presented and appropriately discussed. In this study, the computational fluid dynamics (CFD) approach was utilized to obtain the propeller performance.

5.1 Effect of Various Number of propeller Blades

Referring to the Figure 7, the K_T and K_Q decreases by increasing the advance ratio, J. The effect to the K_T occurs due to low axial velocity, the water surrounding the propeller will be accelerated from a low velocity. While at higher advance ratio, the water surrounds the propeller already moving at high velocity and makes less change to the water velocity [22]. Besides that, the effect to torque coefficient due to the decreasing drag force on the blade surface. According to [22], the water pressure surrounding the blade was high when the water condition at a low velocity and directly contributes as pressure drag on the blade surface. While at high water velocity, the water pressure surrounding the blades will drop slowly and the pressure drag also drop. This can be explained by the fact that the lowest pressure drags at leading edge (blue color) resulted in higher scalar torque value.

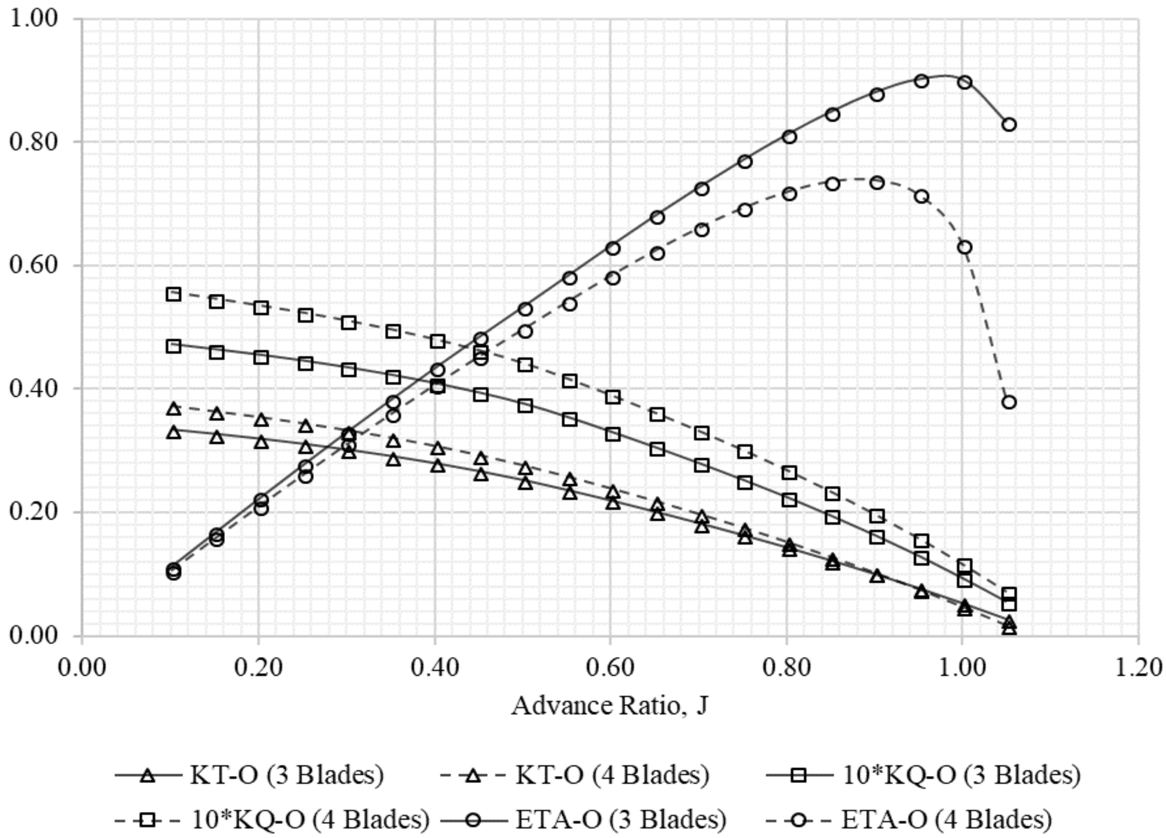
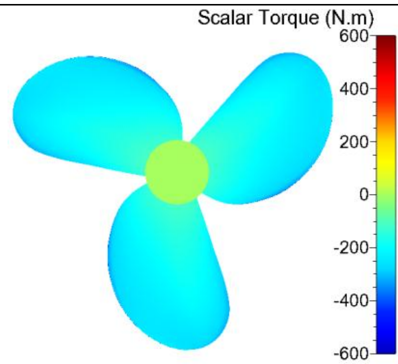
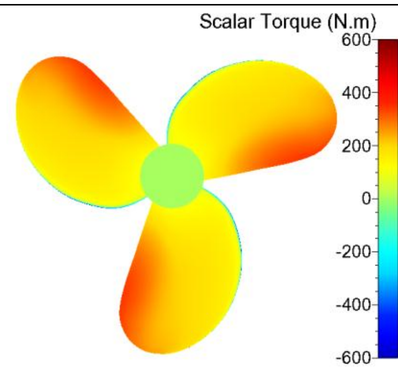


Figure 7: Thrust coefficient, torque coefficient and efficiency for propeller at various number of blades with 1200 RPM.

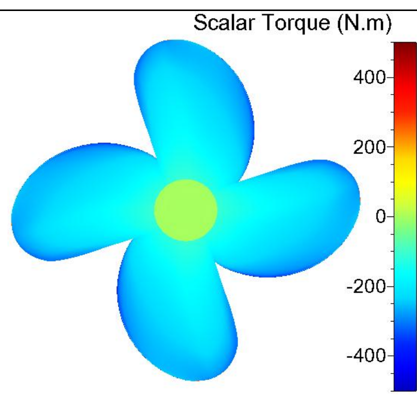
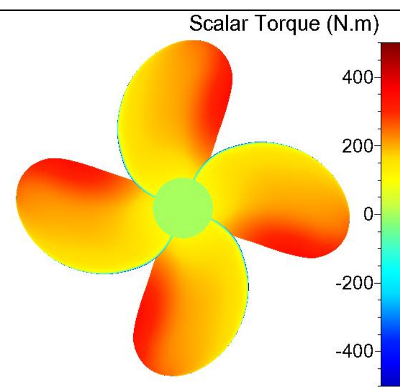
Table 4: Thrust coefficient, torque coefficient and efficiency for propeller at various number of propeller blades, Z.

Z = 3				Z = 4			
Advance Ratio, J	Thrust Coefficient, K_T	Torque Coefficient, K_Q	Efficiency, η	**Advance Ratio, J**	Thrust Coefficient, K_T	Torque Coefficient, K_Q	Efficiency, η
0.10	0.3338	0.0472	0.1125	**0.10**	0.3727	0.0557	0.1065
0.15	0.3268	0.0464	0.1681	**0.15**	0.3638	0.0546	0.1590
0.20	0.3193	0.0455	0.2232	**0.20**	0.3546	0.0535	0.2108
0.25	0.3109	0.0446	0.2775	**0.25**	0.3445	0.0524	0.2617
0.30	0.3015	0.0435	0.3310	**0.30**	0.3334	0.0511	0.3116
0.35	0.2912	0.0423	0.3834	**0.35**	0.3213	0.0497	0.3603
0.40	0.2797	0.0409	0.4349	**0.40**	0.3079	0.0481	0.4077
0.45	0.2668	0.0394	0.4849	**0.45**	0.2930	0.0463	0.4534
0.50	0.2525	0.0377	0.5337	**0.50**	0.2766	0.0442	0.4974
0.55	0.2365	0.0355	0.5832	**0.55**	0.2583	0.0417	0.5416
0.60	0.2194	0.0331	0.6331	**0.60**	0.2389	0.0390	0.5845

0.65	0.2014	0.0306	0.6814	**0.65**	0.2186	0.0362	0.6248
0.70	0.1829	0.0280	0.7277	**0.70**	0.1975	0.0332	0.6616
0.75	0.1637	0.0253	0.7718	**0.75**	0.1754	0.0302	0.6940
0.80	0.1437	0.0225	0.8129	**0.80**	0.1522	0.0269	0.7201
0.85	0.1226	0.0195	0.8500	**0.85**	0.1277	0.0234	0.7368
0.90	0.1007	0.0164	0.8814	**0.90**	0.1021	0.0198	0.7393
0.95	0.0778	0.0130	0.9028	**0.95**	0.0751	0.0159	0.7155
1.00	0.0534	0.0094	0.9020	**1.00**	0.0465	0.0117	0.6336
1.05	0.0279	0.0056	0.8334	**1.05**	0.0165	0.0072	0.3822

J	Pressure Side	Suction Side
0.95		

(a) Scalar torque for optimum advance ratio at Z=3.

J	Pressure Side	Suction Side
0.90		

(b) Scalar torque for optimum advance ratio at Z=4

Figure 8: (a) Z=3 and (b) Z=4 Scalar torque on pressure side and suction side

Besides that, the number of propeller blades has a positive effect on the open water performance [23]. As showed in Figure 7 and detail result presented in Table 4, the efficiency for Z=3 and Z=4 increase steadily with advance ratio from J= 0.1 up to 0.95 and J=0.1 up to 0.90. However, the Ƞ decreased for the further advance ratio. The propeller optimum, Ƞ value for Z=3 and Z=4 occurred at about J= 0.95 and 0.90 respectively. For Z = 3, peak performance was archive at more than 73.93% while for Z = 4 at 90.28%. In the other hand, the K_T at peak Ƞ for Z=3 achieved at 7.78%, Z=4 at 10.21%. As compared with various number of blades, the propeller optimum for Z=3 give the best efficiency. This can be explained by less total blades area that create less turbulence and reduce scrambling up each other's water flow.

6.0 CONCLUSION

The Computational Fluid Dynamics (CFD) simulation on hydrodynamic analysis of propeller is performed using NUMECA Fine^{TM}/Turbo software. The hydrodynamic analysis of propeller was summarized as follow:

i. The higher value of J lead to decrease K_T and K_Q due to low axial velocity and pressure drag surrounding the propeller.

ii. The Ƞ curve maintained, a large drop in Ƞ at higher values of J and a linear decrease in efficiency at lower value of J.

iii. As compared with various number of blades, the propeller optimum for Z=3 at J= 0.95 with K_T= 77.80%, 10*K_Q= 1.30% and Ƞ= 90.28% give the best efficiencies.

In general, the effect of number propeller blades, Z has positive influence on the open water characteristics of the marine propeller and a propeller with three number of blades gives the best efficiency. Therefore, these CFD results are useful as preliminary prediction for propel the ship at desired forward speed.

ACKNOWLEDGEMENT

The authors wish to greatly thank for the special financial support from School of Ocean Engineering Universiti Malaysia Terengganu.

REFERENCES

1. Husaini, M., Z. Samad, and M.R. Arshad. *Optimum Design of URRG-AUV Propeller Using PVL*. in *2nd Technical Seminar on Underwater System Technology: Breaking New Frontiers*. 2008.
2. Mizzi, K., et al., *Design optimisation of Propeller Boss Cap Fins for enhanced propeller performance.* Applied Ocean Research, 2017. **62**: p. 210-222.
3. Zhu, Z.-f. and S.-l. Fang, *Numerical investigation of cavitation performance of ship propellers.* Journal of Hydrodynamics, Ser. B, 2012. **24**(3): p. 347-353.
4. Rahman, A., M.R. Ullah, and M.M. Karim, *Marine Propeller Design Method based on Lifting Line Theory and Lifting Surface Correction Factors.* Procedia engineering, 2017. **194**: p. 174-181.
5. Epps, B., J. Ketcham, and C. Chryssostomidis. *Propeller blade stress estimates using lifting line theory*. in *Proceedings of the 2010 Conference on Grand Challenges in Modeling & Simulation*. 2010. Society for Modeling & Simulation International.
6. Ekinci, S., *A practical approach for design of marine propellers with systematic propeller series.* Brodogradnja: Teorija i praksa brodogradnje i pomorske tehnike, 2011. **62**(2): p. 123-129.
7. Taheri, R. and K. Mazaheri, *Hydrodynamic Optimization of Marine Propeller Using Gradient and Non-Gradient-based Algorithms.* Acta Polytechnica Hungarica, 2013. **10**(3): p. 221-237.
8. Elghorab, M., et al. *Experimental Study of Open Water Non-Series Marine Propeller Performance*. in *Proceedings of World Academy of Science, Engineering and Technology*. 2013. World Academy of Science, Engineering and Technology (WASET).
9. Arazgaldi, R., A. Hajilouei, and B. Farhanieh, *Experimental and numerical investigation of marine propeller cavitation.* 2009.
10. Prakash, S. and D.R. Nath, *A computational method for determination of open water performance of a marine propeller.* International Journal of Computer Applications, 2012. **58**(12).
11. Maghareh, M. and H. Ghassemi, *Propeller Efficiency Enhancement by the Blade's Tip Reformation.* American Journal of Mechanical Engineering, 2017. **5**(3): p. 70-75.
12. Ting, F.Y., et al., *Development of Seakeeping Test and Data Processing System.* International Journal of Computational Engineering Research (IJCER), 2015: p. 33-39.
13. Fitriadhy, A., et al., *Computational fluid dynamics investigation on total resistance coefficient of a high-speed" deep-V" catamaran in shallow water.* International Journal of Automotive and Mechanical Engineering, 2017. **14**: p. 4369-4382.
14. Fitriadhy, A., et al., *Computational fluid dynamics analysis on the course stability of a towed ship.* Journal of Mechanical Engineering and Sciences, 2017. **11**(3): p. 2919-2929.
15. Colley, E., *Analysis of Flow around a Ship Propeller using OpenFOAM*. 2012, Thesis, Curtin University.
16. Turunen, T., et al. *Open-water computations of a marine propeller using openfoam*. in *ECFD VI-6th European Congress on Computational Fluid Dynamics, Barcelona, Spain, 20-25 July 2014*. 2014.
17. Felicjancik, J., et al., *Numerical simulations of hydrodynamic open-water characteristics of a ship propeller.* Polish Maritime Research, 2016. **23**(4): p. 16-22.
18. Sun, H., et al., *Motion prediction of catamaran with a semisubmersible bow in wave.* Polish Maritime Research, 2016. **23**(1): p. 37-44.

19. Shamsi, R., S. Soheili, and A. Hamooni. *Hydrodynamic analysis of marine propellers using computational fluid dynamics*. in *Proceedings of the 17th international conference on mechanical engineering*.
20. Fitriadhy, A. and N.A. Adam, *Heave and pitch motions performance of a monotricat ship in head-seas.* International Journal of Automotive and Mechanical Engineering, 2017. **14**: p. 4243-4258.
21. Fitriadhy, A., N. Razali, and N. AqilahMansor, *Seakeeping performance of a rounded hull catamaran in waves using CFD approach.* Journal of Mechanical Engineering and Sciences, 2017. **11**(2): p. 2601-2614.
22. Husaini, M., Z. Samad, and M.R. Arshad, *Autonomous underwater vehicle propeller simulation using computational fluid dynamic*, in *Computational Fluid Dynamics Technologies and Applications*. 2011, InTech.
23. Boucetta, D. and O. Imine, *Numerical Simulation of the Flow around Marine Propeller Series*. Journal of Physical Science and Application, 2016. **6**(3): p. 55-61.