Ocean Engineering 154 (2018) 143–152

Contents lists available at ScienceDirect

Ocean Engineering

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

Comparative study on steady wave-making problem using viscous and potential-flow methods

Limin Chen [a], Guanghua He [a,*], Atilla Incecik [b], Dazheng Wang [a]

[a] School of Naval Architecture and Ocean Engineering, Harbin Institute of Technology, Weihai, Weihai, Shandong, China
[b] Department of Naval Architecture and Marine Engineering, University of Strathclyde, Glasgow, G1 1XQ, United Kingdom

ARTICLE INFO

Keywords:
Hydrodynamic
Wave-making problem
Rankine panel method
HOBEM
CFD
FINE/Marine

ABSTRACT

Accurate predictions of free surface and pressure are devoted to the design and optimization of ship hull. The present study focuses on a comparison of numerical results from different methods on steady wave-making problem of a standard Wigley hull, especially for pressure distribution on the hull. Two numerical approaches are employed: a higher-order boundary element method, which uses Rankine panel method based on potential flow theory, and a computational fluid dynamics model based on the software FINE/Marine. The computed results of the wave profile along the hull, wave pattern around the hull, pressure distribution on the hull and wave-making resistance are validated by comparing them with experimental measurements and show favorable agreements. Through above comparisons, it is confirmed that the inclusion of nonlinear and viscosity effects improves the accuracy of numerical results for predicting free surface and pressure distribution in spite of heavy computation.

1. Introduction

The prediction of wave-making resistance plays an important role in ship design. Over the past two decades, numerical calculation method in ship hydrodynamics has been rapidly developed with the development of computer science and technology. And it becomes an effective tool for ship design and hull optimization by applying it to hydrodynamic performance analysis and forecast.

The numerical methods can be divided into two categories, namely the potential-flow-based method and the viscous-flow-based method. The traditional methods based on the potential flow theory have been developed early and widely used for predicting wave loads and ship motions in industry, since they require much less computing resources. Up to now, these methods are still widely adopted by industry due to their merits of being fast and easy to implement, such as Zakerdoost et al. (2013), Peng et al. (2014), Ginnis et al. (2014) and so forth. The-strip-theory is the most popular one, which is a 2-D method and has limitations in predicting reliable hydrodynamic forces at high Froude number and low frequency waves. Then 3-D methods became popular in seakeeping research. The available 3-D method can be broadly categorized into two major group: the free-surface Green-function method and the Rankine panel method (RPM). However, the free-surface temporal Green-function is time-consuming in long-term simulation in time-domain analysis. In the last two decades, continual efforts have been made to improve the accuracy and efficiency of these methods, such as Lin and Yue (1991), Bingham et al. (1994), and so forth.

The RPM is widely employed in seakeeping analysis nowadays, since it is flexible in treatment of the complicated free-surface conditions. Raven (1996) developed an iterative code based on the RPM on the raised free surface to solve the fully nonlinear wave resistance problem. Markov and Suzuki (2000) used B-spline scheme in the higher-order Rankine source method for solving 3-D steady ship wave problem. Kara et al. (2007) solved the three-dimensional fully nonlinear steady body-wave interaction problem in time domain based on the Rankine source method. He (2013 and 2014) used a higher-order boundary element method (HOBEM) to solve the steady and unsteady seakeeping problems. A principal disadvantage of the above potential-flow-based methods should be noted that it can not take the influence of viscosity into account. To improve the efficiency and accuracy of numerical simulation as close to actual physical phenomena, the research on viscous flow then becomes essential and important. It is necessary to tackle highly distorted or broken free surface. Conventional numerical methods based on potential-flow theory are not applicable to extremely nonlinear problems.

The development of computational fluid dynamics (CFD) was limited at early stage due to its extensive requirement of computing resources.

* Corresponding author.
E-mail address: ghhe@hitwh.edu.cn (G. He).

https://doi.org/10.1016/j.oceaneng.2018.02.002
Received 29 April 2016; Received in revised form 4 June 2017; Accepted 1 February 2018

However, the problem of heavy computations is solved benefiting from rapid evolution of computer capability. Compared with potential flow theory, CFD is more capable of handling with nonlinear and/or viscous free surface flow problems which are closer to physical phenomena. Yang and Löhner (2002) calculated ship sinkage and trim using a finite element method and unstructured grids based on Euler solver and obtained more satisfactory prediction by considering nonlinear ship waves effects. Gradually, the CFD techniques have been widely used in the simulation of ship resistance and motions by solving Navier-Stokes equations. Biausser et al. (2003) analyzed the internal kinematics and dynamics of three-dimensional breaking waves on slopes by numerical method. Rhee (2005) made research on unstructured grid based Reynolds-Averaged Navier-Stokes (RANS) method for liquid tank sloshing. Hu and Kashiwagi (2009) adopted a constrained interpolation profile based Cartesian grid method for strongly nonlinear wave-body interaction problems and validated the method by a newly designed experiment in a two-dimensional wave channel. Benefiting from the previous progress in CFD techniques, many softwares are developed and applied to scientific research and production, such as Fluent (Wang and Zou, 2014), CFX (Židonis et al., 2015), Star CCM+ (Cinosi et al., 2014), OpenFOAM (Yang et al., 2011) and so forth. Numerical wave tank based on viscous-flow theory has been developed and extended to unsteady seakeeping problems (He et al., 2011; Ferrer et al., 2016). FINE/Marine (NUMECA, 2013) is an integrated CFD software environment for the simulation of mono-fluid and multi-fluid flows around kinds of vessels developed by NUMECA. It specializes in solving the complex free surface problems. The RANS and continuity equations are solved with k-ω (SST-Menter) turbulence model. Gamma differencing scheme (Jasak, 1996) is adopted in discretizing the momentum equation. Free surface is obtained by blend reconstruction interface capturing schemebrics. FINE/Marine has been applied to vehicle and propeller industry in recent years; nevertheless, the research of differences in hydrodynamics analysis compared with other numerical methods is rarely reported.

In the present paper, two 3-D time-domain numerical models based on HOBEM and FINE/Marine are established for steady wave-making problem, respectively. A standard Wigley hull was selected to perform the numerical comparisons, due to its detailed experimental measurements especially for pressure distribution. First, the mathematical formulation of numerical methods are described. After convergence studies with respect to ramp function, domain size and grid number, the steady waves generated by the hull are simulated by both numerical models. The computed wave profile, wave pattern, pressure distribution and wave-making resistance at several Froude numbers are illustrated and compared with measured data reported by Kajitani et al. (1983). The comparison between two numerical methods is discussed and provides an indication about the influence of nonlinear and viscosity effects. An understanding of differences is expected from viscous and potential flow methods which make their complementary application useful for predicting resistance and pressure of ship in still water.

2. Model description

An HOBEM based RPM (He and Kashiwagi, 2014a) and a CFD method based on FINE/Marine are adopted in the comparative study on numerical simulation of steady wave-making problem. In simulation, a rectangular computation domain is introduced as Fig. 1. A Cartesian coordinate system (x, y, z), fixed to a ship in steady motion, is introduced. The horizontal x-y plane is set on the still water surface with its origin placed on the center of the body, and the z-axis is positive upward. A ship is translating at constant forward speed U with respect to a space-fixed reference frame (x_0, y_0, z_0).

2.1. The higher-order boundary element method

For self-containing and clarity, the formulations of the numerical model used in He and Kashiwagi (2014a, 2014b) is outlined here. The fluid is assumed to be incompressible and inviscid with irrotational motion. The velocity potential can be introduced and the total potential is divided into basis, Φ_b, disturbed, ϕ_d, and incident-wave, ϕ_I, potentials:

$$\Psi = \Phi_b + \phi_d + \phi_I \tag{1}$$

$$\zeta = \zeta_d + \zeta_I \tag{2}$$

where ζ is the elevation of free surface; subscripts b, d and I denote the basis, disturbed and incident-wave flows, respectively. The basis flow is assumed to be the largest component with its order $\Phi_b = O(1)$, and the disturbed, ϕ_d, and incident, ϕ_I, flows are assumed small with their order $O(\varepsilon)$.

The basis flow is expressed as

$$\Phi_b = -\boldsymbol{U}\cdot\boldsymbol{x} + \Phi \tag{3}$$

This flow can be either Neumann-Kelvin flow or double-body flow. In the present study the former case is considered.

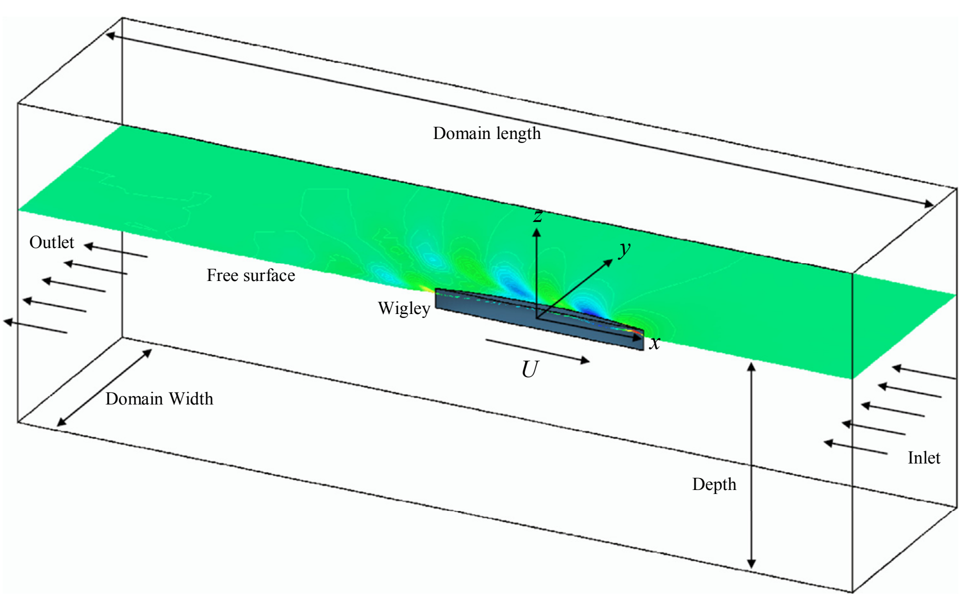

Fig. 1. The rectangular computational domain.