Performance analysis of a horn-type rudder implementing the Coanda effect

Dae-Won Seo\textsuperscript{a}, Jungkeun Oh\textsuperscript{b}, Jinho Jang\textsuperscript{c,}\textsuperscript{*}

\textsuperscript{a} Technical Division, Korean Register, Busan, South Korea
\textsuperscript{b} Department of Naval Architecture and Ocean Engineering, Kunsan National University, Kunsan, South Korea
\textsuperscript{c} Central Research Institute, Samsung Heavy Industries Co., Ltd., South Korea

Received 28 July 2016; revised 2 September 2016; accepted 8 September 2016

Available online ▪▪▪

Abstract

The Coanda effect is the phenomenon of a fluid jet to stay attached to a curved surface; when a jet stream is applied tangentially to a convex surface, lift force is generated by increase in the circulation. The Coanda effect has great potential to be applied practically to marine hydrodynamics where various lifting surfaces are being widely used to control the behavior of ships and offshore structures. In the present study, Numerical simulations and corresponding experiments were performed to ascertain the applicability of the Coanda effect to a horn-type rudder. It was found that the Coanda jet increases the lift coefficient of the rudder by as much as 52\% at a jet momentum coefficient of 0.1 and rudder angle of 10°.

Copyright © 2016 Society of Naval Architects of Korea. Production and hosting by Elsevier B.V. This is an open access article under the CC BY-NC-ND license (http://creativecommons.org/licenses/by-nc-nd/4.0/).

Keywords: Coanda effect; Coanda jet; Horn-type rudder; Coanda rudder

1. Introduction

Slow-speed ships having full hull form such as oil tankers sometimes have difficulty in getting sufficient lift force for securing maneuverability with a rudder. Particularly, even though the rudder angle increases when a ship sails at low speed, insufficient lift force of the rudder due to a low inflow velocity causes a lack of maneuverability, which may lead to a dangerous marine accident.

Accordingly, since IMO (International Maritime Organization) raised issues about the maneuverability of ships in order to prevent marine pollution due to the accidents of crude oil carriers, studies to develop high-lift rudders have been actively conducted (Choi and Kim, 2004; Hasegawa et al., 2006; Kim et al., 2012a,b; Nagarajan et al., 2008).

The Coanda effect is known as an effective way to generate the high lift force when applied to air foils. This is the phenomena in which a jet flow attaches itself to a nearby surface and remains attached even when the surface curves away from the initial jet direction (Fig. 1). Hence, it is thought that the Coanda effect may have practical applications in the field of marine engineering as well since ships and other mobile units that operate in marine environments exploit hydrodynamic lift in many ways (Ahn and Kim, 1999, 2003; Berman, 1985; Chau et al., 2005; Kim et al., 2012a,b).

The sensitivity of the grid spacing to numerical solutions for Coanda foils and the influence of turbulence models have been verified and compared with existing experimental results on flow fields around two-dimensional elliptic foils (Shrewsbury, 1985; Linton, 1994; Jung et al., 2012).

In the present study, the Coanda effect was applied to a horn-type rudder to enhance the maneuvering performance during low-speed operation. Numerical simulations have been performed to investigate the characteristics of the boundary layer and change in the circulation on a two-dimensional section of the rudder with the Coanda system. In addition, change in the lift force was examined according to various jet momentums and angles of attack. A mechanism for jet phenomena...
injection was prepared inside the horn and a partial opening was placed to inject jet flow tangentially along the suction side, regardless of the rudder angle. Experiments and numerical simulations for the flow fields around a Coanda rudder were carried out to investigate the effectiveness of the Coanda devices in terms of enhancing the rudder performance over a practical range of the jet momentum and rudder angle.

2. Numerical simulation

The Reynolds averaging approach for turbulence modeling is applied. The continuity equations and the Navier–Stokes equations can be written in Cartesian tensor form as:

\[ \frac{\partial \rho}{\partial t} + \frac{\partial}{\partial x_i} (\rho u_i) = 0 \]  

\[ \frac{\partial \rho u_i}{\partial t} + \frac{\partial}{\partial x_j} (\rho u_i u_j) = - \frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j} \left[ \mu \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} - \frac{2}{3} \delta_{ij} \frac{\partial u_k}{\partial x_k} \right) \right] 
+ \frac{\partial}{\partial x_j} \left( - \rho \overline{u_i' u_j'} \right) \]  

where \( x_i \) are Cartesian coordinates, \( u_i \) are the corresponding velocity components, \( p \) is pressure, \( \rho \) is the density, \( \mu \) is the viscosity and \( -\rho u_i' u_j' \) represents the Reynolds stresses. These Reynolds stresses must be modeled in order to close Eq. (2), i.e., for solving the turbulence closure problem.

The \( k-\omega \) model is one of the most widely used turbulence models for external aero- and hydro-dynamics. Reynolds stress turbulence model (hereafter, RSTM) is the most advanced turbulence model for engineering applications and has shown better potential than other models to predict the key features of the present flow. The detailed implementation of the models in the present CFD code is described in Kim (2001) and Kim and Rhee (2002).

A numerical study has been carried out using FLUENT, a general-purpose commercial software. The numerical approach used in the present study employs a cell-centered finite-volume method along with a linear reconstruction scheme that allows the use of computational cells with arbitrary polyhedral shapes. The convection terms are discretized by a second-order accurate upwind scheme, while the diffusion terms are discretized by a second-order accurate central differencing scheme. For transient flow calculations, the time-derivative terms are discretized by a first-order backward implicit scheme. The velocity-pressure coupling and overall solution procedure are based on a SIMPLE-type segregated algorithm that is adapted to an unstructured grid. The discretized equations are solved using point-wise Gauss-Seidel iterations, while the algebraic multi-grid method accelerates solution convergence. The relaxation factors are set to 0.3 for the pressure, 0.5 for the momentum, and 0.5 for the turbulence.

While the computational scheme is important when performing numerical simulations, the selection of an appropriate turbulence model and grid composition for computational quality has a greater effect on the results of analysis (Slomski et al., 2002; Wilcox, 1993). In a previous study (Seo et al., 2008a,b; Park and Lee, 2000), to examine the influence of the turbulence model and grid dependency, calculations were performed for various cases. It was found that the solutions under the RSTM with the minimum \( y^+ \) being less than 1 were more consistent with the experimental results than those with the \( k-\omega \) or \( k-\varepsilon \) turbulence model (Pulliam et al., 1985; Rhee et al., 2003). Hence, in the present calculations, \( y^+ \) less than 1 is used for the RSTM model. In addition, Seo (2011) has examined the influence of Reynolds effect by CFD analysis, the results show that CFD analysis should be carried out in more than Reynolds number of \( 5 \times 10^5 \).

The computational domain is of the H–H type with ranges, \(-4 \leq X/C \leq 5 \) and \(-4 \leq Y/C \leq 4 \), where \( C \) means the chord length of the rudder section. The computational mesh is shown in Fig. 2. It consists of 97,000 quadrilateral cells, and the first cell spacing \( (y^+) \) off the solid surface is approximately one in terms of the wall.

For the comparison with the experimental results, three-dimensional numerical simulations were carried out with following conditions. The composition of the computational grids for the three-dimensional rudder is also generated using Gridgen Ver. 15.08. The grid is a C–H type. The number of surface grids equals 126 \( \times \) 76 = 9576 on each side of the rudder surface; the number of grids across the gap between the horn and the rudder surface is taken as 20, as shown in Fig. 3. The computational domain was extended to \(-4 \leq X/C \leq \pm 5 \), \(-4 \leq Y/C \leq \pm 4 \), and \( 0 \leq Z/C \leq \pm 4 \), and the total number of grids used in the computation was about one million points. The \( k-\omega \) turbulence model was employed in the present computations since the RSTM consumes excessive computational resources (Seo et al., 2008a).

The lift, drag, and jet momentum coefficients defined as:

\[ C_L = \frac{L}{\frac{1}{2} \rho V_\infty^2 C S} \quad C_D = \frac{D}{\frac{1}{2} \rho V_\infty^2 C S} \quad C_j = \frac{m V_{jet}}{\frac{1}{2} \rho V_\infty^2 C S} \]  

Here, \( L \) and \( D \) is lift and drag force, \( m \) refers to the mass flux rate \((\text{kg/s})\), \( V_{jet} \) refers to the average speed of flow that ejects through the slit, and \( \rho \) and \( V_\infty \) refer to the density and speed of the incoming flow. \( C \) and \( S \) is chord and span length respectively.

The jet flow supplied to the foil was varied in the range of \( 0 < C_j < 0.4 \), and the characteristics of the boundary layer and
Fig. 2. Grid system of the Coanda rudder section.

Fig. 3. Grid system of the horn-type rudder applied the Coanda system.

Fig. 4. Geometry of the Coanda foil section for numerical simulation.
circulation were investigated when the length of the flap and height of the slit and gap were fixed at 0.6C and 0.005C, respectively (Fig. 4).

Numerical simulation was conducted for various angles of attack and jet momentums. The results are shown in Fig. 5; the lift force increased in all cases. Especially, the rate of increase in the lift in the range of $C_j < 0.07$ was greater than for $C_j > 0.07$.

The lift performance mostly increased in the vicinity of 0.05 for $C_j$. But in the case of $C_j > 0.05$, the rate of increase in the lift was relatively small. The lift coefficients of $C_j = 0.07$ and 0.1 enhanced the lift by 145% and 208%, respectively, compared to the case of $C_j = 0$, when the angle of attack was 10.

In addition, the lift performance under a rudder angle of 20° and $C_j$ of 0.1 and 0.15 also increased by about 110% and 160%, respectively. The drag force decreased while the jet flow was supplied especially in the range of $C_j < 0.05$ and $\alpha = 20°$.

Figs. 6 and 7 show the streamlines near the leading edge and velocity vectors near the trailing edge for various jet momentums and angles of attack. It was found that the separation near the trailing edge was delayed as supply jet momentum. The entrance angle of flow to the leading edge increases according to the increase in circulation around the foil. Accordingly, the circulation causes a stagnation point around the leading edge is pushed toward the pressure side, and the lift performance increases. Therefore, the separation point around the leading edge becomes a standard for judging the strength as the circulation increases.

Judging from the streamlines around the leading edge in Figs. 6 and 7, the stagnation point is seen to be pushed toward the pressure side according to the supply of the water jet; further, the stagnation point is regularly pushed toward the pressure side beyond $C_j = 0.05$, where separation ceases. Such a trend is apparently seen at $\alpha = 20°$, and the stagnation point is pushed regularly at the degree in comparison with 10° beyond $C_j = 0.05$ (where separation ceases). This is because the strength of the circulation around the foil increases when the water jet is supplied, when the rudder angle is 20°.

Moreover, judging from the vectors around the leading edge in Fig. 6, an adverse pressure gradient is seen to originate at the position of $X/C = 0.3$ when the angle of attack is 10° and $C_j = 0.0$; furthermore, massive separated flows occur behind the position of $X/C = 0.3$. However, in the case of $C_j = 0.02$, the separation is seen to be pushed by a trailing edge of the foil, and in the case of $C_j = 0.05$, the separation is then pushed backward and disappears over the surface of the foil.

Then, it is seen that the speed profile becomes considerably faster inside the boundary layer due to the Coanda jet in
Fig. 7. Computational results of streamlines near the leading edge and tailing edge at $\alpha = 20^\circ$ and $Re = 7 \times 10^5$.

(a) $C_f=0.00$  
(b) $C_f=0.02$  
(c) $C_f=0.05$  
(d) $C_f=0.10$

Fig. 8. Profile and arrangement of a horn-type rudder.

Fig. 9. Arrangement of a Coanda device.
trailing edge, in case $C_j = 0.1$. Such a trend is more easily shown in Fig. 7 when the angle of attack is $20^\circ$. When the angle of attack is $20^\circ$ and $C_j = 0.0$ (Fig. 7(a)), the separation point starts from $X/C = 0.15$ but in case $C_j = 0.05$ (Fig. 7(c)), the separation point is extended to the trailing edge and the stall is delayed. Accordingly, it is possible to delay the separation through Coanda devices by using the water jet and thus control the stall effectively.

3. Experimental setup

Based on the two-dimensional numerical simulation results shown in the previous section, a Coanda system was applied to the three-dimensional horn-type rudder of a 47K Product Carrier. Then, experiments and computations were carried out to investigate the enhanced performance under the Coanda rudder.

Fig. 8 shows a profile of the horn-type rudder of a 47K PC. The average length of chord and span are 261 mm and 387 mm, respectively. The vertical gap clearance of the sections is 2.3 mm, and the horizontal gaps at the top and bottom of the pintle block are also 2.3 mm.

The height of the nozzle for injecting the Coanda jet on the trailing edge of the horn is 0.0019C (0.5 mm), and the span of the Coanda device is 135 mm, as shown in Fig. 9 and Table 1.

For the experiment, a 1/21.4-scaled UV-resin model of the 47K PC rudder was prepared as shown in Fig. 10. The tests of the model were conducted in a circulating water channel, and the hydrodynamic forces acting on the horn-type rudder with and without jet injection were measured by a three-component load cell. Free surface-suppressing plates were also installed to remove the free-surface effects.

To minimize the fluctuation in the jet pressure and maintain stable injection, an accumulation tank filled with compressed nitrogen gas acting as a damper as shown in Fig. 11 was used. The jet injection pressure was kept constant by adjusting the pressure regulator, and the jet velocity was estimated by taking the mean of the flow rates at the nozzle section. In the system,
a high-pressure pump automatically supplied pressurized water to compensate the pressure drop.

4. Results and discussion

Fig. 12 shows the change in lift performance through experimentation and computations at angles of 10° and 20° when the same amount of jet flow is supplied. The lift increased while injecting the jet momentum as per the results of two-dimensional computations. The slope of the lift-performance curve changes most significantly near $C_j = 0.05$. The lift performance for a rudder angle of 10° and $C_j$ of 0.1 and 0.15 increased by about 52% and 60%, respectively. In addition, these trends were clearly seen in the numerical results.

Fig. 13 shows the experimental results regarding the lift and drag performance of the horn-type rudder with and without the Coanda jet. The lift force was greatly increased and the drag force reduced by supplying the jet momentum but the stall phenomenon was not delayed unlike the results of two-dimensional computations as shown in Fig. 5. In order to investigate why the stall of the horn-type rudder under the Coanda effect was not delayed, experiments in terms of flow visualization and three-dimensional computations were conducted.

As the angle of attack is small, the jet flow that is exhausted through the slit comes out well and regularly in the spanwise direction. However, when the angle of attack is 20° or more, the Coanda jet flow is distorted by the vortex that is generated at the leading edge of the pintle portion is rolled up in the direction of the arrow due to the intersection of the fixed horn and the rudder plate and the flow that is drawn through a horizontal gap. Moreover, the Coanda jet is seen not to flow along the rudder surface due to such a vortex; thus, the rate of increase in lift is reduced (Fig. 14).

5. Conclusions

The lift performance of a two-dimensional NACA wing section under the Coanda effect was numerically studied to investigate the characteristics of the boundary layer and circulation.

Experimental and numerical studies were performed to observe the effectiveness of the Coanda device in terms of enhancing the rudder performance over a practical range of the jet momentum and rudder angle.
The slope of the lift performance changed greatly near $C_j = 0.05$. The rate of increase in the lift was higher when $C_j$ was less than 0.05 and lower when $C_j$ exceeded 0.05.

For rudder angles of 10° and 20°, the lift performance of a horn-type rudder section under the Coanda system improved by 208% and 160%, respectively, compared to that of a rudder without the Coanda system.

The experimental results on the horn-type rudder in the presence of the Coanda effect showed that the lift increased by 52% and 32%, respectively, when the angles of attack were 10° and 20° at $C_j = 0.1$.

The rate of increase in the lift for the horn-type rudder with the Coanda system was lower than the corresponding results of two-dimensional computations due to the vortex that was generated on the upper pintle. Hence, the Coanda jet could not flow along the rudder surface due to such a vortex. To overcome this problem, an improved Coanda rudder will be studied soon after.

Acknowledgements

This work was supported by the National Research Foundation of Korea (NRF) Grant funded by the Korean Government (MOE) (NRF-2014R1A1A2059291).

References


+ MODEL

Please cite this article in press as: Seo, D.-W., et al., Performance analysis of a horn-type rudder implementing the Coanda effect, International Journal of Naval Architecture and Ocean Engineering (2016), http://dx.doi.org/10.1016/j.ijnaoe.2016.09.003