THE LITTLE REVIEW OF SIMULATION OF FLOW AROUND THE SHIP

Pham Thanh Cuong[1]

[1] *Faculty of Mechanical, Electrical and Electronic Technology, Thai Nguyen University of Technology, Vietnam*

ABSTRACT

The research describes the results of calculation of ship resistance, dynamic sinkage and trim in calm water condition by solving unsteady Reynolds Averaged Navier-Stockes equation (RANSE). The free surface is modeled by applying Volume-of-Fluid method. To achieve a high level of accuracy at some certain cells, the unstructured hexahedral grid type has been employed and locally refined in the importance regions. The effect of grid size, y+ value on the predicted results are analyzed. It is indicated that the satisfied accuracy can be obtained with meshes of lower million cells generated for half of the ship geometry. The well-known KCS containership is applied to verify and validate the accuracy of case studies. With the research objectives set out, the author organizes this research into four main parts as follows. Part 1: The Little Review; Part 2: Materials and Methods; Part 3: Numerical simulation; Part 4: Results and Discussions.

Keywords: *RANSE, Resistance, CFD, trim, sinkage.*

---

1. INTRODUCTION

In ship hydrodynamics computation, the problem of determining the ship resistance is a key factor in the design of the propulsion system. As it involves estimating the ship required horsepower to gain the design speed. In order to achieve a reliable result for ship resistance prediction, it is necessary to perform the model tests in towing tank. The obtained measured results then it will be extrapolated to the full scale based on extrapolation methods has been proposed by International Towing Tank Conference (ITTC). Nevertheless, the experimental method has disadvantage are time and cost consuming due to both model manufacturing and the experiment itself. Hence, this method is not used in the initial ship design stage.

2. MAIN CONTENTS

Nowadays, Computational Fluid Dynamics (CFD) has made a considerable progress in the field prediction of ship hydrodynamics in general and ship resistance in particular. With the fast advance of computational resources, CFD methods have been widely used in ship designing and ship resistance predictions. CFD methods provides relatively accurate results, fast and inexpensive in comparison with the experimental method. Moreover, they can provide the visualization of flow quantities, i.e. wall shear stress, pressure distribution on ship hull, wave elevation contour and streamlines, which provide designers information to develop or improve ship hull form with respect to minimize ship resistance. The group of CFD methods applied to predict ship hydrodynamics characteristics includes: Large Eddy Simulation, Potential flow theory and Reynolds Averaged Navier-Stokes equations (RANSE). At the moment, the most popular method is widely used in predict ship resistance is the RANSE method, since it gives a sufficient level of accuracy of obtained results for engineering purposes at acceptable computational time [2-10]. Hence, this paper uses the RANSE method for simulation flow around the ship.
Some useful results applying the RANSE method for simulation flow around the ship have been already achieved by researchers. Tahara et al. [12] and Kim et al. [13] have carried out comparison of turbulence models for prediction of wake around the ship. Islam et al. [14] performed uncertainty analysis in ship resistance evaluation for different ship model. Aiguo et al. [15] used RANSE method with turbulence model SST k-ω to predict the ship resistance for DTMB vessel. The deviation between the predicted results and the measured value varies from 1.25 to 3.86%.

Chengsheng et al. [16] used RANSE method with k-ε turbulence model to evaluate the ship resistance for DTMB vessel ship in model scale ranges from 0.12 to 2.36%. The error between the calculated results and the measured data is less than 4% in all range of Froude numbers. Those previous researches mentioned above gives a helpful data source for the further research flow around ship using RANSE method.

3. CONCLUSIONS

The above studies play an essential role in further research about the influence of turbulence models on the accuracy of ship resistance and discrepancies about flows around the ship. However, present studies reviewed above still lack analysis of the effect of turbulence model on the accuracy of obtained ship resistance, trim and sinkage, wave pattern, convergence time, and computational time-consuming. It is necessary to consider these mentioned issues to avoid the errors caused by choosing the turbulence model, reducing the calculation time, and maintaining accuracy. This is also the content that will be presented in future studies.

4. ACKNOWLEDGEMENT

This work was supported by the Thai Nguyen University of Technology.

5. REFERENCES

[1]. Holtrop, J., & Mennen, G. G. J. (1982). An approximate power prediction method. *International Shipbuilding Progress*, *29*(335), 166-170.

[2]. Król, P. (2021). Hydrodynamic state of art review: rotor–stator marine propulsor systems design. *Polish Maritime Research*, *28*(1), 72-82.

[3]. Raven, H., Van der Ploeg, A., Starke, A. R., & Eça, L. (2008). Towards a CFD-based prediction of ship performance—progress in predicting full-scale resistance and scale effects. *International Journal of Maritime Engineering*, *150*, 31-42.

[4]. Demirel, Y. K., Turan, O., & Incecik, A. (2017). Predicting the effect of biofouling on ship resistance using CFD. *Applied ocean research*, *62*, 100-118.

[5]. Bhushan, S., Yoon, H., Stern, F., Guilmineau, E., Visonneau, M., Toxopeus, S. L., ... & Grigoropoulos, G. (2019). Assessment of computational fluid dynamic for surface Combatant 5415 at Straight Ahead and Static Drift β= 20 deg. *Journal of Fluids Engineering*, *141*(5).

[6]. Bhushan, S., Xing, T., Carrica, P., & Stern, F. (2009). Model-and full-scale URANS simulations of Athena resistance, powering, seakeeping, and 5415 maneuvering. *Journal of Ship Research*, *53*(04), 179-198.

[7]. Roh, M. I., & Lee, K. Y. (2018). *Computational ship design*. Springer Singapore.

[8]. Molland, A. F., Turnock, S. R., & Hudson, D. A. (2017). *Ship resistance and propulsion*. Cambridge university press.

[9]. Larsson, L. (2010). Ship resistance and flow. *Published by The Society of Naval Architects and Marine Engineers, SNAME, The Principles of Naval Architecture Series, ISBN: 978-0-939773-76-3*.

[10]. Ferziger, J. H., Perić, M., & Street, R. L. (2002). *Computational methods for fluid dynamics* (Vol. 3, pp. 196-200). Berlin: springer.

[11]. Wang, J., & Wan, D. (2019). Numerical simulations of viscous flows around JBC ship using different turbulence models. In *11th International Workshop on Ship and Marine Hydrodynamics (IWSH2019)*.

[12]. Tahara, Y., Hanaoka, A., Higaki, K., & Takai, T. Investigation on Effective Turbulence Models for Predicting Tanker Stern Flows.

[13]. Kim, W. J., Kim, D. H., & Van, S. H. (2001). Comparison of Turbulence Models for the Prediction of Wakes around VLCC Hull Forms. *Journal of Ship and Ocean Technology*, *5*(2), 27-48.

[14]. Islam, H., & Soares, C. G. (2019). Uncertainty analysis in ship resistance prediction using OpenFOAM. *Ocean Engineering*, *191*, 105805.

[15]. Shia, A., Wu, M., Yang, B., Wang, X., & Wang, Z. (2012). Resistance calculation and motions simulation for free surface ship based on CFD. *Procedia Engineering*, *31*, 68-74.

[16]. Wu, C. S., Yang, L., Zhang, Z. R., Zhao, F., & Yan, K. (2010). CFD simulation of ship model free to sinkage and trim advancing in calm water. In *Proceedings of Gothenburg 2010: a workshop on numerical ship hydrodynamics, Gothenburg* (pp. 435-440).