Available online at www.sciencedirect.com
ScienceDirect
Publishing Services by Elsevier
International Journal of Naval Architecture and Ocean Engineering xx (2017) 1—16
http://www.journals.elsevier.com/international-journal-of-naval-architecture-and-ocean-engineering/

Numerical simulation of unsteady propeller/rudder interaction

Lei He [a], Spyros A. Kinnas [b,*]

[a] *MODEC International, Inc., United States*
[b] *Ocean Engineering Group, ECJ8.604, The University of Texas at Austin, Austin, TX 78712, United States*

Received 19 May 2016; revised 14 February 2017; accepted 16 February 2017
Available online ■ ■ ■

Abstract

A numerical approach based on a potential flow method is developed to simulate the unsteady interaction between propeller and rudder. In this approach, a panel method is used to solve the flow around the rudder and a vortex lattice method is used to solve the flow around the propeller, respectively. An iterative procedure is adopted to solve the interaction between propeller and rudder. The effects of one component on the other are evaluated by using induced velocities due to the other component at every time step. A fully unsteady wake alignment algorithm is implemented into the vortex lattice method to simulate the unsteady propeller flow. The Rosenhead-Moore core model is employed during the wake alignment procedure to avoid the singularities and instability. The Lamb-Oseen vortex model is adopted in the present method to decay the vortex strength around the rudder and to eliminate unrealistically high induced velocity. The present methods are applied to predict the performance of a cavitating horn-type rudder in the presence of a 6-bladed propeller. The predicted cavity patterns compare well with those observed from the experiments.

1. Introduction

As the main device of controlling the ship's manoeuvring performance, the rudder is normally placed at the ship's stern behind the propeller to utilize the rotational energy from propeller and increase the propulsive efficiency. With an increasing demand of high speed, large capacity vessels in both navy and commercial applications, the driving power of those vessels has persistently increased in recent decades. The hydrodynamic loading upon propellers and the unsteadiness of its induced wake flow are also increased. In consequence, the interaction between operating propeller and rudder has become increasingly crucial for both propeller and rudder design and analysis.

There has been a considerable amount of investigation into the interaction between propeller and rudder in the past. Lewis (1973) conducted tests to measure the propeller excited forces on rudder. Minson (1974) investigated the phase angle between the impingement of propeller tip vortex and the vibratory forces on a spade rudder. Stierman (1989) conducted systematic tests with various propeller/rudder configurations to evaluate the influence of the rudder on propulsive performance.

Turnock (1993) proposed a panel method for calculation of interaction effects of a propeller and rudder. The theory is based on a lifting surface method where both rudder and propeller geometries are modeled. The interaction between rudder and propeller is accounted for by modification of the inflow fields for the two components. Fujino (1996) discusses different models based on the panel method for calculation of the interaction between rudder, propeller and hull. Li (1996) investigated the propeller-rudder interactions by applying a lifting line theory for the propeller and a vortex lattice method for the rudder. The deformation of slipstream from propeller was taken into account by satisfying the boundary conditions. Han et al. (2001) used a surface panel method to solve the flow around a horn-type rudder and a vortex lattice method to solve the flow around the propeller. The three-dimensional flow around the rudder and the propeller was computed

* Corresponding author.
E-mail address: kinnas@mail.utexas.edu (S.A. Kinnas).
Peer review under responsibility of Society of Naval Architects of Korea.

http://dx.doi.org/10.1016/j.ijnaoe.2017.02.004

simultaneously. The calculated results were compared with the measurements from other experiments. Szantyr (2007) investigated the hydrodynamic interaction between operating propeller and rudder by using a program based on the unsteady lifting surface model representing the propeller and on a boundary element method representing the rudder. Kinnas et al. (2007) predicted sheet cavitation on a rudder in a propeller slipstream, including the effects of propeller as well as of tunnel walls. In their work, a vortex lattice method coupled with a finite volume method representing propeller and wall effect and a boundary element method representing the rudder were applied to predict time-averaged sheet cavitation on rudders. The interactions between propeller, rudder and tunnel wall are included via effective velocity predicted by a finite volume method. However, in those methods the unsteadiness caused by propeller shed wake and its impingement with rudder surface are ignored.

In this paper, a numerical approach based on a panel method and a vortex lattice method for the unsteady propeller/rudder interaction is presented. The developed methods allow simulating the propeller-rudder interaction in a fully unsteady way. The interaction between propeller wake and rudder surface is desingularized by introducing a viscous vortex core model. Furthermore the propeller shed vorticity is eliminated through this vortex core model in a more natural way. Numerical results using the current method are presented and compared with experiments.

2. Formulation for flow around rudder

In this paper, the performance prediction of wetted and cavitating rudders subjected to unsteady propeller flow is performed by using a 3-D potential based panel method.

2.1. Governing equation

Assuming the flow is incompressible, inviscid and irrotational, the total velocity $\vec{q}(x,y,z,t)$ at any point in the fluid domain can be expressed as follows:

$$\vec{q}(x,y,z,t) = \vec{q}_{in}(x,y,z,t) + \nabla\phi(x,y,z,t) \quad (1)$$

where $\vec{q}_{in}$ is the inflow wake respect to the rudder surface.

The perturbation potential, $\phi(x,y,z,t)$, satisfies the Laplace's equation in the fluid domain.

$$\nabla^2\phi(x,y,z,t) = 0 \quad (2)$$

At every point $p(x,y,z)$ on the wetted rudder surface, S_{Rw}, or on the cavitating surface S_C, the perturbation potential, $\phi(x,y,z,t)$, must satisfy Green's third identity:

$$2\pi\phi = \int\int_{S_{RW}\cup S_C} \left[\phi_q \frac{\partial G(p;q)}{\partial n_q} - G(p;q)\frac{\partial \phi_q}{\partial n_q}\right] ds + \int\int_{S_W} \Delta\phi_W \frac{\partial G(p;q)}{\partial n_q} ds \quad (3)$$

where the subscripts q and p correspond to the variable point in the integration and the field point, respectively. $G(p;q) = 1/R(p;q)$ is the Green's function with $R(p;q)$ being the distance between points p and q. $\vec{n}$ is the unit vector normal to the rudder and wake surface with a positive direction pointing into fluid domain. S_W is the trailing wake sheet shed from rudder trailing edge. $\Delta\phi_W$ is the potential jump across the rudder trailing wake sheet.

The integral Eq. (3) implies that the potential ϕ at any field point can be expressed as the integration of induced potential due to sources and dipoles over the rudder and cavity surfaces; and can induce potential due to dipoles on the trailing wake surfaces. In the panel method, the modeled geometries are discretized using quadrilateral panels with constant strength dipoles and sources distributed over the rudder, cavity, and wake sheet surfaces.

2.2. Boundary conditions

The solution of integral Eq. (3) can be uniquely determined by applying appropriate boundary conditions on the rudder and wake surfaces. The boundary conditions on the rudder and wake surfaces for the wetted and cavitating problems are as follows:

1. Kinematic boundary condition or tangent condition on wetted surface: the total flow $\vec{q}(x,y,z,t)$ is tangent to the rudder surface.

$$\frac{\partial\phi}{\partial n} = \vec{q}_{in}\cdot\vec{n} \quad (4)$$

2. Kutta condition: the velocities at the trailing edge of the rudder are finite.

$$|\nabla\phi| < \infty \quad \text{at T.E.} \quad (5)$$

An iterative pressure Kutta condition was applied for the analysis of unsteady fully wetted and cavitating rudders by (Kinnas and Hsin, 1992). This condition enforces the pressure on the suction and pressure sides equal at the trailing edge of the hydrofoil.

3. Dynamic boundary condition on the cavity surface: the dynamic boundary condition on the cavity surface requires that the pressure everywhere on the cavity to be constant and equal to the vapor pressure, p_v.

$$p_v + \frac{\rho}{2}\left|\vec{q}(t)\right|^2 + \rho g y = p_\infty + \frac{\rho}{2}\left|\overrightarrow{q_{in}}(t)\right|^2 + \rho g y_{shaft} - \rho\frac{\partial\phi}{\partial t} \quad (6)$$

where p_∞ is the absolute pressure at the propeller shaft axis far upstream and y_{shaft} is the depth of submergence of the shaft axis. ρ is the fluid density and g is the acceleration of gravity.

4. Cavity closure condition: the cavity has to be closed at the cavity trailing edge, a split-panel technique and a Newton-