Ocean Engineering 146 (2017) 125–139

Contents lists available at ScienceDirect

Ocean Engineering

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

Review

Numerical simulation of dam breaking and the influence of sloshing on the transfer of water between compartments

M.A.C. Celis [a,*], J.B.V. Wanderley [b], M.A.S. Neves [b]

[a] Federal University of Pernambuco, Naval Architecture Group, Department of Mechanical Engineering, Recife, PE, Brazil
[b] Federal University of Rio de Janeiro, Program of Ocean Engineering, Rio de Janeiro, RJ, Brazil

ARTICLE INFO

Keywords:
Dam breaking
Sloshing
CFD
FDM
TVD
Acoustic waves

ABSTRACT

Dam breaking and sloshing are hydrodynamic phenomena of great importance in the assessment of progressive flooding of ships and offshore platforms. Recent experimental tests have generated more data with different configurations of openings and flooded compartments. The internal geometry of the compartments is important on the behavior and impact of the flooded water on the vessel motion. In this paper, a computational code is validated to simulate dam breaking and sloshing with two different configurations. Numerical results were obtained for recent experiments on transfer of water between connected neighboring compartments with two configurations of openings. The comparison with experimental data validated the numerical code and showed some important details of the flow field. The two configurations of sloshing are excited in sway motion, where certain amount of water flows from one side to the other through an opening and the other way round. The implemented numerical scheme is based on the finite difference method, where the Euler equations are solved using an upwind total variation diminishing scheme with a structured computational mesh.

1. Introduction

The motion of a fluid into the space of a vessel during flooding is violent and complex. In the first stage, dam breaking has a strong effect on the stability of the ship, particularly in the roll motion. This stage refers to the transient phase, according to IMO SLF46/INF.3 (2003) report. Transient, progressive flooding and steady state stages of flooding may take place. It is noted that the damaged naval artifact usually reaches a maximum roll angle in the transient phase. The first stage is followed by stable progressive flooding without sinking. Another situation would be the capsizing of the ship as a consequence of a big dam breaking and finally sinking.

The effect of pressure on the doors and windows of the vessels has been studied and reported experimentally and numerically. This study has been presented in Ruponen and Routi (2011), Naar and Vaher (2010), and is available in IMO SLF 53/INF.2 (2010). The collapsing of doors at the bottom is logical since most of the pressure is on the lower part of the door. The Stability in Waves (SiW) committee of the 27th ITTC (2014) recommended the investigation of scale effects and the air pressure effects during the flooding process. In this paper, some effects of the acoustic wave propagation are registered numerically as affected by the development of the pressure field.

During the progress of flooding, sloshing may be present. Large moments and forces affect the behavior of the vessel during the flooding progress. Computational fluid dynamics (CFD) simulations may provide detailed information on the air and water behavior. There are effects on the air-water interface with large variations of flow properties. However, the simulation requires careful initial and boundary conditions with a good computational grid. This paper focuses on the transfer of mass, hydraulic jump, pressure and acoustic waves (AW) without consideration of viscous effects. Euler equations are used because of the low computational time required. The dam breaking and sloshing can occur in a very short time interval when the opening is large and inertia mass is more important than viscous effects. The dam breaking simulations were validated and compared with experimental results obtained by Martin and Moyce (1952), and numerical results from Nielsen (2003) and Colagrossi and Landrini (2003).

A lot of time and effort is nowadays spent on the research of flooding of ships and identifying parameters to predict the occurrence of flooding and sloshing. Numerical simulations are necessary to predict the pressure distribution behavior and other effects during dam breaking. Manderbacka et al. (2014a) presented two experimental configurations and results with openings at the bottom and near the bottom between compartments. The experimental apparatus is shown in Fig. 1.

* Corresponding author.
E-mail address: miguel.celis@ufpe.br (M.A.C. Celis).

https://doi.org/10.1016/j.oceaneng.2017.09.029
Received 27 April 2016; Received in revised form 4 May 2017; Accepted 24 September 2017
Available online 2 October 2017

Nomenclature

a	Sound speed
c	Volume fraction
e	Total energy
g	Gravity acceleration
H	Compartment height
L	Length of the compartment
M_∞	Mach number
ρ_w	Water density
ρ_{ar}	Air density
γ	Adiabatic expansion coefficients
u, v	Velocity of the fluid particle in the direction of the axes x, y respectively
F_n	Froude number
$Q_t = \partial Q/_{\partial t}$	Partial derivative respect to time

The sloshing is extensively investigated and studied with three focuses: analytical, numerical and experimental. Abramson (1966) reported the non linearities of the sloshing problem in rigid containers in the context of space technology applications. Ibrahim (2005) presented a quite basic theory and extensive studies of sloshing with different container geometries. Faltinsen and Timokha (2009) presented analytical methods and experiments on sloshing, with a focus on ship tanks. They additionally included nonlinear multimodal methods.

Numerical approximations using CFD methods to simulate the sloshing problem have greatly increased with the development of computer technologies. Antuono et al. (2012) presented an analytical approximation for sloshing in shallow water and compared with numerical results based on smoothed particle hydrodynamics (SPH), a method developed by Gingold and Monaghan (1977) and Lucy (1977) initially for astrophysical problems. Bouscasse et al. (2014a, 2014b) presented a numerical model based on SPH to simulate the sloshing in tank excited in roll motion. The *moving particle semi-implicit* (MPS) method was used by Koshizuka and Oka (1996) to simulate the incompressible fluid and compare with experimental data of the dam break phenomena. In a recent publication MPS method has been used by Fonfach et al. (2016) to simulate the transfer of mass between compartments, presenting a good agreement with experimental and lumped mass method results.

Numerical methods based on grid or mesh have been widely applied to simulate the sloshing and dam breaking such as the finite differential method (FDM) or volume of fluid method (VOF); these are currently the dominant methods to solve problems in engineering and science. The experimental results of Martin and Moyce (1952) on dam breaking has been used by Hirt and Nichols (1981) to validate their numerical model based on VOF. Armenio and Rocca (1996) adopted FDM to simulate the sloshing in a tank with a vertical baffle. They reported that the vertical baffle considerably reduced the sloshing loads in the whole range of roll frequencies.

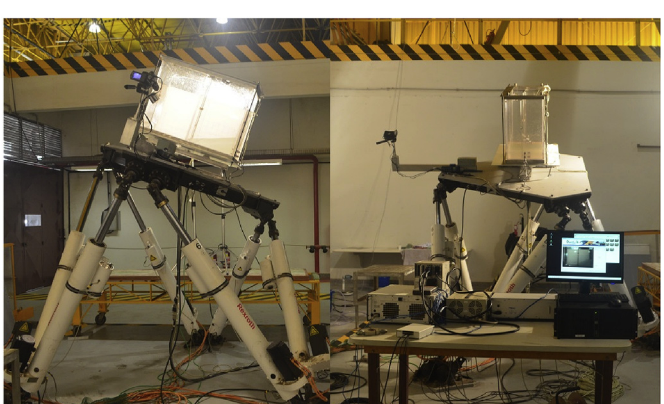

Fig. 1. Experimental tank mounted on the hydraulic platform (LabOceano).

In this paper the volume fraction equation is used to capture the air-water interface displacement in order to investigate the phenomenon. On the air-water interface effect includes the existence of large fluid property gradients, which makes it difficult to simulate numerically this kind of flow without spurious oscillations, Toro (1999). Therefore, the *total variation diminishing* (TVD) scheme of Roe (1984) and Sweby (1984) is used to circumvent this difficulty. The TVD scheme (well known in aeronautical applications) has been used in other flow phenomena like roll damping decay, Avalos et al. (2014) and to study the vortex induced vibrations, Wanderley et al. (2008), Wanderley and Soares (2015). In all these marine applications, excellent results have been obtained. These very successful results have led the Authors to consider in the present paper the introduction of a TVD scheme to deal with the slightly compressible formulation here adopted for the solution of dam breaking and sloshing problems.

2. Mathematical formulation

The mathematical formulation is based on fundamental principles of physics, such as the principles of conservation of mass (continuity), and momentum. These principles can be expressed in terms of mathematical equations in their most general forms as partial differential equations.

2.1. Governing equation

In the present work, the dam breaking and sloshing with transfer of mass of water between compartments are simulated without viscous effects. So, the Euler equations are employed to describe the water and air behavior. The differential form of these governing equations is presented in Eq. (1), see Wanderley (2001).

$$\left.\begin{aligned} &\frac{\partial p}{\partial t} + \nabla.(p\vec{V}) = 0 \\ &\frac{\partial \vec{V}}{\partial t} + \vec{V}.\nabla\vec{V} = -\frac{\nabla p}{\rho} - g\hat{j} \\ &\frac{\partial c}{\partial t} + \vec{V}.(\nabla c) = 0 \end{aligned}\right\} \tag{1}$$

Equation (1) can be written in the vector and non-dimensional forms as shown in Eq. (2). Vector Q is the vector of conserved variables, vectors E and F are the inviscid flux vectors, and H is the forcing term.

$$Q_t + E_x + F_y = H \tag{2}$$

where

$$Q = \begin{Bmatrix} p \\ u \\ v \\ c \end{Bmatrix}, E = \begin{Bmatrix} up \\ u^2 + p \\ vu \\ uc \end{Bmatrix}, F = \begin{Bmatrix} vp \\ uv \\ v^2 + p \\ vc \end{Bmatrix}, H = \begin{Bmatrix} 0 \\ 0 \\ -\left(\frac{M_\infty}{F_n}\right)^2 .g \\ c(\nabla \cdot V) \end{Bmatrix} \tag{3}$$

Pressure distribution on the walls of the compartments is obtained by the numerical solution of the Euler equations to calculate the forces and moments due to the fluid motion. The governing equations are solved numerically with initial conditions, see Eq. (4), and boundary conditions on the walls, as shown in Fig. 2. The Mach number for incompressible flows is assumed to be $M_\infty = 0.2$.