Performance Analysis of a Dolphin-tail Rudder

***K. S. Min* [1], *K. N. Chung* [2], *Y. I. Kim* [3]**
1. HHI, #1, Cheonhadong, Donggu, Ulsan, Korea, minks@hhi.co.kr
2. HHI, #1, Cheonhadong, Donggu, Ulsan, Korea, knchung@hhi.co.kr
3. HHI, #1, Cheonhadong, Donggu, Ulsan, Korea, daroo@hhi.co.kr

Corresponding author ***K. N. Chung***

Abstract

As a part of numerical and experimental research works for the prediction and improvement of ship's maneuvering performance, a study on the performance analysis of two different rudders has been carried out. While the planform shape and the aspect ratio of the rudders have been fixed, section shape has been changed. Conventional type of HMRI NP section and special type of dolphin-tail section have been employed. Performances of the rudders have been investigated by using CFD and compared with experimental data obtained in a wind tunnel. A commercial CFD program has been used to solve the RANS equations. Two-equation k-ω model has been applied to close the governing equations. Block-structured grids are used in the numerical calculation.

Based upon the calculation results, the rudder with dolphin-tail section has shown a possibility of significantly improving rudder performance if utilized as the section of ship rudders.

Keyword: *rudder, dolphin-tail section, CFD, k-ω model, block-structured grid*

1. Introduction

Recent accidents of ship collision and subsidence increase international interest on the controlablity of a ship such that IMO(International Maritime Organization) has suggested a standard of IMO Guideline 751. This guideline will come into effect from January of 2004. There are two kinds of control devices in ships. For example, hull form is classified as one of passive control devices, and a rudder is an active control device. Especially, rudder is the unique active control device for large ships. Thus, rudder design is important as well as hull form design to improve the controliblity of a ship.

Performance of a rudder is determined by geometrical properties and rudder area. Here, geometrical properties represent section shape, plan form and aspect ratio [1]. However, rudder area is limited since rudder is placed behind a propeller. Sometimes large area reduces the controliblity. So that rudder design should be performed to have best performance with a certain fixed area.

In the present study, the section shape of a rudder has been changed while the planform and the aspect ratio of the rudder have been fixed. Conventional type of HMRI-NP section[1] and special type of dolphin-tail(DT) section have been used. Performance of the rudders have been predicted using by computational fluid dynamics(CFD), and calculation results have been compared to the experimental data obtained in a wind tunnel.

2. Numerical Method

In this study the whole flow field is assumed to be turbulent, thus we can say that the governing equation is the RANS(Reynolds-averaged Navier-Stokes) equations. TASCflow [2] has been used to solve these governing equations. This CFD program has the following characteristics. In the program, the transport equations are discretized using a finite volume method. The momentum equations and the continuity equation are solved in a coupled manner. The system of discretized algebraic equations is solved by incomplete LU factorization method, and an algebraic multi-grid method is employed for rapid convergence. Modified linear profile skew-upwind scheme with physical advection correction

Paper No. : 2-1D-4

has been used to discretize convection terms of the RANS equations. Two-equations of k-ω model of Wilcox or standard k-ε model is used as the turbulence model to close the governing equations.

3. Rudder Geometry and Computational Grids

3.1 Geometry of the rudders

Fig. 1 shows the schematic view of the planform of the rudders. In this study, forward trapezoidal form is employed. Detail dimensions are denoted in the figure, and the length unit is centimeter. Aspect ratio of the rudders is about 1.70. Here, aspect ratio is defined as the ratio of the span length and the chord length of the mid span. In the figure, arrow represents flow direction. Thus, left-hand side of the figure represents upstream. Fig. 2 depicts the section shape of the rudders.

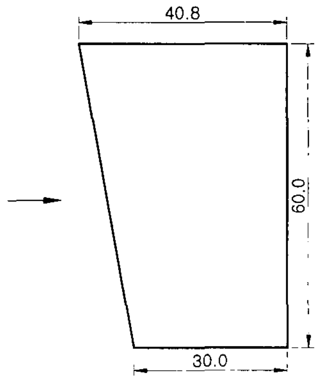

Fig. 1. Schematic view of the planform

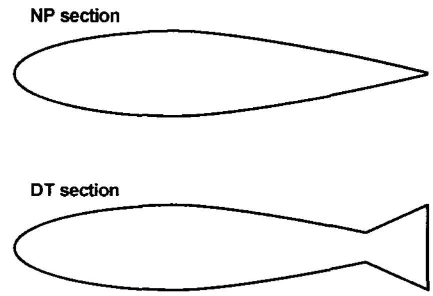

Fig. 2. Section shape of the rudders

3.2 Computational grids

An elliptic grid generation method [3] has been used to obtain smooth computational grids. Single-block grids have been used to the 2-D foils of NP and DT sections. Computational grids of the rudders consist of three blocks, that is, main block, top block and bottom block. Schematic view of a computational grid for the DT rudder is shown in Fig. 3. In the figure, parts of the surface grid are depicted. The total number of grid points is about 680,000.

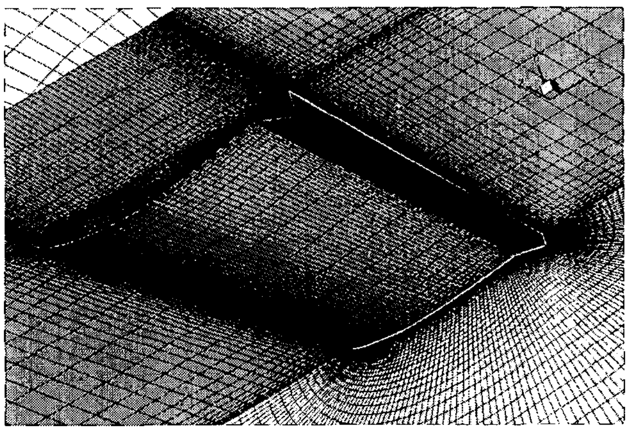

Fig.3. Computational grid of the DT Rudder

4. Calculation Results

4.1 2-D foil section with camber

Prior to rudder flow calculations, a 2-D foil with camber has been considered as a first step of a series of calculations. Accuracy of the present calculation method is evaluated by this problem. This foil section has the conventional type of NP foil section with 4 % camber. Angle of attack has varied from

–4 degrees to 4 degrees as in the experiment. Fig. 4 shows computed surface pressure distributions at zero and four degrees angle of attack. In the figures, α means angle of attack. The solid line denotes calculation result of the CFD method, and circles represent experimental data. Results of a panel method [4] are also given and denoted by the dashed line. Compared to the experimental data, satisfactory results are obtained by the CFD method.

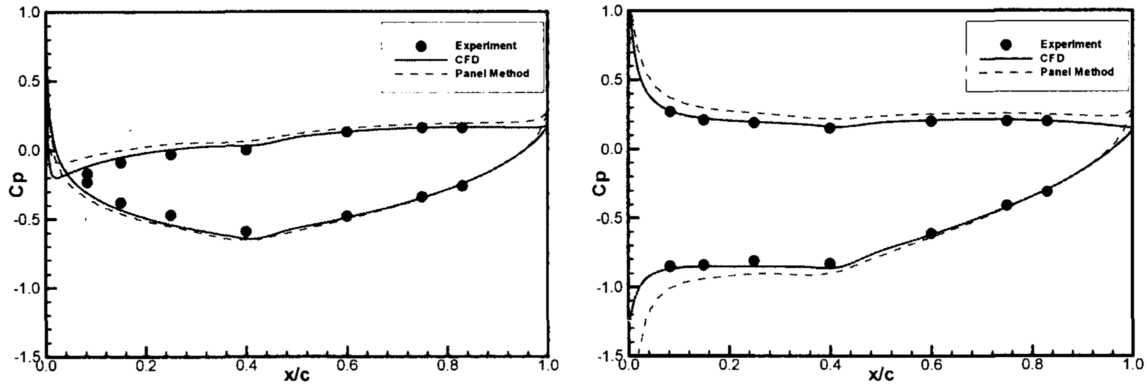

Fig. 4. Surface pressure distributions for a 2-D cambered foil

4.2 2-D rudder sections

As a next step, flow fields of 2-D NP section and 2-D DT section are solved from 0 to 16 degrees of angle of attack. In this case, cambers of the sections are zero. Steady solutions are not converged from 20 degrees of angle of attack. Oscillations of residuals are captured from this large angle of attack. Thus, unsteady calculations have conducted above this angle of attack. In order to know the effect of Reynolds number(Re), the inflow velocity has been changed from 1 m/s to 5 m/s. Such that two Reynolds numbers are considered in the present study. The corresponding values of Re are 1 x 10^6 and 5 x 10^6, respectively. Fig. 5 depicts calculated lift coefficients for the 2-D sections.

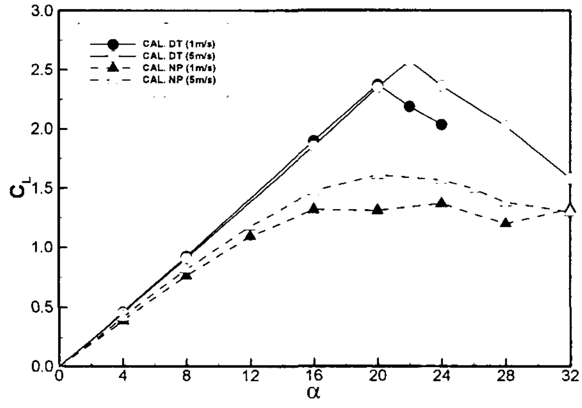

Fig. 5. Lift coefficient of the 2-D sections

5. Conclusions

In the present study, performance of two different rudders has been studied by using CFD method. While the planform shape and the aspect ratio of the rudders have been fixed, section shape has been changed. Although three-dimensional rudder flow calculations do not predict accurate rudder performance, two-dimensional calculations give some useful information. Compared to the conventional type of NP rudder, special type of DT rudder has shown larger lift coefficient, maximum lift coefficient and stall angle. Delay of stall with Reynolds number is also confirmed in the DT rudder. For large ships, these features can improve ship's maneuvering performance. Thus, we can say that the rudder with dolphin-tail section has a possibility of improving rudder performance.