Available online at www.sciencedirect.com
ScienceDirect
Publishing Services by Elsevier
International Journal of Naval Architecture and Ocean Engineering xx (2016) 1–8
http://www.journals.elsevier.com/international-journal-of-naval-architecture-and-ocean-engineering/

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

Dae-Won Seo [a], Jungkeun Oh [b], Jinho Jang [c,*]

[a] *Technical Division, Korean Register, Busan, South Korea*
[b] *Department of Naval Architecture and Ocean Engineering, Kunsan National University, Kunsan, South Korea*
[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 applied 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°.


*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

* Corresponding author.
*E-mail address:* sbeart@naver.com (J. Jang).
Peer review under responsibility of Society of Naval Architects of Korea.

http://dx.doi.org/10.1016/j.ijnaoe.2016.09.003

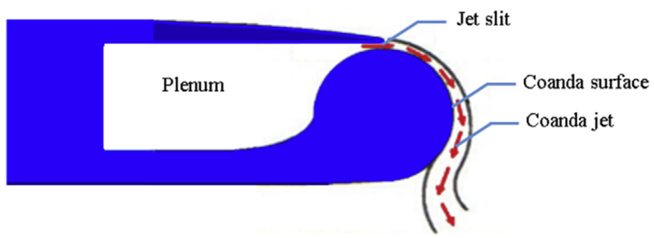


Fig. 1. Basic concept of the Coanda effect.

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:

$$\frac{\partial \rho}{\partial t} + \frac{\partial}{\partial x_i}(\rho u_i) = 0 \tag{1}$$

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

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 $-\rho\overline{u_i' u_j'}$ 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, $-4 \leqq X/C \leqq 5$ and $-4 \leqq Y/C \leqq 4$, 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 $-4 \leqq X/C \leqq 5$, $-4 \leqq Y/C \leqq 4$, and $0 \leqq Z/C \leqq 4$, and the total number of grids used in the computation was about one million points. The k-ω 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:

$$C_L = \frac{L}{\frac{1}{2}\rho V_\infty^2 CS},\quad C_D = \frac{D}{\frac{1}{2}\rho V_\infty^2 CS},\quad C_j = \frac{\dot{m}V_{jet}}{\frac{1}{2}\rho V_\infty^2 CS} \tag{3}$$

Here, L and D is lift and drag force, $\dot{m}$ refers to the mass flux rate ($kg/s$), $V_{jet}$ 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