称号及び氏名	博士（工学）　NGO VAN HE
学位授与の日付	平成 25 年　9 月 30 日
論　　文　　名	「APPLICATION OF CFD TO OPTIMIZATION OF HYDRODYNAMIC PERFORMANCES OF NON BALLAST SHIPS」 （ノンバラスト船の流体力学的性能最適化への CFD の応用）
論文審査委員	主査　　池田　良穂 副査　　馬場　信弘 副査　　深沢　塔一

Summary

For long time, model experiments in towing tanks have been playing important roles in developing innovative ship shapes. Because ship hydrodynamics are very complex due to waves generated on free surface between water and air, viscosity of water, vortex shedding flow and so on, very sophisticated experimental methods to grasp hydrodynamic performance of a ship have been established. The methods of model experiences, however, are not suitable to optimize ship shapes because a lot of models and huge numbers of tests are needed. To overcome the problem, for these thirty years, many researchers in Naval Architecture field have worked on developing theoretical methods, called as Computation of Fluid Dynamic (CFD), to solve the Navier Stokes Equation for a ship. In CFD, ship shapes can be easily changed and the hydrodynamic forces acting on many ship shapes can be calculated. Then optimization of ship shapes can be carried out faster and cheaper than model experiments which are expensive and consume a lot of time. Now a day, CFD becomes a popular and powerful tool for developing and designing ships. Many commercial codes of CFD are distributed, and some of them can be applied to calculate hydrodynamic forces acting on ships.

In this Thesis, a commercial CFD code “Fluent” version 14, which develops by ANSYS group, is applied to computation of hydrodynamic performance of Non Ballast Ships in calm water, in head waves and in head winds. Model experiments are indispensable to validate CFD results. CFD simulated results must be regularly tested by corresponding experiments because CFD sometimes gives wrong results under inappropriately calculating conditions. Therefore, some experiments to measure resistances acting on ships are carried out to validate the CFD results. The Practical Guideline for Ship CFD Applications updated in 2011 by the 26th International Towing Tank Conference (ITTC) was also referred for the tunings of the calculating conditions. After validated CFD results, the CFD code is applied to optimization

of bow shapes to reduce resistances acting on the ships in calm water and in regular head waves, and optimization of accommodation shapes to reduce interaction effect between hull and accommodation on air resistance acting on ship. Results of the study are presented in five chapters in this thesis including introduction chapter as follows.

In Chapter 2, validations of the CFD results of the flow around ship hulls and resistance acting on them in calm water and in regular head waves are carried out by comparison the results with the experimental one. As well known, CFD results depend on calculating fluid domain, number and shape of meshes and turbulent models. A lot of calculations of the CFD for various calculating conditions are carried out, and the results are carefully compared with the experimental results. As the results, appropriate conditions in the CFD are determined. The conclusions of this chapter are that accuracy of the CFD code is fairly good for the present purposes and that the CFD code could give us much information about pressure distributions, wave patterns and flow field in boundary layer as well as the total resistance acting on ship hulls.

In Chapter 3, the CFD is applied to optimization of the bow shapes in terms of the resistances in calm water and in regular head waves. In these applications of Non Ballast Water Ship (NBS) which was developed at Laboratory of Prof. Ikeda in Osaka Prefecture University is selected as an object ship. In this chapter, some series of bow shapes for NBS is systematically developed. Resistance acting on each hull is calculated and compared among hulls of the series to find the minimum resistance hull form. Computed results given by the CFD like resistance, pressure and wave pattern generated by ship movement both of in calm water and in waves are used for developing hull shapes in the optimizing processes. An optimal bow shape is found by comparison the results each other in series of bow shapes. In the computation in waves, the ship is in fully captured condition because shorter waves in $\lambda/L_{pp}<0.6$ are assumed here. From results of comparison among the CFD results of pressure distribution over hull surface, wave patterns and added resistance acting on the ships, the assessments of effect of bulbous bow on resistance acting on the ships in calm water and in regular head waves are carried out. The calculated resistances acting on the hull with the optimum bulbous bow are compared with the experimental results. Conclusions of this chapter are that series of bow shapes to reduce the resistance in waves as well as in calm water are developed and the optimum bow shape is found. The CFD results show that the optimum hull shape in calm water depends on Froude number although this is already common in the ship hydrodynamic field. Information of the CFD results plays an important role for understanding the phenomenon which causes the added resistance due to waves. The calculated results are in good agreement with the experimental one. The optimum hull shape can decrease the added resistance in regular head waves by 60% and total resistance by 15% in moderate short head waves (H_w=0.02m and $\lambda/L_{pp}<0.6$) at Froude number of 0.163.

In Chapter 4, the CFD is used to reveal the characteristics of the added resistance due to high waves. The effect of wave height on the added resistance due to waves is clarified. Three kinds of bow shapes of the ship which were developed at previous chapter, NBS-original without bulbous bow, improved bulbous bow shape NBK-N6 and optimum bow shape NBK-N5, added resistances acting on hulls of them are computed in higher regular head waves. The range of waves height is in 0.02~0.07m for the 2m model, and the ratio of wave length to ship length is smaller than 0.6 ($\lambda/L_{pp}<0.6$). By comparing the computed results of pressure, wave pattern and resistance among three ships, the best bow shape for ships in high waves is found. Measurements of resistances acting on the three ships in high waves are carried out and the experimental results are compared with those of CFD results. Conclusions of this chapter are that CFD code “Fluent” gives us fairly good results of resistances acting on hulls of the ships in regular head waves, with high waves, H_w>0.02m. The computed results like pressure distribution, profile of wave patterns are useful to understand the causes of reduction of added resistance in waves. The model NBK-N5 is confirmed to be the best one in this research. It could significantly reduce total resistance in shorter head waves, by up to 25% in high waves. Reduction of the total resistance of NBK-N5 in head waves is kept even in high waves. It was theoretically and experimentally confirmed that added resistance due to waves is not proportional to the square of wave height. The CFD can correctly show the dependency of the added resistance on wave height.

In Chapter 5, air resistance acting on ships is computed by the CFD to find the optimum shape above water surface with minimum air resistance. At first, air resistances acting on some kinds of accommodations of ships are investigated. By comparison among the calculated results of pressure distribution and air resistances acting on the accommodations, an optimum accommodation shape is found. And then, air resistance acting on a hull with an accommodation on its deck is computed. From the calculated results of air resistances acting on the hull, the accommodation and the hull with the accommodation on the deck, the interaction effects between hulls and accommodations are clarified. By comparison among the calculated results of air resistances each other, the best location of an accommodation on the deck and the best shape of accommodations with the smallest air resistance are found. The computed results suggest that interaction effects between the hull and the accommodation of a ship on its air resistance are important. Conclusions of this chapter are that air resistance acting on a box shape accommodation decreases with its slenderness because smaller frontal projected area of slender ones when floor area in the accommodation is assumed to be constant. The streamlined accommodation developed in this study can drastically reduce air resistance because of weaker separation of flow. Accommodations constructed by only flat plates with almost the same air resistance as that of the streamlined accommodation are developed. Interaction effects between a hull and an accommodation on air resistance increase the air resistance acting on whole ship by up to 30% compared with that of a ship with a box shape accommodation. The hydrodynamic interaction effect depends on location and shape of an accommodation. The accommodation located at bow makes the air resistance smaller. For the box shape accommodation the interaction effect increases the air resistance, but for the streamlined accommodation the effect decreases the air resistance.

In Chapter 6, the conclusions obtained in the present study are summarized.

審査結果の要旨

本論文は、大阪府立大学で開発されたバラスト水を積む必要のない画期的な船舶であるノンバラスト船を対象に、数値流体力学用のコンピュータ・コード(CFD コード)を利用して船型の最適化を行ったものであり、以下の成果を得ている。

(1) 本研究で使った CFD コードをノンバラスト船型等に適用し、静水中および波浪中において、十分な精度で計算できることを模型実験結果と比較することで確認した。
(2) 同 CFD コードを用いて、船首形状の最適化を行い、オリジナル船型に比較して静水中および波浪中での抵抗を大幅に低減できる形状を求め、実験によって同様の抵抗低減が得られることを確かめた。また、計算によって得られる圧力分布、造波波形などが、現象の理解および最適化のプロセスにたいへん有益であることを明らかにした。
(3) 理論計算が難しかった高い波高中での抵抗増加問題に、同 CFD コードを適用し、模型実験とよく合う結果が得られることを確認した。また、波浪中抵抗増加の波高影響特性についても明らかにした。さらに、波浪中での境界層内速度分布の変動の計算結果から、波浪中では伴流が弱くなることを明らかにした。
(4) 強風中の船舶の抵抗増加の一因となる風圧力を、同ＣＦＤコードを用いて計算をし、上部構造および船体に働く風圧力を求め、模型実験結果とよく合うことを確認した。また上部構造物と船体との相互干渉効果を明らかにした。

以上の成果は、数値流体力学の船舶の最適化問題への応用の実用性を示したものであり、船舶工学分野に貢献するところ大である。また、申請者が研究者として自立して活動できる能力と学識を有することを証するものである。