OpenFOAM-based Computation of Added Resistance in Head and Oblique Seas

Shuguang Wang, Byung-Soo Kim, Zhang Zhu, Jun-Hoi Choi, Yonghwan Kim
Department of Naval Architecture and Ocean Engineering, Seoul National University
Seoul, Korea

ABSTRACT

This work presents CFD computations of ship added resistance and motion responses in head and oblique seas using an in-house CFD solver, snuMHFoam, developed on OpenFOAM. Performance of a ship model named H-CNTR was computed with various wave frequencies and directions to study the wave effects. Effects of ship speed and heading control were also discussed. It was found that the propeller model had little effect on ship added resistance and motion responses. The rudder controller introduced significant low-frequency components to roll and yaw motion. Compared with EFD, the CFD solver showed reasonable predictions of ship performance with respect to wavelength and direction, implying its ability for wave cases.

KEY WORDS: OpenFOAM; added resistance; motion response; controller effects.

INTRODUCTION

A good prediction of ship performance in waves is crucial for the elevation of not only the operational performance of ships in a real sea state, but also the minimum power that ensures the safety. The experimental model test including the captive test (Sigmund and El Mocat, 2018) and free running test (Sanada et al., 2021) is the most reliable method so far because of its real physical phenomenon in the model scale. Its data is usually used for validation. The numerical technique has been widely used for predicting ship added resistance in waves and can be briefly divided into the potential-theory-based method and Computational Fluid Dynamics (CFD). The detailed comparative study of those numerical methods can be referred to Lee et al. (2021).

For CFD computations of ship performance in waves, extensive studies can be found in the Gothenburg 2010 CFD Workshop (Larsson et al., 2013) and the Tokyo 2015 CFD Workshop (Hino et al., 2017). Because of its high resolution of flow information and consideration of viscous effects, it has been becoming popular in ship hydrodynamics, even though it still has evident uncertainties and low efficiency for some practical problems in ship hydrodynamics.

In this work, a container ship model (H-CNTR) designed by Hyundai Heavy Industries was numerically tested in waves using an in-house CFD solver (snuMHFoam) developed on OpenFOAM. Several cases in head and oblique seas were carried out to study the effects of speed and heading control, on ship performance in waves including added resistance and motion responses. Wave effects were also discussed in detail with the comparison to experimental data.

NUMERICAL METHODS

The CFD solver used in this work solves the unsteady Reynolds-Averaged Navier-Stokes (URANS) with a k-omega SST turbulence model in a multiphase flow. The continuity and momentum equation are given as follows.

\[ \nabla \cdot \mathbf{u} = 0 \]

\[ \rho \frac{\partial \mathbf{u}}{\partial t} + \rho (\mathbf{u} \cdot \nabla \mathbf{u}) = -\nabla p + \mu \nabla^2 \mathbf{u} + \mathbf{S}_M \]  

Here, \( \mathbf{u} \) represents the flow velocity, \( \rho \) represents fluid density, \( p \) represents the pressure, \( \mu \) denotes the dynamic viscosity, and \( \mathbf{S}_M \) denotes the source term.

The k-omega SST turbulence model was applied to model the turbulence. Details of this model can be found in Menter et al. (2003). Here are only shown the equations of the turbulence kinetic energy \( k \), turbulence specific dissipation rate \( \omega \), and the turbulence viscosity \( \nu_t \) respectively as follows.

\[ \frac{\partial}{\partial t} (\rho k) = \nabla \cdot (\rho D_k \nabla k) + \rho G - \frac{2}{3} \rho k \nabla \cdot \mathbf{u} - \rho \beta' \omega k + S_k \]

\[ \frac{\partial}{\partial t} (\rho \omega) = \nabla \cdot (\rho D_\omega \nabla \omega) + \frac{\nu_t}{\nu} \nabla^2 \omega - \frac{2}{3} \rho \gamma \omega (\nabla \cdot \mathbf{u}) - \rho \beta \omega^2 - \rho (F_1 - 1) \frac{1}{CD_{kw} + S_\omega} \]

\[ \nu_t = \frac{\alpha_1}{\max(\alpha_1 \omega, k_{Fz2}, S)} \]

Here, \( D_k/D_\omega \) represents the turbulent diffusivity coefficient for \( k/\omega \), \( G \) represents turbulent kinetic energy production rate due to the anisotropic part of the Reynolds-stress tensor, \( S_k/S_\omega \) denotes the source term of \( k/\omega \) equation. \( \nu \) is the kinematic viscosity, \( F_1 \) and \( F_{2z} \) represent bending functions, \( S \) is the invariant measure of the strain rate, and all others are model coefficients.

The Volume of Fluid (VoF) method was used for the multiphase