Applied Ocean Research 58 (2016) 281–291

Contents lists available at ScienceDirect

Applied Ocean Research

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

Numerical investigation of wave elevation and bottom pressure generated by a planing hull in finite-depth water

Angelantonio Tafuni[a,*], Iskender Sahin[a], Mark Hyman[b]

[a] Department of Mechanical and Aerospace Engineering – New York University Tandon School of Engineering, 6 MetroTech Center, Brooklyn, NY 11201, USA
[b] Naval Surface Warfare Center – Panama City Division, 110 Vernon Avenue, Panama City, FL 32407, USA

ARTICLE INFO

Article history:
Received 8 October 2015
Received in revised form 1 April 2016
Accepted 2 April 2016
Available online 10 May 2016

Keywords:
Free-surface
Bottom pressure
Planing
High-speed vessel
Waves
Shallow water
Froude number
SPH
Transom

ABSTRACT

A numerical investigation of the bottom pressure and wave elevation generated by a planing hull in finite-depth water is presented. While the existing literature addresses the free-surface deformation and pressure field at the seafloor independently, this work proposes a direct comparison between the two hydrodynamic quantities. The dependence of the pressure disturbances at the ocean floor from the waves generated at the free-surface by a planing hull is studied for several values of both the depth and hull Froude numbers. The methodology employed is Smoothed Particle Hydrodynamics (SPH), a numerical technique based on the discretization of the continuum fields of hydrodynamics through mesh-less particles. The SPH code herein chosen is initially validated against experimental data for transom-stern flow. Subsequently, numerical simulations are presented for a planing hull in high-speed regimes. The results show a direct correlation between surface wave dynamics and hydrodynamic pressure disturbances at the seafloor as the value of the Froude number is varied. This is assessed by studying the inverse dependence of the low-pressure wake angle with the Froude number and by comparison of SPH results with similar works in the cited literature.

1. Introduction

The hydrodynamics of fast hulls cruising in finite-depth water is a recurring topic in the marine community that has received notable consideration in recent times. One of the driving factors of this rising interest is the ever increasing presence of high-speed vessels (HSV) near the coastlines, which not only affect the surrounding environment (e.g. banks erosion, seafloor disruption, etc.), but also influence the water navigability for nearby craft [1]. Therefore, the accurate prediction of field variables such as hydrodynamic pressure and wave elevation generated by HSV is the current subject of several marine engineering studies, see for example [2–4].

The study of water waves generated by a moving disturbance on the free-surface is a subject that has been investigated for many decades, with the pioneering work of Lord Kelvin dating back to the late 1800s [5,6]. In his work, Kelvin had shown that the wake generated by a ship moving in calm water consists of a V-shaped wedge with a wake half-angle of $\sim 19.5^\circ$. Furthermore, he showed that this value remains approximately constant regardless of the ship speed. More recently, Rabaud and Moisy [7] have presented airborne images of ship wakes with a significant narrower angle than the one predicted by Kelvin, and have proposed an analytical model for this phenomenon that has later been verified in [8,9] and reviewed in [10]. Several other studies have been directed towards understanding the wake generated by HSV, with emphasis on the influence of the water depth [11,12], the hull shape [13,14], the interference between the divergent or transverse waves generated by the bow and the stern [15,16], and transom-stern flow [17–19].

In addition to free-surface disturbances, the passage of a ship also generates hydrodynamic pressure signals that can reach the coast or the seafloor when the craft is relatively close to them. When displacement hulls such as trawlers or sailing boats are considered, notable pressure disturbances at the ocean bottom consist of a large depression approximately below amidships and two smaller areas of high pressure, one beneath the bow and one beneath the stern. At cruising speeds characteristic of these craft, the intensity of the low-pressure disturbance can reach two to ten times that of the high pressure, depending on the boat speed and water depth [20–22]. A completely different bottom pressure field is observed for a boat in planing regime: only one high-pressure area with a significantly larger magnitude is now present and it is located in correspondence of the beam waterline, at the point of contact between the hull and the water. The low-pressure region attains a V-shape that resembles that of water waves previously mentioned, with narrower angles at larger depth Froude numbers, $\mathrm{Fr_h} = U/\sqrt{gh}$

* Corresponding author. Tel.: +1 646 997 3940; fax: +1 646 997 3552.
E-mail address: atafuni@nyu.edu (A. Tafuni).

http://dx.doi.org/10.1016/j.apor.2016.04.002

[23,24]. One topic of importance for the safety of ships operating in finite-depth water is the presence of mines on the ocean bottom and their pressure-induced detonation due to the passage of a craft nearby. The neutralization process (mine-sweeping) of such devices consists in utilizing magnetic or acoustic fields to detonate the mine in absence of a target vehicle [22]. In general, the pressure field at the seabed depends on the ambient sea conditions, and can vary dramatically with weather and location, as well as with the action of waves and tides, see for example [25]. However, the design and fabrication of underwater mines allows for an easy recognition of signals with shorter or longer period than the one characterizing the transit of a ship [26]. Therefore, it is crucial to predict the hydrodynamic pressure field generated by a craft in an accurate manner for a better understanding of ships vulnerability to underwater mines and to design suitable mine countermeasures. Analytical and numerical methods have been employed over the past few decades to predict the bottom pressure field generated by travelling surface disturbances. For example, a numerical solution of the potential flow around a submerged Rankine body in a uniform stream of finite-depth water has been proposed in [27,28]. Pressure coefficients therein are calculated for several geometries and depth Froude numbers ranging in [0.3–0.9], showing spatial features of the bottom pressure field and its dependence on $\mathrm{Fr_h}$. In a later work [29], this methodology has been used to predict the flow field generated by an air-cushion vehicle, modelled as a pressure patch moving on the surface of water. Therein, a few supercritical cases are considered up to $\mathrm{Fr_h} = 2.00$, providing the first useful comparisons between bottom pressure generated at subcritical and supercritical speeds. More recently, a hybrid methodology for the computation of the near and far-fields generated by planing boats has been presented in [23]. For the flow in the vicinity of the hull, a Reynolds Averaged Navier Stokes (RANS) solver is employed to accurately model viscous and nonlinear effects, while potential theory is utilized in the outside region representing the far-field, with the RANS solution used as a boundary condition at the interface. Results in this work have been subsequently verified with a good qualitative agreement in [24], with numerous simulations conducted to investigate the variation of the bottom pressure due to the passage of a planing boat at several supercritical speeds.

In this paper, a numerical study of the bottom pressure and wave elevation generated by a planing hull in finite-depth water is presented. The computational methodology is Smoothed Particle Hydrodynamics (SPH) [30–32], a numerical technique that discretizes the continuum fields of hydrodynamics through the use of mesh-less particles. SPH is a robust and reliable CFD method that has been widely used to study marine engineering problems, including slamming in head seas [33], wave breaking of fast ships [34], ship-bubbles interactions [35], fluid–structure interaction [36] and other free-surface problems [37,38]. The choice of SPH as a viable option for the computational work is mainly due to its versatility and simple approach to CFD. The ease with which SPH can provide a large dynamical range in spatial resolution is unmatched in Eulerian methods. Moreover, the complete absence of a grid allows SPH to readily deal with free-surface flow and large regions of space characterized by low particle resolution. The SPH code herein chosen is initially validated against experimental data for transom-stern flow and subsequently used for numerical simulations of a planing monohull in high-speed regimes.

2. Numerical approach

2.1. Smoothed Particle Hydrodynamics

Smoothed Particle Hydrodynamics is a Lagrangian method of probabilistic nature formulated nearly 40 years ago to simulate problems of astrophysical gas dynamics [30,31]. During the last three decades, the method has been modified for deterministic use in the field of hydrodynamics, with the first free-surface works being published in the mid 1990s [39,40]. Currently, SPH represents one of the most robust among particle methods for numerical hydrodynamics and therefore it is used as the main methodology in this study. In the next subsections, a brief overview of the SPH fundamentals is presented as well as some details on the chosen simulation parameters.

2.1.1. Mathematical considerations on SPH

In Smoothed Particle Hydrodynamics, a set of Lagrangian particles is used to discretize the computational domain. These particles have material properties and constitute mesh-less nodes for evaluating the hydrodynamic variables of the flow under investigation. Furthermore, they represent interpolation targets at which the convolution of a field function evaluated at neighboring particles with a smooth interpolant (kernel) provides an approximation of that same function at the target particle. As an example, the two SPH interpolation steps performed on a generic function, $f(\mathbf{x})$, are provided below

$$\langle f(\mathbf{x})\rangle \triangleq \int_{\Omega} f(\mathrm{x}')W(\mathbf{x}-\mathrm{x}', h)\,d\mathrm{x}' \tag{1}$$

$$\langle f(\mathbf{x}_k)\rangle = \sum_{l=1}^{N} \frac{m_l}{\rho_l} f(\mathbf{x}_l) W_{k,l} \tag{2}$$

where $\mathbf{x}$ is the target position vector, $\mathbf{x}'$ represents the position vector of a particle located within the kernel support Ω, N is the total number of particles inside the kernel domain centered at particle k, m_l and ρ_l are the mass and density of the interpolating particle l, and W is the kernel function, with $W_{k,l} = W(\mathbf{x}_k - \mathbf{x}_l, h)$. The angular brackets denote the SPH approximation. The parameter h is the kernel smoothing length and controls the support of W, i.e. the set of neighboring particles that contribute to the computation of $f(\mathbf{x})$ at every interpolation step.

2.1.2. Governing equations

At a macroscopic scale a physical medium is composed of continuous matter, whose state is usually defined by a set of governing equations. In the case of a fluid, two important governing equations on a continuum scale are the continuity and Navier–Stokes equations, defined as

$$\frac{D\rho}{Dt} + \rho\,\nabla\cdot\mathbf{u} = 0 \tag{3}$$

$$\frac{D\mathbf{u}}{Dt} = -\frac{1}{\rho}\nabla P + \mathbf{g} + \nabla\cdot\mathbf{T} \tag{4}$$

where D/Dt is the total derivative, $\nabla\cdot$ is the divergence operator, ∇ is the gradient operator, t is time, $\mathbf{u}$ is the velocity vector, P is the pressure, ρ is the fluid density, $\mathbf{g}$ is gravity and $\mathbf{T}$ is the deviatoric component of the total stress tensor. Applying the notation introduced in Eqs. (1) and (2) to Eqs. (3) and (4) yields the continuity and momentum equations in the SPH framework

$$\frac{D\rho_k}{Dt} = \sum_{l=1}^{N} m_l \mathbf{u}_{k,l}\cdot\nabla_k W_{k,l} \tag{5}$$

$$\frac{D\mathbf{u}_k}{Dt} = -\sum_{l=1}^{N} m_l \left(\frac{P_l}{{\rho_l}^2} + \frac{P_k}{{\rho_k}^2} + T_{k,l}\right)\nabla_k W_{k,l} + \mathbf{g} \tag{6}$$

with $T_{k,l}$ representing the discrete viscous stress tensor. Although the focus of this work is on an incompressible fluid, the compressible continuity and Navier–Stokes equations are deliberately used