Ocean Engineering 159 (2018) 358–371

Contents lists available at ScienceDirect

Ocean Engineering

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

Numerical study on characteristics of dam-break wave

Shaolin Yang [a], Wanli Yang [b,c,*], Shunquan Qin [d], Qiao Li [a], Bing Yang [e]

[a] Department of Bridge Engineering, School of Civil Engineering, Southwest Jiaotong University, Chengdu, 610031, China
[b] Key Laboratory of High-speed Railway Engineering, Ministry of Education, Chengdu 610031, China
[c] School of Civil Engineering, Southwest Jiaotong University, Chengdu 610031, China
[d] CAE (Chinese Academy of Engineering) Academician, Department of Bridge Engineering, School of Civil Engineering, Southwest Jiaotong University, Chengdu, 610031, China
[e] Department of Geotechnical Engineering, School of Civil Engineering, Southwest Jiaotong University, Chengdu, 610031, China

ARTICLE INFO

Keywords:
Dam-break wave
FLOW-3D
Wave propagation
Surface profile
Bottom resistance
Turbulent kinetic energy

ABSTRACT

The influence of the downstream water depth on dam-break wave is systematically and comparatively studied by simulating the physical experiments in this study. Commercial CFD software package FLOW-3D is chosen as the simulation tool and is validated firstly. Three experimental cases of different initial downstream water depth, including dry-bed case ($r = 0$), shallow downstream water case ($r = 0.1$) and deep downstream water case ($r = 0.4$) carried out by Ozmen-Cagatay and Kocaman (2010) are then simulated. The generation and propagation process of dam-break wave is divided into two and three stages, respectively, for dry-bed and wet-bed cases. And the characteristics of the dam-break wave including free surface profile, wave front velocity, mark point movement and trajectory, interface between upstream and downstream water, bottom resistance and turbulent kinetic energy are compared and analyzed elaborately in each stage for each case. The differences of the characteristics between different downstream water depth cases are achieved and the reasons of the differences are explored, which present a comprehensive understanding of dam-break wave generation and propagation.

1. Introduction

Dam-break wave could cause catastrophic flood disaster to the downstream area due to its large velocity, deep submergence and huge flux. The accurate prediction of the wave characteristics, i.e. propagation velocity, submerged depth and arrival time, has genuine practical significance for early warning and emergency evacuation to reduce the loss of life and property damage. And it is also the basis of hydrodynamic force calculation of houses and bridges in the downstream area. Valuable researches in the field of dam-break wave have been done through theoretical analysis, experimental research and numerical calculation.

Dam-break wave propagates over dry-bed and wet-bed are usually studied separately in literature due to their significant differences in flow patterns. Ritter (1892) studied the problem of dam-break wave propagates over idealized frictionless horizontal dry-bed by solving the Saint-Venant equations (nonlinear shallow-water equations). Due to the neglect of the bottom resistance, velocity and surface profile of the wave front are different from reality. Keulegan (1950), Dressler (1952), Whitham (1955), Lauber and Hager (1998) and Chanson (2006) improved Ritter (1892) solution by considering the effect of bottom resistance, thus the calculation accuracy has been improved greatly at the wave front. Stoker (1957) extended Ritter (1892) solution to the wet-bed condition by solving the Saint-Venant equations through application of the method of characteristics. The theories above are all deduced from shallow water equations that based on the shallow water and long wave assumptions, in which the vertical velocity and acceleration are neglected and the pressure is hydrostatic. Experimental studies Martin and Moyce (1952), Dressler (1954), Stansby et al. (1998), Lauber and Hager (1998) and Çağatay and Kocaman (2008) show differences in the very initial stage and the upstream edge area between the theoretical solutions Ritter (1892) and Stoker (1957)) and real situations, as the shallow water assumptions are not valid at the upstream edge area and the solution by Stoker (1957) cannot describe the fluctuation evolution process and the corresponding wave front surface profile.

Previous experimental studies either focus on the wave front velocity and surface profile (Martin and Moyce (1952), Dressler (1954), Stansby et al. (1998), Bukreev and Gusev (2005), Oertel and Bung (2012), LaRocque et al. (2012), Jánosi et al. (2004), Lauber and Hager (1998)), or

* Corresponding author. Key Laboratory of High-speed Railway Engineering, Ministry of Education, Chengdu 610031, China
E-mail addresses: yangshaolin1116@163.com (S. Yang), 68360903@qq.com (W. Yang), qinshq@cae.cn (S. Qin), ql3721@163.com (Q. Li), yangb@home.swjtu.edu.cn (B. Yang).

https://doi.org/10.1016/j.oceaneng.2018.04.011
Received 29 September 2017; Received in revised form 7 February 2018; Accepted 7 April 2018

focus on only dry-bed case or wet-bed case Martin and Moyce (1952), Dressler (1954), Lauber and Hager (1998), Arnason (2005), Arnason et al. (2009), Oertel and Bung (2012), LaRocque et al. (2012), Lobovský et al. (2014), or focus on the very initial stage of dam-break (Jánosi et al. (2004), Lobovský et al. (2014), Bukreev and Gusev (2005)). The comparative experimental studies on dam-break wave propagation over both dry-bed and wet-bed are few, e.g. Ramsden (1996), Stansby et al. (1998), Jánosi et al. (2004), Bukreev and Gusev (2005), Çağatay and Kocaman (2008), Ozmen-Cagatay and Kocaman (2010), Kocaman and Ozmen-Cagatay (2015), mainly focus on the wave surface and wave height, while other flow field characteristics are not included.

With the rapid development of computer and CFD technology, the RANS equations can be used to simulate the whole process of dam-break wave propagation over both dry-bed and wet-bed accurately. However, numerical studies Biscarini et al. (2010), Yang et al. (2010), Ozmen--Cagatay and Kocaman (2010), Kocaman and Ozmen-Cagatay (2015) mainly focus on the validation of accuracy of numerical model by comparing with the experimental wave velocities and wave surfaces. The numerical studies on the wave characteristics are seldom, e.g. Park et al. (2012) studied the effect of turbulence intensity on the bottom shear, frictional resistance and wave front surface profile; Shigematsu et al. (2004) studied the influence of the downstream water depth on the turbulence intensity for both dry-bed and wet-bed. The difference of dam-break wave propagation over dry-bed and wet-bed (with different initial water depth) cannot be ignored, which will eventually introduce a significant influence on the wave characteristics. However, the systematical and comparative numerical studies on the influence of the downstream water depth on the wave characteristic have not been reported to our knowledge.

In general, theoretical solutions based on shallow water and long wave assumptions cannot describe the fluctuation evolution process and the corresponding wave front surface profile, previous experiments mainly focus on the wave surface and wave height, and the difference of wave propagation over dry and wet-beds are seldom compared. Numerical studies mainly focus on the verification of calculation precision but ignore the flow field analysis. So this paper aims at a systematical comparative study on the influence of the downstream water depth on the flow field by simulating the three experiments carried out by Ozmen-Cagatay and Kocaman (2010).

2. Numerical method and validation

Commercial CFD software package FLOW-3D is used to simulate the dam-break wave generation and propagation in this study as it has been widely used in hydrodynamic calculations Choi et al. (2007), Chopakatla et al. (2008), Jin and Meng (2011), Chen and Hsiao (2016) and dam-break wave simulations Bradford (2000), Ozmen-Cagatay and Kocaman (2010, 2011, 2012, 2014), Kocaman and Ozmen-Cagatay (2015).

2.1. Basic theory of numerical method

2.1.1. Governing equations

The general governing equations for Newtonian incompressible fluid flow, i.e. mass conservation and momentum conservation equations, which constitute the so called RANS equations, can be expressed as follows:

$$\frac{\partial}{\partial x_i}u_iA_i = 0 \tag{1}$$

$$\frac{\partial u_i}{\partial t}+\frac{1}{V_F}u_jA_j\frac{\partial u_i}{\partial x_j} = -\frac{1}{\rho}\frac{\partial p}{\partial x_i}+G_i+f_i \tag{2}$$

where: $i = 1, 2, 3$, and x_i represents the x, y, z coordinate respectively, u_i is the mean velocity component, A_i is the fractional area open to flow, G_i is the body acceleration, t represents time, V_F is the fractional volume open to flow, ρ is the fluid density, p is pressure, and f_i represents the viscous acceleration which can be expressed as follows:

$$f_i = \frac{1}{V_F}\left[\frac{\tau_{b,i}}{\rho}-\frac{\partial}{\partial x_j}(A_jS_{ij})\right] \tag{3}$$

where $\tau_{b,i}$ represents wall shear stress, $S_{ij} = -(\upsilon+\upsilon_T)\left[\frac{\partial u_j}{\partial x_i}+\frac{\partial u_i}{\partial x_j}\right]$ represents the strain rate tensor, υ represents kinematic viscosity, υ_T represents kinematic eddy viscosity, which can be calculated from turbulence model.

2.1.2. Turbulence model

A variety of turbulence modes are available to solve the RANS equations in FLOW-3D. The three classic turbulence models, namely standard $k-\varepsilon$ Harlow and Nakayama (1967), RNG $k-\varepsilon$ Yakhot and Smith (1992) and $k-\omega$ Wilcox (1998), are compared with the experimental data. Comparisons show that there were no obvious differences between the three turbulence models when simulating the experiments carried out by Ozmen-Cagatay and Kocaman (2010). Again the three classic turbulence models are compared with the experimental data presented by Arnason (2005), results show that with the increase of the initial upstream water depth, the $k-\omega$ Wilcox (1998) turbulence model shows better agreement than the other two turbulence models on the slope of wave front profile when the wave front arrives. So the $k-\omega$ Wilcox (1998) turbulence model is employed in this study. The detailed comparison figures are omitted here for brevity. υ_T could be determined as follows,

$$\upsilon_T = k/\omega \tag{4}$$

where $\omega \equiv \varepsilon/k$, k and ε represent turbulence kinetic energy and turbulent dissipation rate per unit mass respectively. They can be solved by the following equations:

$$\frac{\partial k}{\partial t}+\frac{1}{V_F}\left(u_iA_i\frac{\partial k}{\partial x_i}\right) = \frac{1}{V_F}\frac{\partial}{\partial x_i}\left[\left(\upsilon+\frac{\upsilon_T}{\sigma_k}\right)A_i\frac{\partial k}{\partial x_i}\right]+\frac{\upsilon_T}{V_F}\left(\frac{\partial u_j}{\partial x_i}+\frac{\partial u_i}{\partial x_j}\right)A_i\frac{\partial u_j}{\partial x_i} - \beta^*k\omega \tag{5}$$

$$\frac{\partial \omega}{\partial t}+\frac{1}{V_F}\left(u_iA_i\frac{\partial \omega}{\partial x_i}\right) = \frac{1}{V_F}\frac{\partial}{\partial x_i}\left[\left(\upsilon+\frac{\upsilon_T}{\sigma_\omega}\right)A_i\frac{\partial \omega}{\partial x_i}\right]+\alpha\frac{\omega}{k}\frac{\upsilon_T}{V_F}\left(\frac{\partial u_j}{\partial x_i}+\frac{\partial u_i}{\partial x_j}\right)A_i\frac{\partial u_j}{\partial x_i} - \beta\omega^2 \tag{6}$$

in which, $\beta^* = \beta_0^* f_{\beta^*}$, $\beta_0^* = 0.09$, $f_{\beta^*} = 1$ when $\chi_k \le 0$, and $f_{\beta^*} = \frac{1+680\chi_k^2}{1+400\chi_k^2}$ when $\chi_k > 0$, $\chi_k \equiv \frac{1}{\omega^3}\left(\frac{\partial k}{\partial x}\frac{\partial \omega}{\partial x}+\frac{\partial k}{\partial y}\frac{\partial \omega}{\partial y}+\frac{\partial k}{\partial z}\frac{\partial \omega}{\partial z}\right)$, $\sigma_k = 2.0$, $\alpha = 13/25$, $\sigma_\omega = 2.0$, $\beta = \beta_0 f_\beta$, $\beta_0 = 9/125$, $f_\beta = \frac{1+70\chi_\omega}{1+80\chi_\omega}$, $\chi_\omega \equiv \left|\frac{\Omega_{ij}\Omega_{jk}S_{ki}}{(\beta_0^*\omega)^3}\right|$, $\Omega_{ij} = \frac{1}{2}\left(\frac{\partial u_i}{\partial x_j}-\frac{\partial u_j}{\partial x_i}\right)$, $S_{ij} = \frac{1}{2}\left(\frac{\partial u_i}{\partial x_j}+\frac{\partial u_j}{\partial x_i}\right)$.

2.2. Numerical model setup and validation

The physical experiments carried out by Ozmen-Cagatay and Kocaman (2010), including dry-bed and wet-bed cases, are employed to validate the FLOW-3D numerical model. The setup of their experiments is detailed as follows. The glass water channel is 8.90 m in length, 0.30 m in width, 0.30 m in height, shown in Fig. 1. The length of the upstream and the downstream are 4.65 m and 4.25 m, respectively. In their experiments, the upstream water depth h_1 is constant as 0.25 m, while the downstream water depth h_0 is 0 m, 0.025 m, 0.1 m, which indicates the ratio of initial downstream water depth to upstream water depth, i.e. $r =$