+ MODEL
ARTICLE IN PRESS
+ MODEL
Available online at www.sciencedirect.com
ScienceDirect
Publishing Services by Elsevier
International Journal of Naval Architecture and Ocean Engineering xx (2017) 1—15
http://www.journals.elsevier.com/international-journal-of-naval-architecture-and-ocean-engineering/

Development of a numerical simulation tool for efficient and robust prediction of ship resistance

Geon-Hong Kim*, Sanghoon Park

Hyundai Maritime Research Institute, Hyundai Heavy Industries Co., Ltd, Republic of Korea

Received 7 June 2016; revised 5 December 2016; accepted 8 January 2017
Available online ■ ■ ■

Abstract

In this paper, a two-phase flow solver HiFoam® has been developed based on the OpenFOAM® to predict resistance of a ship in calm water. The VOF method of OpenFOAM® was reviewed and a simple flux limiter was introduced to enhance the robustness of the solver. The procedure for predicting ship motion was modified by introducing Quasi-Steady Fluid-Body Interaction (QS-FBI) with least square regression to improve the efficiency. Other minor factors were considered as well in terms of the efficiency and robustness. The HiFoam was applied to KCS and JBC simulations to validate its efficiency and accuracy by comparing the results to experimental data and STAR-CCM+. The HiFoam® was also applied to various ships and it showed good agreements to the experimental data.

Keywords: OpenFOAM; Resistance; CFD; VOF; QS-FBI; Flux limiter for VOF

1. Introduction

Computational Fluid Dynamics (CFD) has advanced rapidly in recent years and become as one of the most important technique in engineering fields. Especially the CFD technique plays an important role in shipbuilding industries by replacing experiments successfully at the early design stages. Potential flow codes have been applied to many marine hydrodynamic problems such as resistance, seakeeping, propeller and wave load analysis since it provides solutions in a few minutes with moderate accuracy (Newman, 1992). As an interest in environment grows and the crude oil price rises, the demand on a ship with lower emission and fuel consumption has been increased as well. Consequently, recent ships have more complicated geometries with various appendages that can be hardly handled by the potential codes. Such a problem has been remedied by applying viscous solvers using free surface capturing techniques such as Volume of Fluid (VOF) or level set method. A number of general CFD tools like CFX, Fluent, and STAR-CCM+ have been introduced and applied to ship hydrodynamics problems successfully.

It is well known that such viscous solvers provide reasonable solutions to general CFD problems. Furthermore, as computing hardware and techniques on parallel computing developed very rapidly, CPU time for carrying out a simulation using the viscous solver has been significantly reduced. However, it still takes too long time to be practically used in industrial fields. For instance, a trim optimization analysis requires hundreds of simulations, which should be completed within limited time. If a simulation of a single case takes an hour, it will take more than 4 days to complete 100 simulations. A statistical technique such as Design of Experiment (DOE) might be considered to reduce the time but the computational efficiency should be considered prior to all other factors. Robustness is another important property of a CFD tool. If a CFD tool is too sensitive and easily diverged, it cannot be trusted and used in an industrial field though it provides highly accurate solutions.

* Corresponding author.
Peer review under responsibility of Society of Naval Architects of Korea.

http://dx.doi.org/10.1016/j.ijnaoe.2017.01.003

Please cite this article in press as: Kim, G.-H., Park, S., Development of a numerical simulation tool for efficient and robust prediction of ship resistance, International Journal of Naval Architecture and Ocean Engineering (2017), http://dx.doi.org/10.1016/j.ijnaoe.2017.01.003

Open source techniques have been attracted a lot of attention for the last decade since one can access its source codes to use or modify without paying for it. Many open source CFD codes like SU2 (Palacios et al., 2013, 2014) and PyFR (Witherden et al., 2014) have been introduced to CFD field. The OpenFOAM® is one of the most popular open source CFD codes since it was released to public in 2004. It provides libraries and applications with source codes written in C++ language for the solution of continuum mechanics problems. For ship hydrodynamics problems, the OpenFOAM® provides *interFoam*, a multi-phase solver using interface capturing method based on VOF scheme. It has been studied to predict the resistance of a ship on water including heave and pitch motions accurately by using the OpenFOAM® (Park et al., 2013; Lee, 2013), and they showed that the errors of resistance prediction lie within the range of ±2% relative to experimental data and other commercial CFD tools like STAR-CCM+ and FLUENT. Based on the feasibility studies on the OpenFOAM®, Hyundai Heavy Industries (HHI) began in earnest to use the OpenFOAM® for evaluating performance of a ship numerically since 2014 and developed HiFoam® (Lee, 2014), which is a program for carrying out CFD simulations automatically from pre to post processing based on the *interFoam* of OpenFOAM-2.1.x.

As we mentioned earlier, the accuracy of OpenFOAM® on predicting ship resistance has been validated a lot. However, it is not as robust and efficient as other commercial codes yet which are essential for being widely used in a shipbuilding company. In this paper, we focused on how to enhance robustness and efficiency of the OpenFOAM® solver, especially HiFoam, to make it comparable to commercial codes without losing accuracy.

2. Numerical methods

2.1. Governing equations

interFoam, a solver for two incompressible, isothermal immiscible fluids using a VOF phase fraction based on interface capturing approach, solves following continuity, transport of phase-fraction, and momentum equations.

$$\nabla\cdot\mathbf{u}=0 \tag{1}$$

$$\frac{\partial\alpha}{\partial t}+\nabla\cdot(\mathbf{u}\alpha)=0 \tag{2}$$

$$\frac{\partial(\rho\mathbf{u})}{\partial t}+\nabla\cdot(\rho\mathbf{u}\mathbf{u})=-\nabla p+\nabla\cdot\mathbf{T}+\rho\mathbf{f}_b \tag{3}$$

where α is phase (or volume) fraction that measures how much portion of a cell is filled with the heavier fluid, usually water. Value of α lies within the range of [0, 1] where 0 represents fully gaseous cell while 1 represents a cell fully immersed in the liquid. The properties of a fluid such as density and viscosity can be estimated by using the phase fraction as following.

$$\rho=\alpha\rho_\ell+(1-\alpha)\rho_g \tag{4}$$

$$\mu=\alpha\mu_\ell+(1-\alpha)\mu_g \tag{5}$$

where the subscripts l and g represent liquid and gas, respectively. $\mathbf{T}$ in Eq. (3) is the deviatoric stress tensor where the stress term can be expressed as following for an incompressible flow.

$$\mathbf{T}=2\mu\mathbf{S} \tag{6}$$

$\mathbf{S}$ is the mean rate of strain tensor defined as $\mathbf{S}=0.5[\nabla\mathbf{u}+(\nabla\mathbf{u})^T]$. For Newtonian and incompressible flows, the stress term in Eq. (3) can be further decomposed with the aid of identity of the divergence operation in vector calculus.

$$\nabla\cdot\mathbf{T}=\nabla\cdot(\mu\nabla\mathbf{u})+\nabla\mathbf{u}\cdot\nabla\mu \tag{7}$$

$\mathbf{f}_b$ in Eq. (3) is a body force term including gravity and surface tension, where the surface tension can be evaluated by using the Continuum Surface Force (CSF) model of Brackbill et al. (1992).

$$\mathbf{f}_\sigma=\sigma\kappa\nabla\alpha \tag{8}$$

where κ is mean curvature of the free surface, obtained from following relation.

$$\kappa=-\nabla\cdot\left(\frac{\nabla\alpha}{|\nabla\alpha|}\right) \tag{9}$$

It is common to use modified pressure in VOF method for a convenience in applying boundary conditions for the pressure since it contains hydrostatic components.

$$p_d=p-\rho\mathbf{g}\cdot\mathbf{x} \tag{10}$$

where $\mathbf{g}$ and $\mathbf{x}$ are gravity and position vectors, respectively. The momentum Eq. (3) may be re-written for a two-phase flow as following by using the above relations.

$$\begin{aligned}&\frac{\partial(\rho\mathbf{u})}{\partial t}+\nabla\cdot(\rho\mathbf{u}\mathbf{u})-\nabla\cdot(\mu\nabla\mathbf{u})-\nabla\mathbf{u}\cdot\nabla\mu\\&=-\nabla p_d-\mathbf{g}\cdot\mathbf{x}\nabla\rho+\sigma\kappa\nabla\alpha\end{aligned} \tag{11}$$

2.2. Numerical solution procedure

The governing equations described in the preceding section are discretized by means of finite volume method on an unstructured grid. The van Leer scheme (van Leer, 1979) and linear interpolation scheme are used to discretize the convection and diffusion terms, respectively. The Local Time Stepping (LTS) scheme was selected for temporal discretization for an efficient quasi-steady simulation. The gradients of flow properties can be estimated by using either linear interpolation or least square method.

The *interFoam* uses Pressure-Implicit with Splitting of Operator (PISO) (Rhie and Chow, 1982) algorithm for the pressure−velocity coupling procedure, which is shown in