Available online at www.sciencedirect.com
ScienceDirect
Journal of Hydrodynamics
2017,29(1):154-167
DOI: 10.1016/S1001-6058(16)60727-8
www.sciencedirect.com/
science/journal/l0016058

Numerical estimation of bank-propeller-hull interaction effect on ship manoeuvring using CFD method*

S. Kaidi[1,2], H. Smaoui[1,2], P. Sergent[1]
1. *CEREMA-DtecEMF, 60280 Margny-lès-Compiègne, France*
2. *Sorbonne universités, Université de technologie de Compiègne, laboratoire Roberval, 60203 Compiègne cedex, France, E-mail: sami.kaidi@cerema.fr*

(Received November 9, 2015, Revised May 11, 2016)

Abstract: This paper presents a numerical investigation of ship manoeuvring under the combined effect of bank and propeller. The incompressible turbulent flow with free surface around the self-propelled hull form is simulated using a commercial CFD software (ANSYS-FLUENT). In order to estimate the influence of the bank-propeller effect on the hydrodynamic forces acting on the ship, volume forces representing the propeller are added to Navier-Stokes equations. The numerical simulations are carried out using the equivalent of experiment conditions. The validation of the CFD model is performed by comparing the numerical results to the available experimental data. For this investigation, the impact of Ship-Bank distance and ship speed on the bank effect are tested with and without propeller. An additional parameter concerning the advance ratio of the propeller is also tested.

Key words: Viscous fluid flow simulation (CFD), bank effect, bank-propeller-hull interaction, hydrodynamic forces estimations, advance ratio

Introduction

The transport by inland waterways is considered as an alternative to rail and road transport. Over these years this mode of transport has seen a significant increase due to the encouragement of governments to the exploitation of waterways. This is, on the one hand, to relieve the other modes of transport and, on the other hand, for its ecological quality.

Navigation in inland waterways faces a major risk which concerns mainly accidents due to ship controllability. In contrast to the maritime navigation, the waterways navigation environment plays an important role in the ship manoeuvrability (channel geometry, water depth, bank distance, $\cdots$), therefore, it is important to study the manoeuvrability in confined water in order to offer proposals concerning a development and security.

In the present work, we focus on the study of bank effect. Norrbin[1], was the first to work on Ship-bank interaction. The conclusion of shis experiment has shown a significant impact of the banks on the trajectory of ships. His study was improved later by Ch'ng et al.[2] by taking into account new parameters, such as bank slope, hull form, water depth, bank height and ship speed.

Recently, Duffy[3,4] and Vantorre et al.[5] conducted a series of experimental tests and carried out a study in the influence of some parameters such as water depth, distance to the bank, bank slope, bank height and the forward speed on hydrodynamic force and moment by using a Captive Model Test model. The results of their works showed that the sway force and yaw moment are linearly influenced by the distance to the bank. Few years after, an empirical mathematical formula was proposed by Lataire et al.[6] to calculate the bank-ship interaction forces as a function of the bank geometry, ship speed and propulsion system.

With the fast development of the computer technology and the commercial CFD software, the CFD method has interested the inland community. This method is used mostly and it has proved its ability to predict the ship manoeuvring hydrodynamic forces in

* **Biography:** S. Kaidi (1984-), Male, Ph. D., Confirmed Researcher

medium deep waters.

Miao et al.[7] used a potential flow method for calculating the lateral force and yaw moment for a ship sailing in a rectangular channel. Lo et al.[8] performed a series of simulations to estimate the bank effect of a container ship taking into consideration the viscous action, by using the commercial CFD software based on Navier-Stokes equations. The temporal variation of yaw angle and sway force by varying the ship speed and the distance to the bank were also discussed. Recently, Wang et al.[9] studied the bank effect for a series of hull for several water depth-ship draught ratios and ship-bank distance, using a CFD method to estimate the viscous force better. Ma and Zhou[10] studied the hydrodynamic interaction among hull, rudder and bank. The verification and validation of this method was carried out by Zou et al.[11]. This method was used later to simulate different channel geometries by Zou and Larsson[12].

In this investigation, a preliminary study is presented to predict the combined effect of bank-propeller on the ship manoeuvrability. The theme of this work is the use of viscous CFD model to estimate the different hydrodynamic forces acting on the hull with and without propeller, taking into consideration the influence of ship position on bank, ship speed and the advance ratio of the propeller. The viscous CFD model is updated by adding additional volume forces to represent the propeller action.

1. Problem formulation

1.1 *Governing equations*

The governing equations for mass and momentum conservation are the Reynolds averaged Navier-Stokes (RANS) equations for incompressible flow, by using the Einstein notation these equations are given as bellow:

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

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

$$\rho = \sum_{n=1}^{2}\alpha_n \rho_n , \quad \mu = \sum_{n=1}^{2}\alpha_n \mu_n \tag{3}$$

here $x_i \, (i = 1,2,3)$ are Cartesian coordinates, ρ is the water density and t is time, $u_i \, (i = 1,2,3)$, P and μ are velocity components, pressure and dynamic viscosity respectively, α is the phase fraction, $n = 1,2$ denotes the fluid phase number (water and air), δ_{ij} is the Kronecker delta, $u'_{i(j)}$ represents the fluctuating velocity, $-\rho\overline{u_i' u_j'}$ denote the average Reynolds stresses expressed by

$$-\rho\overline{u_i' u_j'} = \mu\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_x}\right) - \frac{2}{3}\left(\rho k + \mu_{ij}\frac{\partial u_i}{\partial x_i}\right)\delta_{ij} \tag{4}$$

The Reynolds stresses introduce new variables, which makes the equation system Eqs.(1)-(2) not closed. To close and solve this system, several complementary mathematical models with additional equations are proposed, these models are called turbulence models. In the present work, among the various turbulence model proposed by Fluent, the implicit Menter Shear Stress Transport (SST) k-ω model[13] was chosen for its robustness and stability. The (SST) k-ω equations are shown as follows:

$$\frac{\partial(\rho k)}{\partial t} + \frac{\partial(\rho u_i k)}{\partial x_i} = \frac{\partial}{\partial x_j}\left(\Gamma_k \frac{\partial k}{\partial x_j}\right) + G_k - Y_k + S_k \tag{5}$$

$$\frac{\partial(\rho \omega)}{\partial t} + \frac{\partial(\rho u_i \omega)}{\partial x_i} = \frac{\partial}{\partial x_j}\left(\Gamma_\omega \frac{\partial \omega}{\partial x_j}\right) + G_\omega - Y_\omega + D_\omega + S_\omega \tag{6}$$

G_k and G_ω denote the generation of turbulent kinetic energy due to mean velocity gradients and ω, Y_k and Y_ω represent the turbulence dissipation of k and ω, D_ω is the cross-diffusion term, S_k and S_ω are user-defined source terms. Γ_k and Γ_ω express the active diffusivity of k and ω.

1.2 *CFD Solver*

In this work, the incompressible free surface flow around the ship hull is studied using the commercial RANS code "Ansys-Fluent" based on the finite volume method. The pseudo transient pressure-based coupled algorithm is adopted to compute the pressure-velocity coupling, the (PRESTO) interpolation method is selected to compute the cell-face pressure.

To capture the free surface in air-water interface, the implicit volume of fluid (VOF) method based on a second order scheme is employed. The VOF is an Eulerian method particularly used for flows with deformed interfaces. Using this approach the air-water interface can be tracked in a fixed grid by solving the continuity equation of the volume fraction (Eq.(7)).