

A preliminary investigation of numerical model for dam break wave flume

Cheng CHEN, Xiao Lei FU, Pu XU

College of Civil Engineering, Fuzhou University, Fuzhou, Fujian 350116, China

Email: chencheng_1117@163.com

Abstract. The dam break wave is a kind of broken wave, which can destroy the downstream structures. To preliminarily investigate the mechanism of the dam break wave, a numerical model was established in this study. This paper introduce the method of the numerical model and the set up of the numerical flume. The dam break numerical flume includes a reservoir, a sluice gate, a test flume, and a structure. The dam break wave was generated by suddenly lift the sluice gate, so the water in the reservoir rushed into the test flume. The generation and propagation of the dam break in the flume was simulated and observed. The time history of the pressure impact on the structure was recorded. Results show that, the dam break wave was generated by reservoir and sluice gate. In the test flume, the dam break wave stabilized, propagated, and finally engulfed the whole structure. The time history of the pressure experienced a significant peak, fluctuated line, and a quasi-steady line.

1 Introduction

Dam break wave is an important issue in hydraulic engineering. Once a dam is broken, the water in the reservoir rushed out, the water level in the upstream suddenly falls, and the water level in the downstream suddenly rises as well. As a result, the flow pattern changes, and this formation of flow fluctuation phenomenon is known as the dam break wave.

Many reserchers have focused on this topic, including case study [1-3], theory and mechanics analysis [4-7], physical and numerical experiment [8-11]. In this study, a numericam model was established, the variation of the dam break flow was observed, and the time history of the pressure due to dam break flow was investigated.

2 Methods

2.1. Governing Equations

The governing equations includes continuity equation and momentum equations, shown as follows:

Continuity equation:

$$\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} + \frac{\partial w}{\partial z} = 0 \quad (1)$$

In which, u , v , w are velocity components in three directions, respectively.

Momentum equations:

$$\frac{\partial(\rho u)}{\partial t} + \text{div}(\rho u \vec{u}) = \text{div}(\mu \text{grad} u) - \frac{\partial p}{\partial x} + S_x \quad (2)$$

$$\frac{\partial(\rho v)}{\partial t} + \text{div}(\rho v \vec{u}) = \text{div}(\mu \text{grad} v) - \frac{\partial p}{\partial y} + S_y \quad (3)$$

$$\frac{\partial(\rho w)}{\partial t} + \text{div}(\rho w \vec{u}) = \text{div}(\mu \text{grad} w) - \frac{\partial p}{\partial z} + S_z \quad (4)$$



In which, p is the pressure for fluid fraction; S_x , S_y , S_z are source terms for momentum equations.

2.2. Boundary Conditions

The boundary conditions in this model can be expressed as follows:

Kinematic boundary conditions:

$$\frac{\partial \eta}{\partial t} + \nabla \phi \cdot \nabla \eta = \frac{\partial \phi}{\partial z} \quad (5)$$

In which, $Z = \eta(x, y, t)$ is free surface equation; ∇ is horizontal gradient, which can be written as:

$$\nabla = \left(\frac{\partial}{\partial x}, \frac{\partial}{\partial y} \right) \quad (6)$$

Dynamic boundary conditions:

$$\frac{\partial \phi}{\partial t} + \frac{1}{2} \left(|\nabla \phi|^2 + \left(\frac{\partial \phi}{\partial z} \right)^2 \right) + g\eta = 0 \quad (7)$$

2.3. Meshes and Grids

In the numerical model, the meshes and grids are basically uniform, which could perform good simulating results if they are chosen a suitable size. However, it is a waste of time and computation work to produce a good result in uninterested areas. Therefore, the unnecessary computation can be reduced by using the following methods: (a) partly densified method; (b) mesh stitching method; (c) grid nesting method. Three methods are shown in Figure 1.

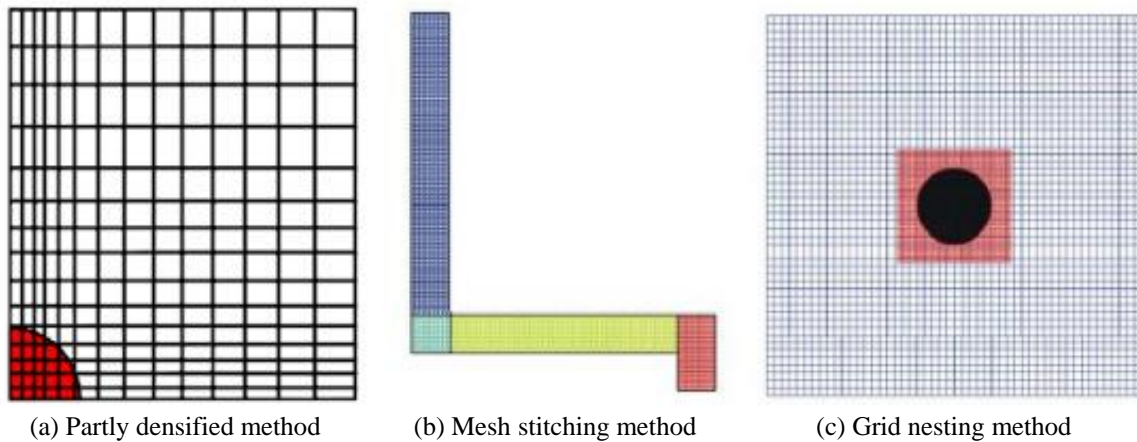


Figure 1 Three meshes and grids methods in the numerical model

2.4. Volume of Fluid

In the numerical model, volume of fluid (VOF) method is used to track the free surface of the fluid (shown in Figure 2). The value of VOF function (F) is between 0 and 1. Specifically, $F=0$ is air, and $F=1$ is fluid.

Transport equation can be written as:

$$\frac{DF}{Dt} = 0 \quad (8)$$

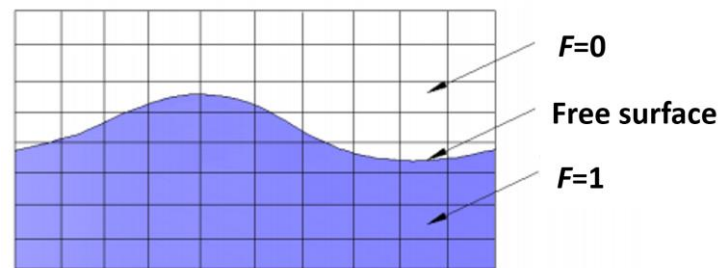


Figure 2 Use the volume of fluid function to track the free surface, the value of F is between 0 and 1

3 Model Set-up

3.1. Establishment of 3D Numerical Model

The generation of the dam break wave was based on a flume system, including a reservoir, a sluice gate, a test flume, and a structure. In this numerical model, a reservoir was set to store a certain amount of water. Once a sluice gate lifted, the water rush into the test flume and generated a dam-break flow. As a result, the dam break flow in this study was simulated.

The establishment procedure of the dam break wave flume is as follows:

- (1) Define the units of the variable as international standard units. Define unit of the water temperature as degree Celsius. Define the simulation time as 8 seconds, which is enough for dam break wave generation and propagation;
- (2) Set the gravitational acceleration as 9.8 m/s^2 ;
- (3) Chose a fluid from the database. In this study, chose the water in 20 degree Celsius;
- (4) Geometry settings and mesh division. The density of mesh is closely related to the stability and accuracy of numerical simulation, so meshing is very important.
- (5) Data input and output. The appropriate data output time is chosen to facilitate the observation and analysis of experimental data.
- (6) Set the density of the grid. In this study, the density is 0.02.

The three-dimensional flume in this paper is set up as follows: the first step is to set a three-dimensional flume with 22 m in x direction, 0.2 m in y direction, and 0.8 m in z direction. The grid density in the three directions of x, y, and z is 0.02. Therefore, this numerical model sets up a total of 440000 grids in the computational domain of the flume. The structure of the flume is divided into three parts, namely the reservoir area, the test flume, and the structure area, as shown in Figure 3. The position of the sluice gate is 0 in x axis, which cover the full width of the test flume. The height of the sluice gate is 0.8 m. The length of the reservoir area in the x direction is 12 m, the length of the test flume is 9 m, and the length of the structure area is 1 m. The grids number in the reservoir area is 240000, the grid number in the sluice gates is 20000, and the grid number in the structure area is 20000. The structure area is composed of two plates, namely a flat deck and a slope. The position of the flat deck is 9 m in x axis, and its thickness is 0.1 m. The flat deck is 0.4 m height over the bed, and the slope is 30 degree.

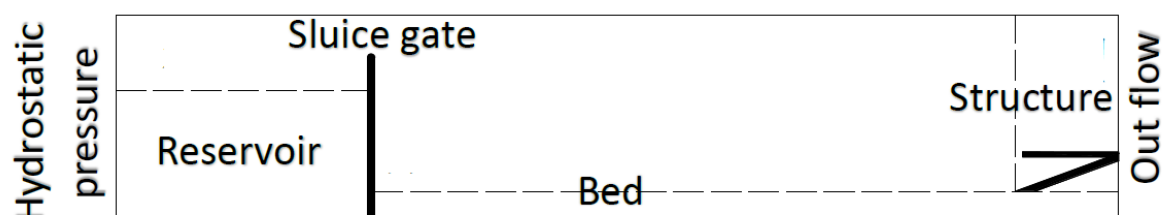


Figure 3 Overview of the numerical model

3.2. Boundary Conditions

There are 6 boundaries in this numerical model, i.e. 2 boundaries in x direction (left and right in Figure 3), 2 boundaries in y direction (front and back in Figure 3), and 2 boundaries in z direction (top and bottom in Figure 3). The left boundary is specified a hydrostatic pressure, the right boundary is out flow. The front and back boundaries are symmetric. The top boundary is the out flow, while the bottom boundary is selected as a wall.

3.3. Movement of the Sluice Gate

The sluice gate has a great effect on the wave form, wave height, and wave speed. For example, a larger wave height and higher wave speed are resulted from a larger gate opening. The steeper wave front is resulted from a faster lift speed of the sluice gate. Therefore, the time history of the lift speed of the sluice gate is shown in Figure 4.

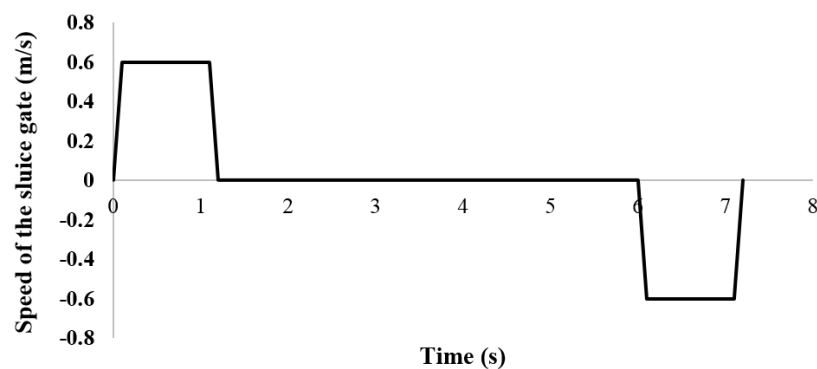


Figure 4 Time history of the lift speed of the sluice gate

4 Results and discussions

4.1. Dam break wave in the flume

Figure 5 shows the variations of the fluid in the numerical flume. Firstly, the sluice gate lifted up and the water rushed into the test flume. Thus the dam break wave was generated. Then the dam break wave became stablized, and keep moving foward. The wave propagate in the test flume. When $t = 3.5$ s, the wave began to hit the structure. After $t = 4.5$ s, the water accumulated in front of the structure. Finally, the structure was fully submerged by the dam break wave.

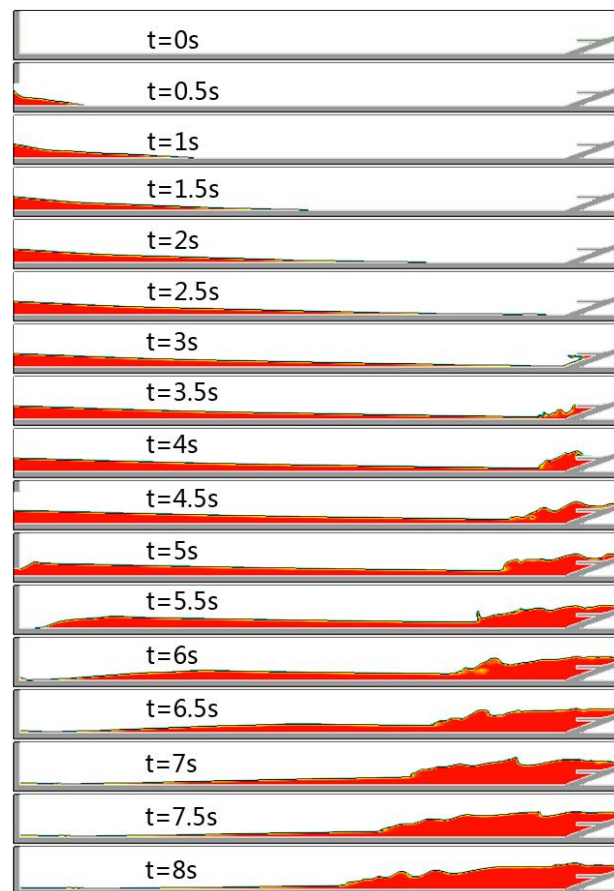


Figure 5 Variations of the fluid in the numerical flume

4.2. Time history of the pressure on the structure

Figure 6 shows the pressure exerted on the structure due to the dam break wave. As figure shows, before $t = 3$ s, the dam break wave has not hit the structure yet. After $t = 3$ s, there is a significant peak in the time history of the pressure. After that, the pressure line fluctuated significantly. Finally, it became stabilized.

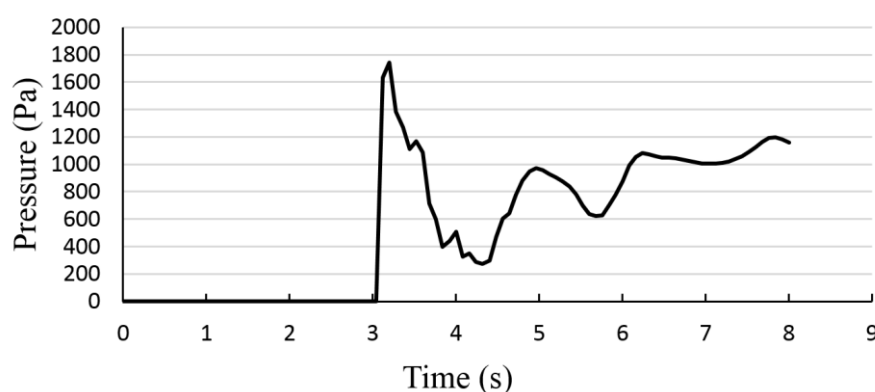


Figure 6 Time history of the pressure exerted on the structure

5 Conclusion

This paper preliminarily investigates the dam break wave flume based on a numerical model. The method of the numerical model was introduced. The numerical model was established, including a

reservoir, a sluice gate, a test flume, and a structure. The dam break wave was generated in the numerical flume. The variations of the fluid in the flume was presented, and the pressure exerted on the structure was investigated. Results show that, the dam break wave rushed into the test flume, stabilized, propagated, and finally engulfed the structure. The pressure line shows a peak, then fluctuated, and finally stabilized.

Acknowledgement

This paper is funded by Fuzhou University scientific start-up expenses (No. GXRC-17012).

References

- [1] Valiani A, Caleffi V, Zanni A. Case Study: Malpasset Dam-Break Simulation using a Two-Dimensional Finite Volume Method[J]. *Journal of Hydraulic Engineering*, 2002, 128(5):460-472.
- [2] Gallegos H A, Schubert J E, Sanders B F. Two-dimensional, high-resolution modeling of urban dam-break flooding: A case study of Baldwin Hills, California[J]. *Advances in Water Resources*, 2009, 32(8):1323-1335.
- [3] Pilotti M, Maranzoni A, Tomirotti M, et al. 1923 Gleno Dam Break: Case Study and Numerical Modeling[J]. *Journal of Hydraulic Engineering*, 2011, 137(4):516-520.
- [4] Hubert Chanson. Application of the method of characteristics to the dam break wave problem[J]. *Journal of Hydraulic Research*, 2009, 47(1):41-49.
- [5] Wang B, Chen Y, Wu C, et al. Analytical solution of dam-break flood wave propagation in a dry sloped channel with an irregular-shaped cross-section[J]. *Journal of Hydro-environment Research*, 2017, 14:93-104.
- [6] Capart H, Young D L. Formation of a jump by the dam-break wave over a granular bed[J]. *Journal of Fluid Mechanics*, 1998, 372(372):165-187.
- [7] Castro-Orgaz O, Chanson H. Ritter's dry-bed dam-break flows: positive and negative wave dynamics[J]. *Environmental Fluid Mechanics*, 2017:1-30.
- [8] Haltas I, Elçi S, Tayfur G. Numerical Simulation of Flood Wave Propagation in Two-Dimensions in Densely Populated Urban Areas due to Dam Break[J]. *Water Resources Management*, 2017:1-23.
- [9] Ye Z, Zhao X. Investigation of water-water interface in dam break flow with a wet bed[J]. *Journal of Hydrology*, 2017.
- [10] Labine A M. An experimental investigation of pressure of a simple dam break generated wave impacting a plate[J]. *Journal of Endocrinology*, 2015, 54(2):317-25.
- [11] Huang J C, Xiao-Yan H E. Unified 2-D numerical model for simulating dam break wave propagation[J]. *Journal of Hydraulic Engineering*, 2006, 37(2):222-226.