Ocean Engineering 42 (2012) 176–187

Contents lists available at SciVerse ScienceDirect

Ocean Engineering

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

# Numerical investigation of the effects of turbulence intensity on dam-break flows

Il-Ryong Park [a,*], Kwang-Soo Kim [b], Jin Kim [b], Suak-Ho Van [b]

[a] Department of Naval Architecture & Ocean Engineering, Dong-Eui University, 995 Eomgwangro, Busan_Jin_Gu, Busan 614-714, Korea
[b] Marine Transportation System Laboratory, Maritime and Ocean Engineering Research Institute (MOERI), KORDI, Jang-Dong 171, Yuseong-Gu, Daejon 305-343, Korea

A R T I C L E  I N F O

*Article history:*
Received 2 November 2010
Accepted 2 January 2012
Editor-in-Chief: A.I. Incecik
Available online 31 January 2012

*Keywords:*
Dam-break flow
RANS
$k$–$\varepsilon$ model
Initial turbulence intensity
Frictional drag
VOF

A B S T R A C T

Resistance to the propagating motion of water in dam-break problems increases where the flow depth becomes small, in particular, in the region close to the water front, resulting in the retardation of the surging front of water and the change in its curvature. This paper is focused on the numerical investigation of the effects of this basal resistance on the unsteady motion of dam-break flows with considering the development of turbulence. For this purpose, a volume of fluid (VOF) advection algorithm coupled with the Reynolds averaged Navier–Stokes (RANS) equations with a two-equations turbulence closure model is employed. For efficiently describing and predicting the degree of turbulence in dam-break flows, the computations are carried out with the variation of initial turbulence intensities. The present numerical observations reasonably describe the relevant physical aspects of dam-break flows, showing a satisfactory agreement in comparison with available experimental data without and with the impact of water.



## 1. Introduction

Dam-break flows occur when a column of water is released instantaneously from rest by the sudden removal of a vertical barrier that initially contains water (Ritter, 1892). In spite of its simplicity, this type of shallow gravity-driven flow has been studied both theoretically and experimentally for many years due to the wide range of its engineering applications in the area of marine hydrodynamics and coastal engineering. For example, this problem has several features related to slamming and green-water loads on ships and offshore structures, sloshing loads in tanks, and wave loads on coastal structures. Experimental studies on the dam-break problem have been carried out based on observations of the shape of the propagating front of water, its velocity and the effects of drag forces on dam-break flows (Keulegan, 1950; Martin and Moyce, 1952; Dressler, 1954; Koshizuka and Oka, 1984; Lauber and Hager, 1998; Stansby et al., 1998; Zhou et al., 1999). To analyze this problem, most theoretical studies have been performed usually based on solutions to the governing shallow water model of the flows with or without the hydraulic resistance modeled in the form of basal friction (Whitham, 1955, 1974; Hogg and Woods, 2001; Hogg and Pritchard, 2004). In the same context, various free surface modeling techniques coupled with the different governing equations have been proposed to reproduce dam-break flows at laboratory scales (Ubbink, 1997; Greco, 2001; Colicchio et al., 2002; ten Caat, 2002; Colagrossi and Landrini, 2003; Nielsen, 2003; Qian et al., 2003; Andrillon and Alessandrini, 2004; Greco et al., 2004; Greaves, 2005; Violeau and Issa, 2007; Garcia-Espinosa et al., 2008; Ferrari et al., 2009; Khayyer et al., 2009; Park et al., 2009; Lee et al., 2010; Lv et al., 2010; Dumbser, 2011). In particular, there are two representative experiments often selected as a benchmark test to validate the numerical performance of the proposed numerical approaches in solving free surface flows. The first one was presented in detail by Martin and Moyce (1952), which was repeated later by other research groups using different techniques (Koshizuka and Oka, 1984; Stansby et al., 1998). The second one is an experiment considering the impact of water conducted by Zhou et al. (1999) for a different purpose of investigating green water impact loads on ships and offshore structures.

This study was inspired by the fact that besides experimental uncertainties, some of the relevant physical aspects not modeled or underestimated in the numerical simulations as discussed in Colagrossi and Landrini (2003) make the differences found between numerical results and experimental data for the unsteady motion of dam-break flows (Colagrossi and Landrini, 2003; Andrillon and Alessandrini, 2004; Greaves, 2005; Violeau and Issa, 2007; Ferrari et al., 2009; Lee et al., 2010; Lv et al., 2010). One of the important physical aspects may be related to the

* Corresponding author. Tel.: +82 51 890 2595; fax: +82 51 890 2634.
*E-mail address:* irpark@deu.ac.kr (I.-R. Park).

0029-8018/$ - see front matter © 2012 Elsevier Ltd. All rights reserved.
doi:10.1016/j.oceaneng.2012.01.005

hydraulic resistance increased due to the turbulence developing in the flows. In terms of the effects of turbulence on dam-break flows, a good example of the comparison between lower and higher order turbulence closure models can be found in the reference by Violeau and Issa (2007). They reported that the accuracy of the free surface prediction can be significantly improved by a higher order complex turbulence closure model, even though further progress for validation is necessary. In particular, they showed that the non-linear eddy viscosity plays an important role to predict the correct behavior of wave breaking by increasing diffusion. It can be noted that this turbulent eddy viscosity may be one of important factors to slow the motion of dam-break flows by increasing basal friction. Based on this numerical observation, in this paper, we focus on the effects of turbulence intensity on frictional drag which cannot be neglected in the dam-break problem, in particular, in the region close to the shallow water front of the flow.

Numerical simulations of dam-break flows are carried out by employing a volume of fluid (VOF) advection algorithm coupled with the Reynolds averaged Navier–Stokes (RANS) governing equations with a two-equations turbulence closure model. This approach is based on the previous work of Park et al. (2009). For efficiently describing and predicting the level of frictional drag caused by the turbulence in the developing dam-break flows, the computations are performed with varying initial turbulence intensities instead of the quantitative prediction by using one of more sophisticated higher order turbulence closure models. The key numerical observations of the changes in the retardation of the motion of water, in the wall shear stress, in the frictional drag and in the shape of the water front are provided for Martin and Moyce's dam-break test case (1952), which has not been yet shown in other numerical results for dam-break problems in the literature. For the same experimental application, an additional simulation with the variation of surface roughness is also performed. In the second dam-break application, the free surface elevations and violent impacts of water are compared with the experimental data provided by Zhou et al. (1999) to further verify the effects of turbulence intensity on dam-break flows.

## 2. Numerical approach

### 2.1. Governing equations

A single set of the mass and Reynolds-averaged Navier–Stokes momentum conservation equations for incompressible two-phase free surface flows is given by

$$\frac{\partial u_k}{\partial x_k} = 0 \tag{1}$$

$$\frac{\partial(\rho u_i)}{\partial t} + \frac{\partial(\rho u_i u_j)}{\partial x_j} = -\frac{\partial p}{\partial x_i} + \frac{\partial \tau_{ij}}{\partial x_j} + \rho g_i \tag{2}$$

where $\rho$ is the density, $t$ is time, $u_i$ are the velocity components in the direction of the Cartesian coordinates $x_i$, $p$ is the static pressure, and $g_i$ is the gravitational acceleration. The mean stress tensor $\tau_{ij}$ related to the Reynolds stress terms which characterize the influence of the turbulence on the mean flow field can be approximated by using Boussinesq's isotropic eddy viscosity hypothesis as follows:

$$\tau_{ij} = \mu_e\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i}\right) - \frac{2}{3}\delta_{ij}\rho k \tag{3}$$

where $\delta_{ij}$ is the Kronecker delta, $k$ is the turbulent kinetic energy, and $\mu_e$ is an effective viscosity, i.e., the sum of the turbulent eddy viscosity $\mu_t$ and molecular dynamic viscosity $\mu$:

$$\mu_e = \mu_t + \mu \tag{4}$$

The turbulent eddy viscosity $\mu_t$ is calculated using the standard $k$–$\varepsilon$ model (Launder and Spalding, 1974). In order to avoid using too many grid points in the viscous sub-layer, the so-called Launder and Spalding's wall function (Launder and Spalding, 1974) is implemented. In the standard $k$–$\varepsilon$ turbulence model, $\mu_t$ is defined by

$$\mu_t = \rho C_\mu \frac{k^2}{\varepsilon} \tag{5}$$

where $C_\mu$ is a dimensionless constant and typically $C_\mu = 0.09$ for steady turbulent flow. The value of $C_\mu$ can be selected more properly based on the shallow water assumption for modeling dam-break flows. We use $C_\mu = 0.06$ determined experimentally in the shallow bottom boundary layer outside surf zone (Cox et al., 1994). $\varepsilon$ represents the dissipation rate of turbulent kinetic energy. The transport equations for the turbulent kinetic energy $k$ and the dissipation rate $\varepsilon$ are given by

$$\frac{\partial(\rho k)}{\partial t} + \frac{\partial(\rho u_j k)}{\partial x_j} = \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_t}{\sigma_k}\right)\frac{\partial k}{\partial x_j}\right] + \mu_t\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i}\right)\frac{\partial u_i}{\partial x_j} - \rho\varepsilon \tag{6}$$

$$\frac{\partial(\rho \varepsilon)}{\partial t} + \frac{\partial(\rho u_j \varepsilon)}{\partial x_j} = \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_t}{\sigma_\varepsilon}\right)\frac{\partial \varepsilon}{\partial x_j}\right] + C_{\varepsilon 1}\frac{\varepsilon}{k}G - C_{\varepsilon 2}\frac{\rho\varepsilon^2}{k} \tag{7}$$

where $\sigma_k$, $\sigma_\varepsilon$, $C_{\varepsilon 1}$, and $C_{\varepsilon 2}$ are the model constants determined experimentally ($\sigma_k = 1.0$, $\sigma_\varepsilon = 1.3$, $C_{\varepsilon 1} = 1.44$, and $C_{\varepsilon 2} = 1.92$).

### 2.2. Initial turbulence intensity and surface roughness

Initial turbulence intensity is an efficient parameter to describe the degree of turbulence in a flow field. When a flow feature is dependent on the initial turbulence intensity, it is important to control this parameter in order to obtain accurate numerical results. The initial values of $k$ and $\varepsilon$ must be imposed in the flow field to start computations since the adopted standard $k$–$\varepsilon$ turbulence equations are homogenous. These initial turbulence quantities can be specified in terms of the turbulence intensity and turbulence length scale defined as follows:

$$k = \tfrac{2}{3}(U_c I)^2 \tag{8}$$

$$\varepsilon = \frac{C_\mu^{3/4} k^{3/2}}{\ell} \tag{9}$$

where $U_c$ is the reference velocity magnitude, $U_c = 2\sqrt{gh}$ which is the theoretical celerity of the dam-break wave front, where $h$ is the initial reservoir height in dam-break problems. $I$ is the initial turbulence intensity. $\ell$ is the turbulence length scale and typically $\ell = 0.07D$ for fully developed pipe flows in which $D$ is the relevant dimension of the pipe. The value of 0.07 is based on the maximum value of the mixing length in fully developed turbulent pipe flow. In this paper, $\ell = 0.04\ d$ which follows the measured result by Cox et al. (1994), where $d$ is the flow depth.

Surface roughness effects can be considered in the computation by the wall function modified as follows:

$$\frac{\rho U_P u_\tau}{\tau_w} = \frac{1}{\kappa}\ln\left(E\frac{u_\tau y_P}{\nu}\right) - \Delta B \tag{10}$$

where $U_P$ is the velocity parallel to the wall at the computational is the kinematic viscosity and $y_P$ is the distance at the point $P$ to the wall. Since this modification is based on the experiments of Nikuradse (1933) on roughness effects on flow in pipes roughened with sand grains, the last term of Eq. (10), $\Delta B$ can be defined as a function of dimensionless sand grain roughness, $K_S^+ = K_S u_\tau/\nu$, where $K_S$ is the equivalent sand grain roughness height. Cebeci and Bradshaw (1977) classified the roughness into three regimes: hydrodynamically smooth ($K_S^+ < 2.25$), transitional ($2.25 \le K_S^+ \le 90$) and fully rough ($K_S^+ > 90$), and provided the formula for $\Delta B$ for the