Ocean Engineering 164 (2018) 414–425

Contents lists available at ScienceDirect

Ocean Engineering

journal homepage: www.elsevier.com/locate/oceaneng

Estimation of hydrodynamic derivatives of a container ship using PMM simulation in OpenFOAM

Hafizul Islam, C. Guedes Soares*

Centre for Marine Technology and Ocean Engineering (CENTEC), Instituto Superior Técnico, Universidade de Lisboa, Portugal

ARTICLE INFO

Keywords:
Static drift
Pure sway
Pure yaw
PMM
Hydrodynamic derivatives
KCS
OpenFOAM

ABSTRACT

Static drift, pure sway and pure yaw simulation results are determined for a container ship. For simulation, open source RANS solver, OpenFOAM was used. The simulation results were compared with two sets of experimental data. Next, hydrodynamic derivatives were predicted from simulation results and compared with experimental ones. The results show good agreement with experimental data, except for some pure yaw cases. Overall, the paper concludes that OpenFOAM is well capable of estimating hydrodynamic derivatives maintaining reasonable accuracy and computational efficiency.

1. Introduction

Maneuverability prediction in the design stage is among the key requirements in ship design to ensure safety and reliability. Traditionally, ship maneuverability characteristics are determined using model tests, which are both expensive and time-consuming. However, with the development of different computational fluid dynamics (CFD) tools and improvement in computational power, designers and researchers are focusing more on CFD to predict ship's seakeeping and maneuverability characteristics.

Ship designers use a number of ways to predict ship maneuverability characteristics in design stage like, theoretical approach, utilization of full-scale database, application of empirical method (Kijima and Nakiri, 2003), model test (Sutulo and Guedes Soares, 2006) and numerical approach using CFD. Theoretical approaches are mostly limited to slender bodies and fail to consider hull and appendage interaction. Utilization of full-scale data is only possible when similar ship models are already in operation. Applications of empirical formulas are also limited to particular hull forms or availability of data. Model tests are among the most popular methods for maneuverability prediction. One common method used in the experimental study to determine ship maneuverability properties is the captive model test, which includes oblique towing test (OTT), rotating arm test (RAT), circular motion test (CMT) and planar motion mechanism (PMM). Similar tests can also be performed using CFD, depending on the capabilities and extent of the solver being used.

The incorporation of CFD in ship maneuverability prediction has been relatively recent. Most of the early works related to CFD based maneuvering were focused on planar motion mechanism (PMM) simulations. Among the early researchers to discuss ship maneuverability using CFD were Simonsen and Stern (2005), Cura-Hochbaum (2006) and Wilson et al. (2006). However, PMM simulations were first widely discussed in SIMMAN 2008 workshop (SIMMAN, 2008), where different research groups presented static drift, pure sway, and pure yaw simulation results. Broglia et al. (2008) showed pure sway and pure yaw motion results for KVLCC1 and 2 models with propeller and rudder simulated using a solver developed by INSEAN. Cura-Hochbaum et al. (2008) simulated static drift, pure sway and pure yaw case for the two tanker models with propeller and rudder, using a self-developed code. Gullmineau et al. (2008) provided PMM results for US Navy frigate using ISIS-CFD solver. Miller (2008) provided PMM calculation for DTMB 5415 using CFDShip-Iowa. Wang et al. (2011) simulated oblique motion for KVLCC2 in deep and shallow water using commercial code FLUENT. Simonsen et al. (2012) presented zig-zag, turning circle and PMM results for an appended KCS model using STAR-CCM+ and compared them with experimental data. Lee et al. (2015) performed PMM simulation for a wind turbine installation vessel using OpenFOAM, ignoring free surface calculation. Later, Shen et al. (2015) incorporated dynamic overset grid in OpenFOAM and presented zig-zag simulation results with self-propulsion. Kim et al. (2015) presented PMM simulation results for KCS model using in-house code SHIP_Motion and predicted hydrodynamic derivatives from simulation results. Hajivand and Mousavizadegan (2015a,b) also performed PMM simulation using STAR-CCM+ and OpenFOAM (static drift only) for DTMB

* Corresponding author.
E-mail address: c.guedes.soares@centec.tecnico.ulisboa.pt (C. Guedes Soares).

https://doi.org/10.1016/j.oceaneng.2018.06.063
Received 23 October 2017; Received in revised form 20 February 2018; Accepted 22 June 2018
Available online 05 July 2018

5512 model and predicted hydrodynamic derivatives from the simulation results. Recently, Yao et al. (2016) presented static drift, turning and pure sway simulation data using OpenFOAM for tanker model KVLCC2.

Although maneuvering related simulations are gaining popularity lately, their applications are mostly limited to large research groups and designers, who have sufficient resources and access to well-developed in-house or commercial codes. An alternative and economic solution to commercial and in-house codes may be the use of the open source CFD toolkit, OpenFOAM. Several researchers have already demonstrated the capability of OpenFOAM in performing maneuverability based simulations for different ship models. However, they were either limited to static cases, or avoided free surface consideration. Furthermore, existing papers do not discuss about the required settings for running such simulations. This paper aims at contributing to the field by presenting static drift, pure sway and pure yaw simulation results for a container ship model (KCS) using OpenFOAM, and estimating the hydrodynamic derivatives for it. The paper aims at demonstrating OpenFOAM's capability in performing maneuvering simulations, with detail regarding the mesh dependency and related solver-setup. It also intends to show that the toolkit is able to perform maneuvering simulations with reasonable accuracy, good efficiency and economy.

2. Method

2.1. Simulation solver

2.1.1. Mathematical model

OpenFOAM (Open Field Operation and Manipulation) is an open source library, written in C++ language following object-oriented paradigm. The code is available under GNU General Public License (GPL). It can be used to numerically solve a wide range of problems in fluid dynamics, from laminar to turbulent flows, with single and multi-phases. It can solve both structured and unstructured polyhedral meshes including h-refinement or hanging nodes and contains an extensive range of solvers to perform different types of CFD simulations. It has several packages to perform multiphase turbulent flow simulation for floating objects. OpenFOAM also allows relatively easy customization and modification of solvers, because of its modular design. The solver has been elaborately described by Jasak (1996, 2009).

The OpenFOAM solver used to perform ship hydrodynamic simulations for this paper simulates incompressible, two-phase flow. The governing equations for the solver are the Navier-Stokes equation (1) and continuity equation (2) for an incompressible laminar flow of a Newtonian fluid. In vector form, the Navier-Stokes and Continuity equation are given by

$$\rho\left(\frac{\partial v}{\partial t} + v.\,\nabla v\right) = -\nabla p + \mu\nabla^2 v + \rho g, \tag{1}$$

$$\nabla v = 0. \tag{2}$$

where v is the velocity, p is the pressure, μ is the dynamic viscosity, g is acceleration due to gravity, and ∇^2 is the Laplace operator. Further, the continuity equation is of the form

The Volume of Fluid (VOF) method is used to model fluid as one continuum of mixed properties. This VOF method determines the fraction of each fluid that exists in each cell, thus tracks the free surface elevation. The equation for the volume fraction is obtained as

$$\frac{\partial \alpha}{\partial t} + \nabla(\alpha U) = 0, \tag{3}$$

where U is the velocity field, α is the volume fraction of water in the cell and varies from 0 to 1, full of air to full of water, respectively.

The unstructured collocated Finite Volume Method (FVM) using Gauss theorem together with user-defined and implemented solution algorithm and time-integration schemes (Drikakis et al., 2007) is used to discretize the governing equations. Time integration is performed by a semi-implicit second-order, two-point, backward-differencing scheme. Pressure-velocity coupling is obtained through PIMPLE algorithm (Ferziger and Peric, 2008), a combination of SIMPLE and PISO. OpenFOAM incorporates three different turbulence models, k-ε, k-ω and SST k-ω. Turbulence is discretized using a 2nd order upwind difference. Turbulence for the presented simulations was modeled with the Reynolds-averaged stress (RAS) SST k-ω two-equation model. The parameters were calculated using common guidelines from an earlier study (Labanti et al., 2016).

2.1.2. Coordinate system

OpenFOAM follows a Cartesian coordinate system, if not specified otherwise. All systems are based on an origin point and coordinate rotation. The solver has a local and a global coordinate system. The local coordinate system is used to define the simulation domain, comparing to global reference point. The global and local coordinate system might be same or different depending on the simulation. In ship simulation, where there is flowing fluid, the local and global coordinates, both are defined using Cartesian coordinate system. The local coordinate systems are generally right handed. The x-axis is positive from stern to bow direction, y is positive at star board side and z is positive upwards, as shown in Fig. 1.

2.1.3. Boundary conditions

The control volume represented a deep water condition, so the two lateral sides and the bottom were symmetry plane type faces; no additional information was required for this kind of boundary condition. Inlet, outlet, and atmosphere were patch faces with specific boundary condition for each one, and hull had a wall type boundary. For the presented simulation cases, boundary conditions used for the fluid properties and turbulence parameters are as shown in Table 1.

Here, FV is fixedValue (Dirichlet Boundary Condition), specified by the user, OPMV is outlet Phase Mean Velocity, PIOV is pressure Inlet Outlet Velocity that applies zero-gradient for outflow, whilst inflow velocity is the patch-face normal component of the internal-cell value and MWV is moving Wall Velocity. FFP is fixed Flux Pressure that adjusts the pressure gradient such that the flux on the boundary is that one specified by the velocity boundary condition; ZG is zero Gradient (Neumann Boundary Condition); TP is total Pressure, calculated as static pressure reference plus the dynamic component due to velocity. IO is inlet Outlet that provides a zero-gradient outflow condition for a fixed value inflow. kqRWF is the wall function for the turbulence kinetic energy, nutkRWF is rough wall function for kinetic eddy viscosity and omegaWF is the wall function for frequency.

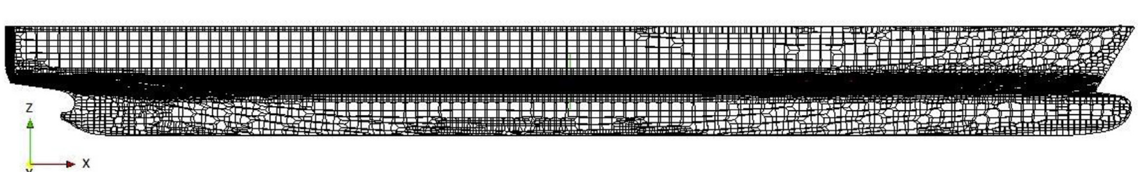

Fig. 1. Local Coordinate system in OpenFOAM.