Journal of Applied Sciences

ISSN 1812-5654

science
alert

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

Journal of Applied Sciences 14 (10): 1043-1048, 2014
ISSN 1812-5654 / DOI: 10.3923/jas.2014.1043.1048
© 2014 Asian Network for Scientific Information

Computational Development of Marine Propeller Design

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

Abstract: This study demonstrates the computational development of marine propeller, starting with the formation of the three-dimensional model through computational software, followed by the fluid dynamics analysis for different profiles such as number of blades, pitch to diameter ratio, rake angle and skew angle. Analysis results showed that marine propeller with three or four blades have better performance while increasing of skew angle decreases the swirling problem. The importance of mesh refinement and locally defined cell sizes are also presented.

Key words: Marine propeller, flow trajectories, mesh refinement, computational domain

INTRODUCTION

Marine propeller design is limited by constraints of hydrodynamic parameters such as Reynolds number, cavitations phenomenon and maximum diameter for weight loading (Justin, 1986). Since water are denser compared to air in physical properties, so the ability of lift force production for thrust amount on each unit of the blade area are also limited. A good computational method to develop and investigate the flow dynamics of marine propeller is important for three-dimensional flow analysis. Theoretical calculation is too complex to consider the irrotational non-uniform fluid behaviour on the inflow of propeller back surface, mixing of irrotational and rotational flow on propeller revolving region and rotational flow behind the face surface of marine propeller.

Rake, skew and pitch angles are some critical parameters in marine propeller analysis. Small rake can reduce drag force on propeller surface, hold water better for each rotation, thus increasing the thrust force and efficiency of the propulsion system. Ghassemi (2009) 5 skew angle profiles found that skew formation on the propeller design could provide stability but low skew angle would induce vibration to the propulsion system.

Pitch angle is the changes of helix angle of the complete blade section located at the root section of the propeller blade. Increasing of pitch to diameter ratio, p/D reduce the interaction area with inflow fluid but increase the face interaction to the rotational motion as it faces perpendicular to the acting rotational force with fluid (Bernitas and Ray, 1982). Theoretically, larger blade area increases the sweeping area perpendicular to the water surface as it rotates which results in higher torque (Oosterveld, 1975). However, increasing of blade area also reduces the efficiency of peak performance curve at the same time (Tsakonas *et al*., 1972).

Computational Fluid Dynamics (CFD) is a very useful approach used routinely in optimization and design for assessment of marine engineering system (Mihaela, 2005). It uses Navier-Stoke equation to solve the nonlinear flow for marine propeller especially when dealing with turbulent, shock wave and break wave (Miyata, 1997). The software decomposes spatial domain into tiny cells to form the volume mesh and a suitable algorithm is applied to solve the equation of motion (Wu, 2010).

Subhas *et al*. (2012) conducted computational estimation of thrust and torque performance of marine propeller for various rotational speeds found an extremely ideal or same result compared with experimental work. In Liu *et al*. (2012), the uncertainties in CFD had lowered the thrust and torque performance by 5% to be the cause of software code lacking of grid density or too much of numerical dissipation in the vicinity of cavity-fluid interface. Researchers such as Takayuki *et al*. (2003) and Shin and Shitalkumar (2003) have also shown successful analysis work of marine propeller using CFD.

GENERATION OF MARINE PROPELLER BY CAD

Initially, the Computer Aided Design (CAD) solid model of the propeller is developed via a two-dimensional coordinate database of the specific hydrofoil series. Two major types of pitch distribution, commonly offered and manufactured, are the full distribution and 80% pitch distribution. Full distribution has a pitch distance as from the hub to tip while 80% pitch distribution is formed by

---

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

taking 80% of pitch length from the hub section and slowly increased until the complete distribution as the design requirement. Figure 1 shows the pitch distribution of two common profiles of marine propeller design.

Gerr (2001) suggested that if the blade angle from hub to tip is the same it will produce tip flow that race ahead of the root section, as the tip rotates with faster linear velocity in a single rotational. From the hydrofoil coordinate database, size of hydrofoil shall be examined due to different enlargement required for the diameter of the propeller for the specific size of blade section. Also, the hydrofoil that formed shall have various attack angles that is needed to be adjusted by law of cosines for the coordinate system. This angle of attack for each hydrofoil is wrapped on cylindrical surface that has various sectional radials from two-dimensional plain surfaces to three-dimensional complex surfaces. Three-dimensional hydrofoils are drawn by 3D sketches, loop by surface loop function, then formed to solid body from surface body. By using the required hub size and length, required number of blades is added evenly around the circular hub. Figure 2 displayed the process of the formation for the marine propeller by using the CAD technique.

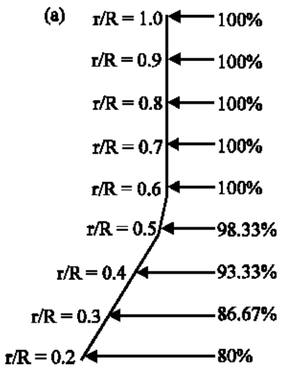

Fig. 1(a-b): (a) An 80% pitch distribution and (b) Full distribution of marine propeller

FLOW TRAJECTORIES WITH INFLOW, PROCESSING AND OUTFLOW

In the comparison of flow trajectories that consisted of inflow and outflow for marine propeller, 2 blades propeller does not have swirling of fluid on the face surface of the hub. Then, the swirling condition increased as the blades number increased. Low passed of inflow and high cutting capacity of blades provides more possibility for propeller to cause this vortex phenomenon

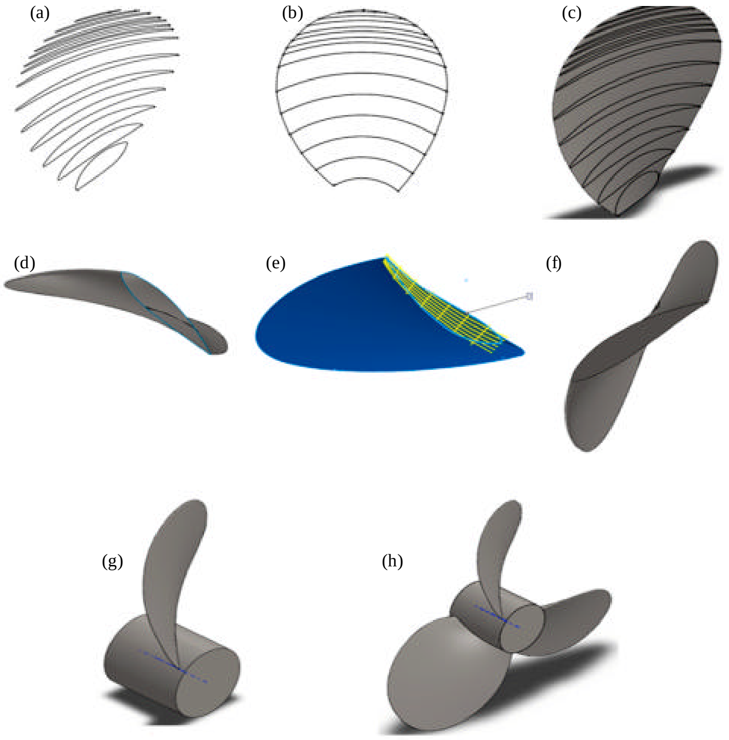

Fig. 2(a-h): Three-dimensional marine propeller generation by CAD software

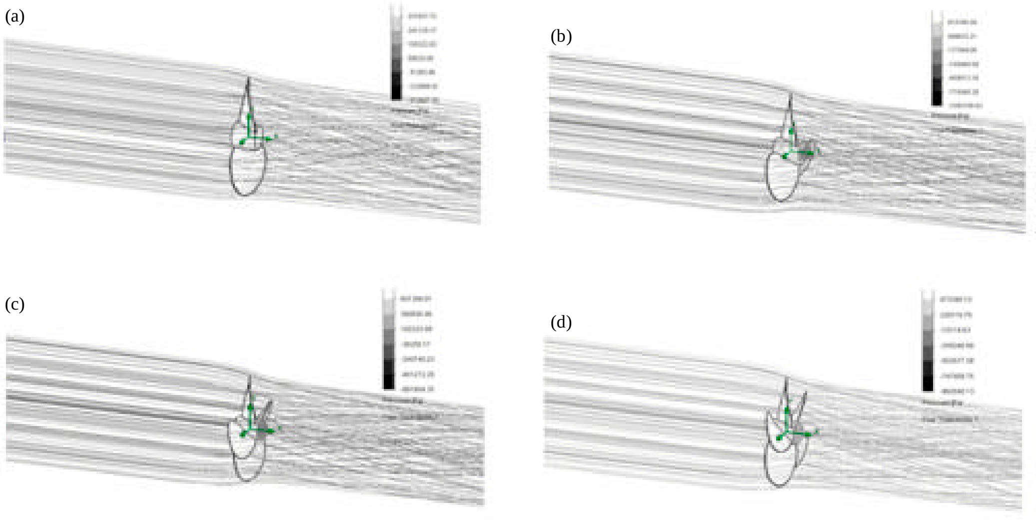

Fig. 3(a-d): Flow trajectories for marine propeller with different No. of blade, (a) 2, (b) 3, (c) 4 and (d) 5 blades

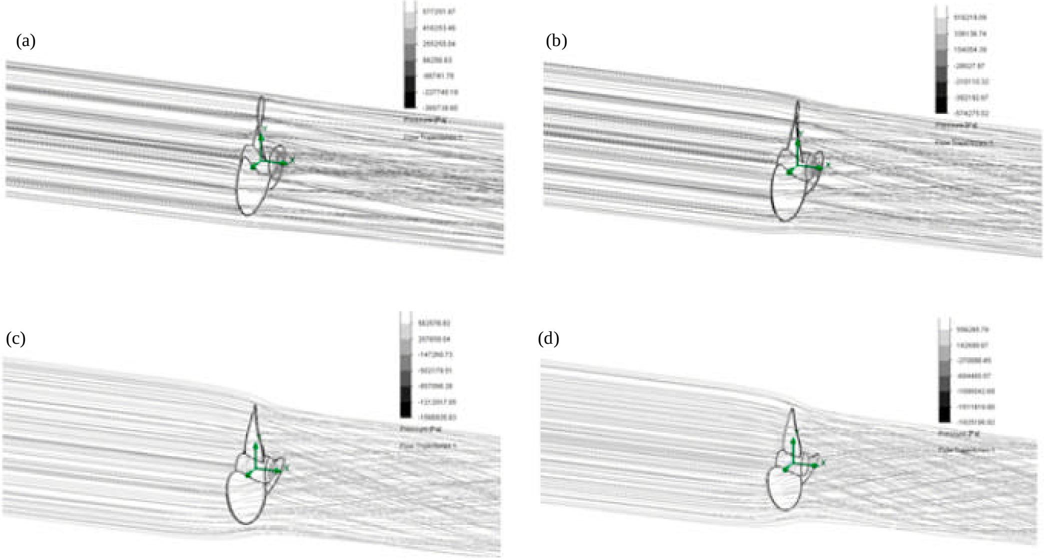

Fig. 4(a-d): Flow trajectories of 3 blades marine propeller at different p/D profile, (a) p/D = 0.6, (b) p/D = 0.8, (c) p/D = 1.2 and (d) p/D = 1.4

to occur. Also, as the blade number increased to 5, a straight and concentrate flow that has higher flow pass, remain as small cylindrical flow on the centre of the face hub. From the analysis, 3 and 4 blades marine propeller would be the best selection, where 3 blades offer good performance, acceleration and top end and 4 blades come with better hole shot, stability and fuel economy. Figure 3 shows the flow trajectories for marine propellers with different number of blades at p/D = 1.0, 0° rake angle and 0° skew angle.

Computational results show that the pitch to diameter ratio, p/D has a concentration of flow vortex on low p/D profile that is closed to the face of the hub surface, where the rotational outflow becomes highly focused and more stable on its flow pattern. However, p/D ratio doesn’t have a consistent formation of swirling near the face on the hub at the outflow. Higher p/D profile does not come with vortex phenomenon as other types of propeller geometry. Figure 4 also shows the flow trajectories of 3 blades marine propeller at different p/D profile.

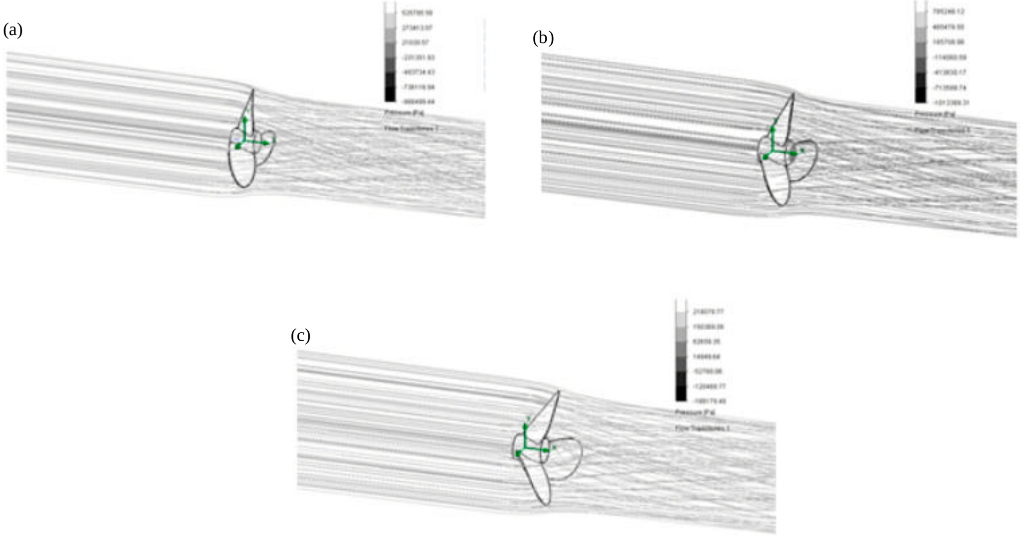

Fig. 5(a-c): Flow trajectories of 3 blades marine propeller at different rake angle, (a) 10° rake angle (b) 20° rake angle and (c) 13° rake angle

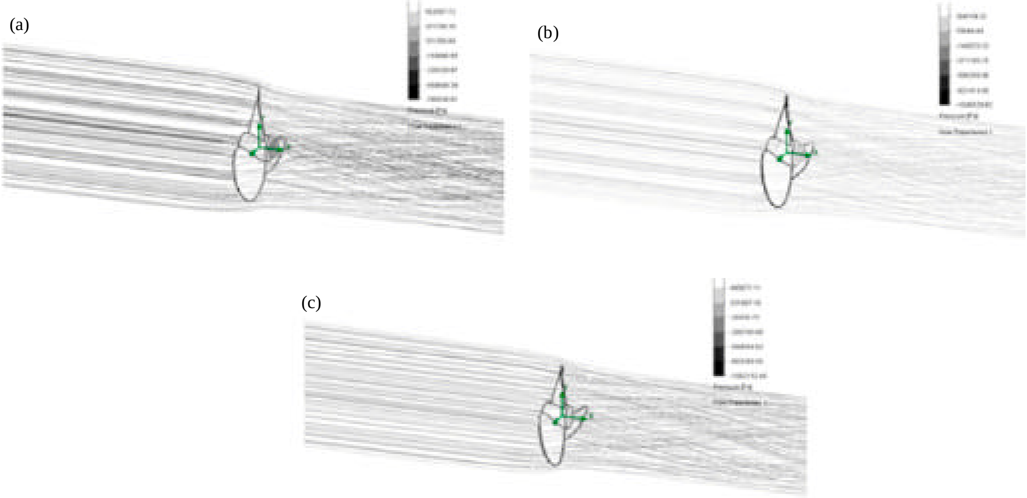

Fig. 6(a-c): Flow trajectories of a 3 blades marine propeller at different skew angle (a) 10° skew angle (b) 20° rake angle and (c) 13°rake angle

Rake angle profile does not come with clear trend of flow trajectories compared to other geometric changes. From the Fig. 5 it can be observed that 0 and 20° of rake angle had high swirling flow phenomenon occurring on the hub surface with 20° comes slightly small compared to 0°. While, 10 and 30° rake angle does not face any of the vortex flow problem. Since mixed flow does not occur on these profiles, the outflows of these two geometries were stable with less loss.

In the other hand, the skew angle profile has very stable decreasing of flow swirling as the skew angle increased to 30°. The trends of skew angles were stable and do not face any swirling of flow that formed vortex phenomenon as other profile. Thus, this propeller geometry has high efficiency because it does not have any losses of flow that dies off itself near to face of the hub surface as others. Figure 6 also shows the flow trajectories of a 3 blades marine propeller at different skew angle.

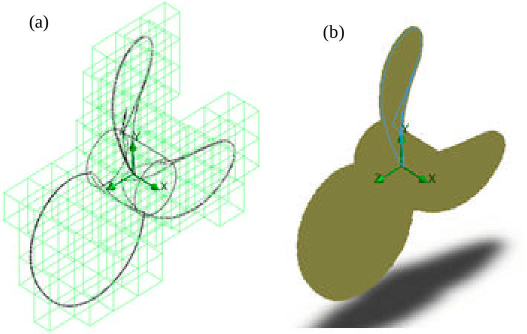

Fig. 7(a-b): Cells refinement on marine propeller (a) Low and (b) High

MESH GENERATION AND REFINEMENT

The refinement of numerical data from CFD analysis could increase the accuracy of the finding. Since flow simulation in CFD analysis software is used to generate the meshes it is split with various x, y and z planes that orthogonal to Cartesian global coordinate system by half each. For the marine propeller, the load distribution over the propeller surface is concerned and should have more a precise and smaller size of cells distribution for a complete range of studies.

In the other hand, the fluid medium surrounding the propeller has less concentration, the cells size can be larger compared to the propeller does not make much differences for in and out flow which is far from the propeller body. Enhancement of cells requires a much more precision on the edge and tip section of the propeller, since these regions are of much thinner sectional area with high changes of angle. In normal procedure of mesh refinement without stressing on the edge it would cause improper distribution and iteration that focus on unnecessary or less important regions.

Higher level of meshes refinement certainly would decrease the error but it will consume large amount of computational time instead. An accurate way of refinement would provide a sum of calculation data with lesser variances of error or precise numerical result for a set of repeating calculation. Also, most importantly this could provide a fine plotting of curve lines as iso-lines and surfaces plot over the solid surface of the propeller. Figure 7 shows the differences of refinement level which affects the accuracy of displayed result.

CONCLUSION

The importance of computational analysis to marine propeller study has been mentioned in the study and followed by the step by step generation of marine propeller model in CFD software. CFD analysis shows that 3 and 4 blades marine propeller would be the best selection, as it offers good performance, stability and fuel economy as compared to the rest. Computational results also showed that pitch to diameter ratio, p/D does not have a consistent formation of swirling near face on hub at outflow. Besides, the skew angle profile has very stable decreasing of flow swirling as the skew angle increased to 30° but the rake angle profile does not come with proper trend of flow trajectories compared to other geometric changes.

For propeller blade, optimization is highly needed on the edge or tip of the blade. These regions consisted of curvature with tip section that formed from hydrofoil section has sharp trailing edge. The smaller cells which enhanced or improved produces much rapid calculation for either fluid to solid, fluid to fluid and solid to solid cells interaction allows increases of accuracy in CFD analysis.

ACKNOWLEDGMENT

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 their support.

REFERENCES

Bernitas, M. and D. Ray, 1982. Optimal revolution B-series propellers. Department of Naval Architecture and Marine Engineering, Michigan, USA. http://deepblue.lib.umich.edu/handle/2027.42/91757.

Gerr, D., 2001. The Propeller Handbook: The Complete Reference for Choosing, Installing and Understanding Boat Propeller. McGraw-Hill, New York, ISBN: 9780071381765, Pages: 176.

Ghassemi, H., 2009. The effect of wake flow and skew angle on the ship propeller performance. Trans. B. Mech. Eng., 16: 129-158.

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

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

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. In: Twenty-First Symposium on Naval Hydrodynamics, NSB and NRC (Eds.). National Academy, USA.

Oosterveld, M.W.C., 1975. Further Computer-Analyzed Data of the Wageningen B-Screw Series. MSMB. Publisher, Rotterdam, Pages: 14.

Shin, H.R. and J. Shitalkumar, 2003. CFD validation for a marine propeller using an unstructured mesh based RANS method. Proceedings of the 4th ASME/JSME Joint Fluids Engineering Conference, July 6-10, 2003, Honolulu, Hawaii, USA.

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.

Takayuki, W., K. Takafumi, T. Yoshihisa, M. Masatsugu and S.H. Rhee, 2003. Simulation of steady and unsteady cavitation on a marine propeller using a RANS CFD code. Proceedings of the 5th International Symposium on Cavitation, November 1-4, 2003, Osaka, Japan.

Tsakonas, S., W.R. Jacobs and M.R. Ali, 1972. An exact linear lifting-surface theory for a marine propeller in a nonuniform flow field. Report SIT-DL-72-1509, Davidson Laboratory, New Jersey, USA., February 1972.

Wu, X., 2010. A Rapid Development Process for Marine Propellers through Design, Simulation and Prototyping. Memorial University of Newfoundland, St. John's, Newfoundland and Labrador, Pages: 244.