Journal of Applied Sciences

ISSN 1812-5654

science alert

ANSInet
an open access publisher
http://ansinet.com

Journal of Applied Sciences 14 (11): 1215-1219, 2014
ISSN 1812-5654 / DOI: 10.3923/jas.2014.1215.1219

Wageningen-B Marine Propeller Performance Characterization Through CFD

Kiam Beng Yeo, Wai Heng Choong and Wen Yee Hau
Unit Kajian Bahan dan Mineral, Sekolah Kejuruteraan dan Teknologi Maklumat,
Universiti Malaysia Sabah, Jalan UMS, Kota Kinabalu, 88400, Sabah, Malaysia

Abstract: This study presents the prediction of Wageningen B-Series three blade marine propeller hydrodynamics performance characteristics through Computational Fluid Dynamic application (SolidWorks Flow Simulation). Five propellers with pitch to diameter ratio (P/D) values (0.6, 0.8, 1.0, 1.2 and 1.4) variances were subjected to the computational flow analysis based on Reynolds Averages Navier-Strokes Equation-RANSE solver. The propeller performance characteristic such as thrust (K_T), torque (K_Q) and efficiency were computed under different types of advance ratio (J) values. Propeller with high P/D pitch-to diameter value is found more efficient where 1.4 P/D achieves about 60.38% (K_Q) and 39.17% (K_T) as compared to 0.6 P/D achieving 20.72% (K_Q) and 19.11% (K_T). The computation had shown that the geometry parameters such P/D has a significant influence on the propeller performance characteristics and the SolidWorks Flow Simulation had successfully utilized for this computational application.

Key words: Wageningen B series propeller, computational fluid dynamic, propeller performance characteristic, Reynolds averages Navier-Strokes equation

INTRODUCTION

Flow characteristics of marine propeller have become an important aspect for investigation in the fluid dynamics area in these years. Challenges of marine and mechanical engineering are focused on how to produce the most effective propulsion system of vessel for its movement against the water resistance as it moves under the sea. As a result, tremendous research activities were conducted in developing and exploring of theoretical and analytical solution for hydrofoil operating in seawater environment to continuously improve the propeller characteristics and geometries. The primary intention of marine engineering is to analyze and improve the flow characteristic of ship propulsion system to achieve the lowest energy consumption for its operation. Consequently, the quality of marine propeller to manipulate with its highest efficiency has become an important issue to be researched.

In most ships design, stern shape is not considered before the formation of propeller geometry. Nevertheless, the hull's flow separation cannot be ignored in the analytical work, which affects the propeller in gaining its thrust or force. Navier-Stoke equation is utilized to describe the hydrodynamic performance parameters, but cannot numerically solve the high Reynolds number flow behavior, which suffers from turbulent flow motion. In the condition where effects of turbulence and no-slip boundary on flow field estimations are important, Reynolds Average Navier-Stoke (RANS) equation with appropriate turbulence model can precisely predict the time average of flow characteristics undergone by the propeller (Valentine, 1993). Most of the researchers had carried out the analysis by the analytical or numerical method; inviscid or fully viscous or boundary layer method; and linearized or exact or partially linearized method (Justin, 1986). Combinations of any three characteristics are quite impossible and perhaps the Reynolds Averages Navier-Stoke (RANS) method can provide average among all these methods.

Computational fluid dynamic or CFD is a very useful method for solving numerous mechanics related problems. The advantage that CFD can perform is a great deal of analytical alternatives that could be gained in a short time (Mihaela, 2005). CFD can be used to analyze the flow field and physical parameters acting on the propeller body simultaneously. It utilizes such Navier-Stoke equation to solve most of the nonlinear flow for propeller especially when dealing with turbulent, shock wave and break wave (Miyata, 1997). Subhas *et al.* (2012) had reported that CFD estimation of marine propeller thrust and torque performances with different rotational speed are extremely ideal with minor differences with the experimental result. Kimura *et al.* (2009), Takashi and Jun (2009), Subhas *et al.*

Corresponding Author: Kiam Beng Yeo, Unit Kajian Bahan dan Mineral, Sekolah Kejuruteraan dan Teknologi Maklumat, Universiti Malaysia Sabah, Jalan UMS, Kota Kinabalu, 88400, Sabah, Malaysia
Tel: +6088320000 Ext: 3236

(2012) and Choong *et al.* (2013) are examples of CFD applications for predicting marine propeller performance characteristics.

This study presents the prediction of preliminary Wageningen B-Series marine propeller with variance pitch to diameter ratio performance characteristic through Computer Fluid Dynamic application. The prediction will provide an outcome for further understanding the propeller hydrodynamic characteristics, evaluating the performance through computational analysis approach, the influence of the geometry parameter towards the propeller performance and providing a optimum propeller pitch to diameter ratio.

MATERIALS AND METHODS

Propeller performance characteristic: Efficiency of a propulsion system relies on propeller performance that involves the thrust force, torque and efficiency, which can be determined by dimensionless analysis through propeller performance in thrust coefficient, torque coefficient and efficiency (K_T, K_Q, η) with respect to the advance coefficient, J (Mehdi *et al.*, 2010):

$$K_T = \frac{T}{\rho N^2 D^4} \quad (1)$$

$$K_Q = \frac{Q}{\rho N^2 D} \quad (2)$$

$$J = \frac{V_a}{ND} \quad (3)$$

where, T, Q, N, D and V_a are the propeller thrust, torque, rotational speed, diameter, water density and advance velocity, respectively. An optimum performance curve could then be obtained to demonstrate the efficiency that eventually decreases after a peak performance (Carlton, 2007). In most studies, the J advance ratio value, which is the forward movement of propeller with respect the diameter and rotational speed for determining the propeller efficiency based on the thrust and torque coefficient has been utilized. K_T and K_Q are performances in non-dimensional parameters are the main concerns in hydrodynamic behaviour of marine propeller, which are linearly decreasing with increasing of J values.

PROPELLER GEOMETRY

Wageningen B or Troost series of propellers are most widely used propeller series over the world, which is invented by Prof. Troost. It has a range of pitch to diameter ratio values (0.6 to 1.4), 3 to 7 number of blades and categories under comprehensive fixed pitch propeller from analysis and experiment by early inventors. The fixed pitch propeller geometry and particular details are listed in the Table 1, respectively and a 3-D model is as in Fig. 1.

Table 1: Wageningen B-series 3 blades propeller with pitch to diameter variances

Parameters	Values
Diameter (D)	250 mm
Mean pitch (P_m)	150 mm (P/D = 0.6)
	200 mm (P/D = 0.8)
	250 mm (P/D = 1.0)
	300 mm (P/D = 1.2)
	350 mm (P/D = 1.4)
Skew angle	0° (Balanced skew)
Rake angle	0°
Propeller expanded area ratio (EAR)	0.5

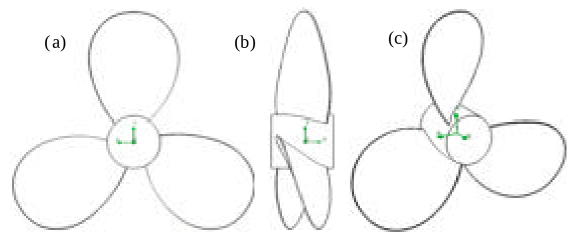

Fig. 1(a-c): Wageningen B-series 3 blades propeller geometry, (a) Front side, (b) Side view and (c) 3-D view

CFD CODE AND SOLVER

Turbulent flow characteristic is not the main aim in obtaining the performance of marine propeller over time, but the details about time taken to obtain its optimum performance, thrust and speed that can be generated from propulsion system. Chen and Lee (2003), in their report, adopting the RANS method to solve the inflow or effective wake of the propeller in nominal wake analysis without considered the operating condition. The operating condition effect is important that can affect the result of loading condition for thrust and pressure on the propeller. Thus, it is important to actually obtain the physical properties especially rotational speed in mean value over the fluctuation phenomenon. Mathematically, this can be done by substituting the Reynolds decomposition into Navier-Stokes Eq. 4 and averaging them (Herbert, 2004):

$$\frac{\partial \bar{u}}{\partial t} + \bar{u}\frac{\partial \bar{u}}{\partial x} + \bar{v}\frac{\partial \bar{u}}{\partial y} = -\frac{1}{\rho}\frac{\partial \rho}{\partial x} + v\left(\frac{\partial^2 \bar{u}}{\partial x^2} + \frac{\partial^2 \bar{u}}{\partial y^2}\right) - \frac{\partial^2 \bar{u}}{\partial x} - \frac{\partial \overline{u'v'}}{\partial y} \quad (4)$$

Where:

$\bar{u}$ = Time averaged value of velocity component in x axis

$\bar{v}$ = Time averaged value of velocity component in y-axis
$\bar{\rho}$ = Time averaged value of density component
$\overline{u'}$ = Time averaged value of the fluctuating velocity in x-axis
$\overline{v'}$ = Time averaged value of the fluctuating velocity in y-axis
t = Time dependant
x = X-axis coordinate
y = Y-axis coordinate

In Solidworks however, the RANSE solver is applied to solve the problem of rotating mechanism in the computational fluid domain (Dassault Systemes, 2012). In practice, this method greatly enhances the accuracy of arbitrary motion, which allows the propeller to act as crossing the fluid domain. For the computational domain, it is evaluated through a wide range of size as optimum as possible that can be observed from the animation flow trajectories. This computational domain shall be as large as possible to not be affected by the dynamic fluid flow due to boundary of wall, as on the actual performance of analytical work to be tested in an infinite fluid domain. For a poor size of computational domain settings of simulation, the boundary will affect the result. The optimum rectangular computational domain 4 m (height)×4 m (width)×7 m (length) has been used with consideration of computer processing unit performance capacity as in Fig. 2. According to Liu *et al.* (2012), error tolerance of 5% is acceptable for CFD application result.

Table 2: Boundary conditions and solver settings

Boundary conditions	Solver settings
Analysis type	External-exclude cavities without flow condition and internal space
	Physical features-heat conduction in solid, gravity and rotational
Default fluid	Liquids-water
Default solid	Solids in the model-AISI type 316L stainless steel
Wall conditions	Roughness-0 μm (Default settings)
Initial and ambient condition	Thermodynamic parameters
	Pressure 101325 Pa
	Pressure potential-activate
	Temperature-297.15 k (24°C)
	Velocity parameters
	Velocity in X direction-7.30773527 m sec^{-1} (for reynolds No. of 2×10^6)
	Relative to rotating frame-active
	Solid parameters
	Initial solid temperature-297.15 K (24°C)

The SolidWorks Flow Simulation mesh presentation has adopted a rectangular computational geometry in Cartesian coordinate system with the plane being diagonal to its axis. The computational mesh can either be system predefined or custom defined as based on the user accuracy requirement. Overall, the five propeller models mesh size is between 800,000 to 1,000,000 elements. As well as the computational boundary condition and solver setting are as indicated in the Table 2.

RESULTS AND DISCUSSION

The investigation of performance and parameters for P/D ratio range was from 0.6 to 1.4. Overall, the computed results trend predicts closely to that of Carlton (2007) results. The finding shows that the propeller efficiency will increase at a lower slope to its highest efficiency, but decreases drastically after the optimum range. The result trend of each P/D values is similar to each other and each profile has a different peak efficiency with respect to the J values. From the Fig. 3 and 4, the K_Q and K_T values are decreasing as J values increases or increasing of rotor speed, N with reduction of advance velocity V_a. Also, the highest $10K_Q$ and K_T value was P/D = 1.4 and the lowest P/D = 0.6, which reduced with the reduction of P/D. As the P/D ratio increase, the efficiency value also increased. The highest efficiency for various P/D ratio was 1.4 of more than 67%, followed by P/D = 1.2 at 61.5%, P/D = 1.0 at 61%, P/D = 0.8 at 59% and the lowest was P/D = 0.6 at 54%.

From the trend of intersection of $10K_Q$ and K_T to, low P/D ratio propeller intersected the efficiency curves at lower figures and higher P/D ratio of propeller profile had

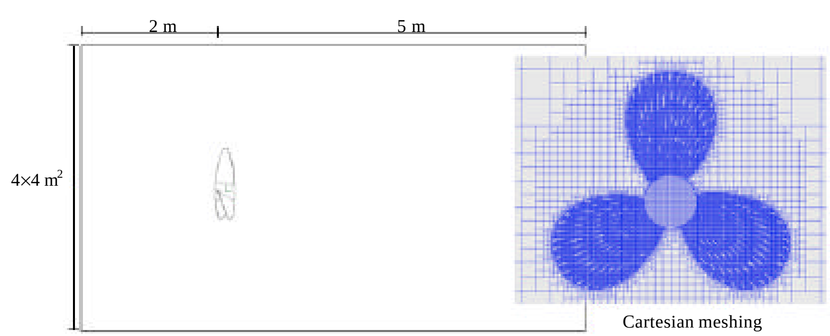

Fig. 2: Computational domain and Cartesian meshing

Table 3: Best performance of torque-thrust-efficiency parameters for different pitch to diameter ratio

P/D ratio	Torque-efficiency	Thrust-efficiency
0.6	20.72% at J = 0.127	19.11% at J = 0.131
0.8	31.81% at J = 0.266	25.57% at J = 0.213
1.0	43.63% at J = 0.452	30.91% at J = 0.312
1.2	53.34% at J = 0.688	35.82% at J = 0.425
1.4	60.38% at J = 0.924	39.17% at J = 0.537

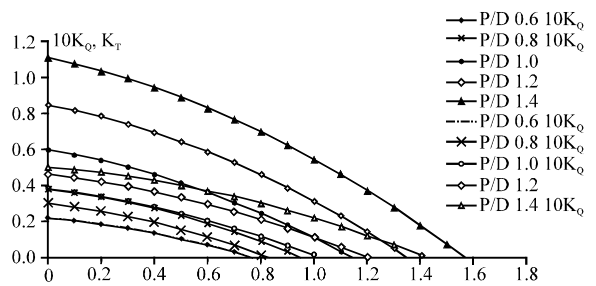

Fig. 3: Graph of $10K_Q$ and K_T with respect to J for different P/D ratio

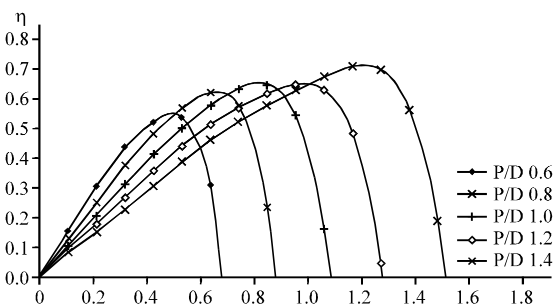

Fig. 4: Graph of with respect to J for different pitch to diameter ratio (P/D)

achieved a much greater value. The optimum performance of torque and thrust when it first strikes the efficiency curve is also tabulated in Table 3.

CONCLUSION

A general methodology of predicting the Wagenigen B-Series 3 blades propeller performance characteristics through RANSE based CFD application is presented. The efficiency of Wagenigen B-Series propeller has increased in a small inclination until it reached to the peak efficiency value and then drops drastically right after the optimum performance. The propeller geometry such as the P/D value has great influence on the propeller performance efficiency. Higher P/D value propeller type is recommended for higher performance efficient application requirement. At this stage, the prediction through SolidWorks Flow Simulation could only be considered as preliminary study and the obtained results shall require further veMarch 8, 2014rification through other CFD application such as Ansys CFX or experimental results.

ACKNOWLEDGMENTS

The authors would like to express their appreciation to the Unit Kajan Bahan dan Mineral (Materials and Minerals Research Unit), School of Engineering and Information Technology, Universiti Malaysia Sabah and the Ministry of Higher Education of Malaysia for the financial support through the research Grant FRG0249-TK-2/2010 and FRG0247-TK-2/2010.

REFERENCES

Carlton, J.S., 2007. Marine Propellers and Propulsion. 2nd Edn., Butterworth-Heinemann, Oxford, UK.

Chen, H.C. and S.K. Lee, 2003. Chimera RANS simulation of propeller-ship interaction including crash-astern conditions. Proceedings of the 13th International Offshore and Polar Engineering Conference, May 25-30, 2003, Honolulu, Hawaii, USA., pp: 334-343.

Choong, W.H., K.B. Yeo, F. Tamiri and K.T.K. Teo, 2013. Outboard marine propeller performance analysis through CFD modelling. Proceedings of the 15th International Conference on Computer Modelling and Simulation, April 10-12, 2013, Cambridge, UK., pp: 310-313.

Dassault Systemes, 2012. Solidworks flow simulation. Dassault Systemes, Solidworks Corporation, Velizy-Villacoublay, France. https://www.solidworks.com/sw/products/simulation/flow-simulation.htm.

Herbert, O., 2004. Prandtl's Essential of Fluid Mechanics. 2nd Edn., Springer, New York.

Justin, E., 1986. Marine propellers. Annu. Rev. Fluid Mech., 18: 367-403.

Kimura, K., T. Kawamura, Z. Huang, A. Fujii and T. Taketani, 2009. Study on unsteady cavitating flow simulation around marine propeller using a RANS CFD Code. Proceedings of the 7th International Symposium on Cavitation, August 16-20, 2009, Ann Arbor, MI.

Liu, D., F. Hong, F. Zhao and Z. Zhang, 2012. The CFD analysis of propeller sheet cavitation. China Ship Scientific Research Center, Wuxi, China.

Mehdi, N., M.J. Abbasi and A.M. Amini, 2010. Assessment of marine propeller hydrodynamic performance in open water v ia CFD. Proceedings of the International Conference on Marine Technology, December 11-12, 2010, Dhaka, Bangladesh, pp: 35-44.

Mihaela, A., 2005. Developments in the design of ship propeller. Scientific Bulletin of the Politehnica University of Timisoara Transactions on Mechanics, University of Galati.

Miyata, H., 1997. Time-marching CFD simulation for moving boundary problems. Proceedings of the 21st Symposium on Naval Hydrodynamics, June 24-28, 1996, Trondheim, Norway.

Subhas, S., V.F. Saji, S. Ramakrishna and H.N. Das, 2012. CFD analysis of propeller flow and cavitation. Int. J. Comput. Applic., 55: 26-33.

Takashi, K. and A. Jun, 2009. Numerical analysis of steady and unsteady sheet cavitation on a marine propeller using a simple surface panel method SQCM. Proceedings of the 1st International Symposium on Marine Propulsors, June 22-24, 2009, Trondheim, Norway.

Valentine, D.T., 1993. Reynolds-averaged navier-stokes codes and marine propulsor analysis. Hydromechanics Directorate Research and Development Report, Carderock Division, Naval Surface Warface Center, Bethesda, MD., USA.