Ocean Engineering 147 (2018) 386–398

Contents lists available at ScienceDirect

Ocean Engineering

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

Asymmetric water impact of a two dimensional wedge: A systematic numerical study with transition to ventilating flow conditions

Vesselin K. Krastev *, Andrea L. Facci, Stefano Ubertini

Department of Economics, Engineering, Society and Business Organization, University of Tuscia, 01100 Viterbo, Italy

ARTICLE INFO

Keywords:
Asymmetric water impact
Hull slamming
Computational Fluid Dynamics
Volume Of Fluid

ABSTRACT

Vessels and other marine structures are frequently subject to intense loads deriving from water impact under various conditions. Such forces influence both the structural resistance and the maneuvering capabilities of boats. Water impacts are also relevant in other fields, such as civil engineering, energy harvesting, and aerospace engineering.

In this paper we numerically investigate the hydrodynamics related to the asymmetric water impact of a two dimensional wedge shape. We study the combined effects of geometrical and kinematic asymmetry by systematically varying the wedge roll angle and the direction of the impact velocity. The Volume Of Fluid (VOF) is utilized to model the free surface flow. The numerical model is validated against independent experimental data available in the literature.

The results provided evidence the capabilities of multidimensional modeling in predicting global hydrodynamics and local fluid dynamics features of asymmetric impacts. Moreover, we demonstrate the importance of some fundamental methodological aspects, such as the correct initialization of the computational domain. The role of kinematic asymmetry in flow ventilation occurrence is also clearly highlighted. Our results are expected to facilitate the development of analytical and semi-analytical models and to offer guidelines for conducting experiments and parametric studies for the design of marine vessels.

1. Introduction

The fluid dynamic processes related to water entry problems are of great interest for ship design (Abrate, 2011) and other engineering applications, such as aerospace (Seddon and Moatamedi, 2006) and off-shore structures (Faltinsen, 1993; Facci et al., 2016). Special attention has been given to the hull slamming phenomenon, which usually occurs when a high speed planing boat hull or a sea plane impacts the water (Abrate, 2011; Faltinsen et al., 2004). The generated fluid flow is quite complex and includes free surface piercing and deformation, water sprays, air entrainment and, under some circumstances, flow separation and ventilation (Judge et al., 2004; Shams et al., 2015). All these aspects significantly influence the maneuvering characteristics, as well as the hydrodynamic loading and, consequently, structural fatigue behavior of marine vessels (Gu and Moan, 2002; Faltinsen et al., 2004; Silva and Ravichandran, 2012). A special case is represented by asymmetric impacts which are typical of heeled or yawed planing hulls and, at high speeds, can produce dynamic instability effects which still have to be fully understood (Judge et al., 2004). Impact asymmetry can also occur when any kind of vessel moves with multiple degrees of freedom into steep waves.

Attempts of solving analytically the slamming problem were initially focused on the hydrodynamic load estimation for two-dimensional wedge-like vertical entry configurations (Von Kármán, 1929; Wagner, 1932). These early studies were then progressively extended to different hull geometries and adding more physics to the underlying basic assumptions (Dobrovol'skaya, 1969; Cointe and Armand, 1987; Cointe, 1989; Cointe, 1991; Howinson et al., 1991; Greenhow, 1987). Most of these studies are focused on vertical symmetric impacts. Although still elegant in their formulation, the few known analytical studies on asymmetric water entry (Korobkin, 1988; Miloh, 1991; Chekin, 1989) are generally not meant to describe the asymmetry-generated flow separation and ventilation.

Most known experimental works on the water entry are also limited to symmetric impacts (Tveitnes et al., 2008; Charca et al., 2009; Lewis et al., 2010; Battley and Allen, 2012; Panciroli et al., 2015a; Russo et al., 2017). Remarkable exceptions can be found in (Judge et al., 2004) and in the work from (Shams et al., 2015) which takes benefit from some

* Corresponding author.
E-mail addresses: v.krastev@unitus.it, v.krastev84@gmail.com (V.K. Krastev).

https://doi.org/10.1016/j.oceaneng.2017.10.048
Received 10 April 2017; Received in revised form 16 July 2017; Accepted 22 October 2017
Available online 5 November 2017

recently developed PIV-based pressure reconstruction techniques (Panciroli and Porfiri, 2013; Jalalisendi et al., 2015; Jalalisendi and Osma, 2015; Panciroli et al., 2015b; Russo et al., 2016). Experimental findings pointed out some differences between geometrical and kinematic asymmetry during the water entry (Judge et al., 2004) and allowed to classify the asymmetric impacts depending on the ventilation occurrence on the low pressure side of the wedge (Judge et al., 2004; Shams et al., 2015).

Numerical solutions are powerful tools to efficiently complement experimental measurements in fluid-structure interaction problems. Currently available methods for the simulation of free surface flows include Boundary Element Method (BEM) (Zhao and Faltinsen, 1993; Lu et al., 2000; Wu et al., 2004; Wu, 2007; Xu et al., 2008), Smoothed Particle Hydrodynamics (SPH) (Shao et al., 2006; Oger et al., 2006; Shadloo et al., 2016), Lattice-Boltzmann (LB) (Falcucci et al., 2011; Zarghami et al., ; De Rosis et al., 2014) and Navier-Stokes (NS) solvers. Additionally, several types of mesh based NS approaches can be found in the scientific literature, such as the Volume Of Fluid (VOF), Level Set (LS) or Artificial Compressibility (AC) methods (Hirt and Nichols, 1981; Osher and Sethian, 1988; Kelecy and Pletcher, 1997; De Jouëtte et al., 2001; Carrica et al., 2010; Chung, 2013; Ma et al., 2015). Recent applications of VOF to symmetric water entry problems can be found in (Facci et al., 2015; Facci and Ubertini, 2015). A two fluid LS method is proposed in (Gu et al., 2014) and utilized to study the water entry of two-dimensional (2D) rigid shapes showing a qualitatively consistent behavior. To the best of our knowledge a systematic validation of CFD models for different oblique entry configurations and an insight on the physical and geometrical parameters which influence the transition from non-ventilating to ventilating flow are missing in the scientific literature.

This paper presents a detailed numerical study of a wedge-like two-dimensional section impacting water surface under a variety of geometrical and kinematic asymmetry conditions. The simulations are performed through a VOF-based algorithm (Rider and Kothe, 1998; Scardovelli and Zaleski, 1999; Rusche, 2002), coupled with a Laplacian mesh-motion solver (Jasak and Tukovic, 2007; Jasak, 2009; Kassiotis, 2008), already employed and validated for symmetric water entry problems (Facci et al., 2015; Facci and Ubertini, 2015). The paper is organized as follows. The main features of the problem in study are described in Section 2. Then, a focus on the numerical approach, including details on the computational domain and numerical setup is given in Section 3.1. In Section 4 the methodology is validated by comparing present numerical results to literature experimental data (Judge et al., 2004). The hydrodynamics related to the water impact is discussed in Section 5. Conclusions are drawn in Section 6.

2. Problem statement

In this paper we investigate the two dimensional hydrodynamics related to the asymmetric water impact of a rigid wedge through Computational Fluid Dynamics (CFD).

The problem is schematically represented in Fig. 1, which reports the relevant characteristics of the specimen and of the fluid flow. The geometry of the wedge is defined in terms of its width $l = 0.20$m, height $h = 0.08$m, and dead-rise angle $\beta = 37^{\circ}$. Such a geometry is selected to reproduce the experiments described by (Judge et al., 2004).

We hypothesize that the wedge breadth is much larger compared to l and to h, such that the geometry, and the hydrodynamics can be considered two dimensional (Jalalisendi et al., 2015). A fixed Cartesian reference frame is selected with the $x-$axis along the horizontal direction (i.e. along the water free surface) and the $y-$axis along the vertical direction.

The hull keel impacts on the water surface at the origin of the reference frame and at time $t = 0$ s. We denote with $\varepsilon(t)$ the abscissa of the wedge keel and with $\xi(t)$ its ordinate at time t. The wedge kinematics is defined by its constant velocity $\boldsymbol{v} = (\dot{\varepsilon}, \dot{\xi})$ being the superimposed dot the time derivative, and by the constant roll angle θ. The velocity ratio $\tau = \dot{\varepsilon}/\dot{\xi}$ quantitatively represents the kinematic asymmetry of the water entry, and the angle θ represents the geometrical asymmetry.

We assume that the two fluids (air and water) are immiscible and the flow is incompressible. Thereafter, the flow field is described by the mass conservation and momentum balance equations, as follows:

$$\nabla \cdot \boldsymbol{u} = 0, \tag{1a}$$

$$\frac{\mathrm{D}(\rho \boldsymbol{u})}{\mathrm{D}t} = -\nabla p + \nabla \cdot \mu \nabla \boldsymbol{u} + \rho \boldsymbol{g}, \tag{1b}$$

where $\boldsymbol{u}$ is the fluid velocity, p is the pressure, μ is the dynamic viscosity, ρ is the density, $\boldsymbol{g}$ is the gravitational acceleration, and $\mathrm{D}(\cdot)/\mathrm{D}t$ denotes the material derivative. Note that ρ and μ vary in space and time according to the fluid phase, air or water. In particular, the air density and viscosity are $\rho_{\mathrm{a}} = 1\mathrm{kg/m}^3$ and $\mu_{\mathrm{a}} = 1.48 \times 10^{-5}$Pa s, respectively, and the water density and viscosity are $\rho_{\mathrm{w}} = 1000\mathrm{kg/m}^3$ and $\mu_{\mathrm{w}} = 1 \times 10^{-3}$Pa s, respectively.

Experimental observations show that for low τ and θ the flow remains attached on both sides of the wedge (Judge et al., 2004; Shams et al., 2015), as qualitatively represented in Fig. 1a, and the flow asymmetry is defined by the ratio $b^* = b_2/b_1$, where b_1 and b_2 are the jet root abscissa on the lower and upper side of the wedge respectively. As in (Judge et al., 2004), from now on we will refer to such a flow as type A flow. Increasing the asymmetry (i. e. decreasing b^*), a cavity should appear on the upper side of the hull (see Fig. 1b) and the flow is denoted as type B flow (Judge et al., 2004).

The encompassing fluid reacts to the wedge motion generating a force that can be expressed as:

$$\boldsymbol{F} = \int_{\Sigma} \left\{ -p\boldsymbol{I} + \mu[\nabla \boldsymbol{u} - \nabla \boldsymbol{u}^{\mathrm{T}}] \right\} \boldsymbol{n} \mathrm{d}\Sigma, \tag{2}$$

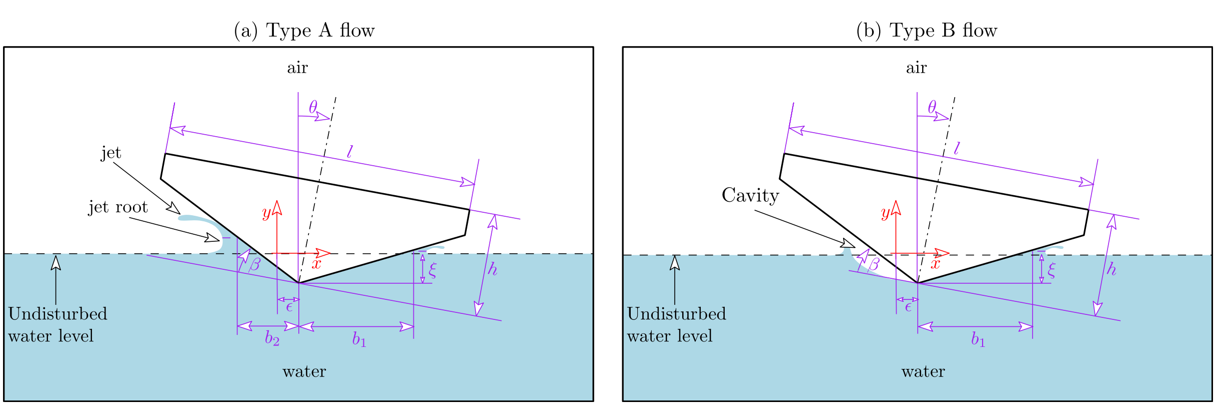

Fig. 1. Schematic representation of the problem in study for the two limiting flow conditions.