称号及び氏名	博士（工学） Nguyen Van Trieu	
学位授与の日付	2016 年 3 月 31 日	
論　　文　　名	「Development of a High Lift Performance Rudder with Wedge Tail」	
論文審査委員	主査	深沢 塔一
	副査	大塚 耕司
	副査	山崎 哲生
	副査	池田 良穂

論文要旨

Recently, as environmental standards for ships have become harder like EEDI, and ship speed became slower, securing of maneuverability of a ship in low speed operation becomes to be an important issue. As well known, marine rudders play the most important role in ship maneuvering and navigating. High lift performance rudders such as fishtail rudders and flap rudders have been widely used for smaller ships in order to improve their maneuvering and navigating in ports and docking. For larger ships such high performance rudders should be developed too. The rudders should be optimized in order not only to maneuver the ship safely, but also to improve the efficiency of her propulsion.

For such high performance rudders viscosity of fluid plays an important role in lift as well as drag forces acting on them. Therefore, for analyzing of hydrodynamic performance of the rudders and development of a higher performance rudder for a ship, Computational Fluid Dynamics (CFD) is mainly used. As well known, CFD has been rapidly developed and extensively used for many marine applications as well as ship design. The advantage of CFD is that it is faster and more economy than experimental methods. There are many CFD commercial codes that can be used for predicting various ship hydrodynamics.

In this study, a CFD commercial code, ANSYS FLUENT, is used to investigate the hydrodynamics performance of the high lift rudders such as fishtail and wedge tail rudders to understand the mechanism of increasing of the lift and drag forces. A smaller

rudder with the same lift performance as a conventional rudder is developed by using the CFD.

The contents of this thesis are as follows.

In Introduction, the research motivation, literature review, aims and objects of this thesis are presented.

In **Part** I (Chapter 1 ~ Chapter 4), the hydrodynamic characteristics of several types of high lift rudder sections or 2-D rudders are numerically investigated. These include: fishtail rudders, wedge tail rudders and a rudder with a flat plate vertically attached at the tail. The forces and moments acting on them are calculated by using the CFD code. At first, validation of the computed results is done by comparing the results with the experimental results. It is confirmed that the computed results are in fairly good agreement with the measured results. The numerical results show that all high performance rudder sections give higher lift at moderate angle of attacks and maximum lift than the conventional NACA 0024 rudder section. They accompany higher drag forces. However, small special tails can improve the lift while the drag does not increase significantly.

Chapter 1 shows the validation of the CFD calculations for a conventional foil section. Comparisons of the calculated and experimental results show that the CFD results of the present study are reliable.

Chapter 2 presents theoretical investigations on the performances of several kinds of fishtail rudder sections and hydrodynamic mechanism for generating higher lift forces due to the fishtails. The hydrodynamic forces acting on the sections and the flows passing them are calculated by using the CFD code. The effect of Reynolds number on the hydrodynamic forces are also investigated. Hydrodynamic characteristics of several kinds of rudder sections with different fishtails are computed to find the effects of the cross section shapes and the size of fishtails on them. The numerical results show that the lift and drag forces acting on the rudders are influenced by Reynolds number but reach an almost constant values at high Reynolds number above six million. Fishtails of a rudder increase the lift force and larger fishtails generate higher lift forces. Similarly, fishtails increase the drag force and the larger ones cause higher drag forces. From the higher lift and lower drag point of view, a smaller fishtail and a rudder section with medium maximum thickness give better hydrodynamic performance than others.

In Chapter 3, the effects of the wedge tail on the hydrodynamic characteristics of a wedge tail rudder are theoretically investigated by using the CFD code. The numerical results show that the wedge tail of a rudder increases the lift force acting on it significantly. The maximum lift increases by 62% in comparison with NACA 0024 section. Furthermore, the hydrodynamic characteristics of the wedge tail rudder are also compared with those of the fishtail ones which are investigated in Chapter 2.

In Chapter 4, the hydrodynamic characteristic of rudder sections with a small flat plate vertically attached at the trailing edge is investigated by using the CFD code. The hydrodynamic forces of the rudder section with the flat plate tail are also compared with those of the fishtail and wedge tail ones. The numerical results show that the flat plate can increase its lift force as well as fishtail and wedge tail rudders. The maximum lift depends on the size of the flat plate. However, the flat plate also accompanies slightly higher drag forces than other two rudders. The rudder section with a medium-size flat plate tail gives better hydrodynamic characteristic than the small and the large ones because it can generate the higher lift and smaller drag forces.

In Part II (Chapter 5 ~ Chapter 8), at first, the 3-D effects on rudder hydrodynamic performance are numerically investigated. Then, the top and bottom end plates have been used to reduce the 3-D effects and increase lift. The different rudders with wedge-tail, wedge-tail and end plates are designed to improve the lift in small angle of attack and the maximum lift. Finally, a rudder with 10-20% smaller chord length and a small wedge but has the same maximum lift as a conventional rudder has been developed. The experiments to measure the rudder forces and moment are carried out in the circulating water channel of OPU to validate the numerical results. The Reynolds number effects are also numerically investigated.

In Chapter 5, the effects of 3-D flow on marine rudders are numerically investigated by using the CFD code. The computed 2-D lift and drag coefficients are compared with the 3-D coefficients computed at the same conditions. Moreover, the end plates are fixed at the top and bottom of the rudder to reduce the 3-D effects and increase lift. The shapes of end plates are designed to optimize the hydrodynamic forces of the rudder. The conclusions of this chapter are that 3-D effects decrease the maximum lift and that the lift/drag ratio and delays the stall of the rudder. The end plate of a rudder increases the lift at small angle of attack and the maximum lift.

In Chapter 6, higher lift rudders with a wedge tail and end plates are developed by using the CFD code. The numerical results show that a small wedge tail of 2.5% chord length increases the maximum lift force by 16%. Moreover, the small wedge tail and two end plates attached at top and bottom of the rudder give 25% higher maximum lift than that of the conventional semi-balanced rudder. Increasing of the drag force due to the wedge tail and end plates is numerically investigated and discussed.

In Chapter 7, a high lift rudder with a wedge tail and an end plate has been developed by using the CFD method. A spade rudder is selected as the based rudder model. The developed rudders with almost same lift force as the spade rudder have shorter chord length by 10-20% than the based one. The breadth of the tail is changed from 2.5 to 5.0% of chord length, and an end plate is attached on the bottom of the developed rudders to increase the lift force. The numerical results show that the rudder with a 20% shorter chord length, a width of wedge tail 2.5% of chord length and an end plate (0.8c-wedge2.5-endplate) gives the highest maximum lift force. Likewise, a rudder with 10% shorter chord length, a width of wedge tail of 2.5% of chord length and without an end plate (0.9c-wedge2.5) also produces higher lift than the based rudder. However, all developed rudders accompany higher drag forces than the based rudder at small angles of attack. At zero angle of attack, the drags increase by 15%, 18% and 22% for the 0.9c-wedge2.5, 0.8c-wedge2.5-endplate, and 0.8c-wedge5.0 rudder models, respectively.

In Chapter 8, smaller rudders with the same performance as a conventional one are developed on the basis of the obtained knowledge on fundamental characteristics of a wedge rudder. Rudders with 10-20% shorter chord lengths are designed, and the lift, drag and moment calculated by the CFD code are compared with experimental ones. Numerical investigations on Reynolds number effect on the rudders demonstrate that smaller models gives lower lift and higher drag than the real scale rudder.

In Part III (Chapter 9 & Chapter 10), firstly, the hydrodynamic characteristics of a Wageningen B screw series in open water are calculated to confirm the accuracy the CFD results. Then, the hydrodynamic forces acting on a wedge-tail rudder in propeller flow are calculated to investigate the effects of propeller flow on the hydrodynamic characteristics of the wedge-tail rudder.

In Chapter 9, a simpler way to calculate the flow created by a propeller and the generated forces acting on it by using the CFD code is developed and validated for the open-water characteristic. For calculations Wageningen B screw series, which is the most extensive and widely used propeller series, are selected. The results of hydrodynamic performance of the propeller are in fairly good agreement with experimental results, and the developed way in the study is confirmed to be able to be safely used for calculations of interacting flow between a propeller and rudders behind it.

In Chapter 10, hydrodynamic performances of a conventional rudder and a wedge-tail rudder behind a propeller are numerically investigated. The numerical results show that the lift and drag forces acting on the rudders, at moderate and high angle of attack increase as the propeller loading increases. However, at small angle of attack below five degrees and heavy propeller load, the drag acting on the rudders becomes negative or the rudders generate thrusts. It is confirmed that the negative drag is caused by propeller rotating flow.

In Conclusion, all outstanding conclusions obtained in the present study are gathered up.

審査結果の要旨

本論文は、船舶の操縦性および安全性向上のために用いられる、従来の舵よりも高い揚力を発生できる特殊舵の1つであるウェッジ付舵について、数値流体力学(CFD)を使って流体力学的な特性把握と最適設計手法の開発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、通常舵とウェッジ付舵に適用した結果を、実験を行って得られた値と比較し、同コードによって十分な精度で計算できることを確認した。
(2)特殊舵であるフィッシュテイル舵、垂直平板付舵、ウェッジ付舵が発生する流体力をCFDで計算し、高い揚力が得られるメカニズムを明らかにした。また、同時に発生する抗力の特性についても明らかにした。
(3)ウェッジ付舵に働く流体力が、強いレイノルズ数影響を受けることをCFD計算で明らかにし、舵の小型模型による水槽実験では、最大揚力が小さく、抗力が大きくなることを示した。
(4)コード長を10〜20%短くしても、ほぼ同じ揚力を得られる小型の高性能ウェッジ付舵を、CFDを援用して開発する手法を示した。
(5)スクリュープロペラの後方に置かれたウェッジ付舵まわりの流れを計算し、プロペラによる加速流によって揚力が増加すると同時に、プロペラ負荷が大きい時には回転流の影響で舵抵抗が負となることを示し、その流体力学的メカニズムを明らかにした。

以上の成果は、粘性影響が大きい船舶の高揚力舵の最適化に数値流体力学を応用すること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。