Numerical and Experimental Investigation of Dam-Break Flow Against a Vertical Cylinder

Changhong Hu*, Mohamed M. Kamra, Seiya Watanabe
Research Institute for Applied Mechanics, Kyushu University, JAPAN
*hu@riam.kyushu-u.ac.jp

1 INTRODUCTION

The objective of this research is to develop a high-fidelity numerical method capable of accurate prediction of violent wave impact flows against complicated geometries and its underlying physical phenomena. Novel computational fluid dynamic (CFD) methods have been studied and several CFD solvers have been developed in our laboratory for this purpose. Recently an unstructured mesh finite volume method (FVM) solver and a lattice Boltzmann method (LBM) solver, have been developed [1][2]. In the FVM solver, UMTHINC method is applied as the interface capturing scheme for free surface; while for the LBM solver, the cumulant model is adopted as the collision model and the phase field method is used as the interface capturing scheme.

In order to provide accurate and reliable measurement data for validation of the CFD codes on the free surface impact prediction, we have carried out a new experiment of the dam break impact on a vertical cylinder placed over a dry horizontal bed [3]. The conducted experiment aims to provide detailed description of the kinetic and dynamics of the dam break impact with a vertical cylinder. Special attention is directed to the existing uncertainties in the dam break experiment especially the gate motion. A new motion profile is proposed to fit the gate motion.

The performance of the two in-house CFD codes has been investigated by performing comparison simulations against the newly conducted dam break experiment.

2 DAM-BREAK EXPERIMENT

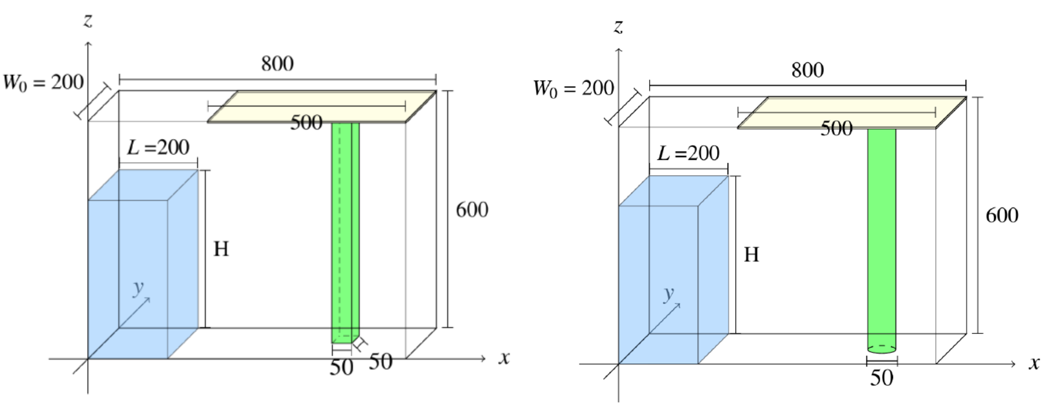
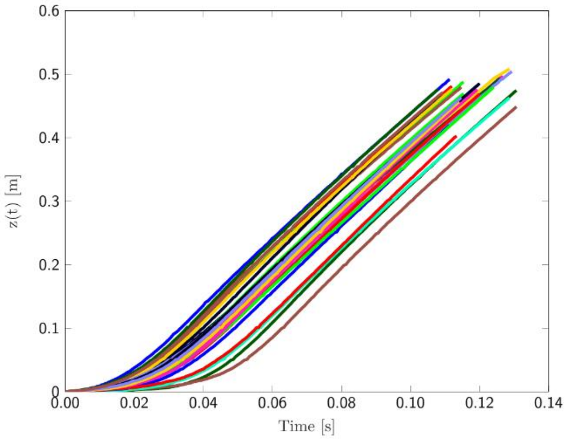


Fig. 1 Schematic drawing of the dam-break experiment with a square cylinder (left) and a circular cylinder (right). All dimensions are in mm.

The experimental setup is shown in Fig.1, which is based on a series of experiments conducted in our labortary between the years 2010 and 2017. Two kinds of vertical cylinder, with square and circular cross-section respectively, are used in the experiment. A total of 28 experiment runs are conducted as illustrated in Table 1. Pressure measurements are performed by two pressure sensors placed on the cylinder surface and on the downstream vertical wall. Free surface variations are recorded by a high-speed digital video camera system.

The gate motion of the experiment has been studied and a gate motion formula is proposed which better fits the experimental data. The gate motion for all 28 runs are presented in Fig. 2 and a gate motion formula based on the experimental data is proposed as shown by Eq. (1). The proposed motion profile suggests a variable acceleration during the initial acceleration stage. The effect of gate obstruction on the experimental measurements is studied. It is found that the time of water impact with the cylindrical obstacle is clearly correlated to the parameters of the proposed motion profile, which means that the effect of gate motion cannot be neglected in the study of dam break flow.



$$z(t) = \begin{cases} a\,t\,e^{b\,t} & t \le t_0 \\ a\,t_0\,e^{b\,t_0} + v_0\,(t - t_0) & t > t_0 \end{cases}$$

$$a = \frac{v_0}{(1 + b\,t)e^{b\,t}} \qquad (1)$$

Fig.2 Gate motion profile measurements

Table 1 Contents of experiment

Experiment Type	Number of Trial Runs
No obstacle case	9
Square obstacle case	10
Circular obstacle case	9

3 TWO NEWLY DEVELOPED CFD SOLVERS

Unstructured Mesh FVM

This in-house code is a Reynolds averaged incompressible Naiver-Stokes equations (RANS) solver coupled with the volume of fluid (VOF) algorithm. The three-dimensional incompressible turbulent flow is solved using the cell-centered finite volume method based on second order discretization in space and time. Unstructured grids are used, in which the grids may consist of orthogonal quadrilateral/hexahedral cells near the boundaries for accurate boundary layer treatment and triangles/prism cells elsewhere.

Interface capturing is performed using the unstructured multi-dimensional tangent hyperbolic interface capturing (UMTHINC) scheme [4]. The UMTHINC method is a VOF method where the interface jump is defined as a smoothed piecewise hyperbolic tangent profile, so the indicator color function $C(\mathbf{x})$ is approximated as

$$C(\mathbf{x}) = 0.5\left(1 + \tanh\left(\beta\left(P(\mathbf{x}) + d\right)\right)\right) \qquad (2)$$

where β is the sharpness parameter which controls how steep is the interface region between the two fluids, and the interface surface equation is defined by $P(\mathbf{x}) + d = 0$. Depending on the interface treatment, $P(\mathbf{x})$ can be a second order polynomial, providing high accuracy and better interface faithfulness, or a first order polynomial which is basically a linear plane. The coefficients of the polynomial $P(\mathbf{x})$ are based on the interface geometric characteristics such as the surface unit normal vector $\mathbf{n}$ and curvature matrix I_{xyz}. Typically reconstruction is done for interface cells that is identified using the volume fraction value (α) with interface cells having $\alpha \in [\epsilon, 1-\epsilon]$ with ϵ set to 10^{-8} in this work. A major advantage of the UMTHINC scheme is that it combines the accuracy of geometric reconstruction-based methods with the simplicity and efficiency of algebraic based VOF methods.

Cumulant Lattice Boltzmann Method

This in-house code is a lattice Boltzmann method solver for incompressible flows in which the velocity distribution function on a lattice is calculated with streaming and collision (relaxation) processes. The time evolution equation of the velocity distribution function f_{ijk} is as follows.

$$f_{ijk}\left(\mathbf{x}+\xi_{ijk}\Delta t, t+\Delta t\right)=f_{ijk}\left(\mathbf{x}, t\right)+\Omega_{ijk}\left(\mathbf{x}, t\right) \tag{3}$$

where, ijk is the speed direction of the distribution function $(i,j,k) \in \{-1,0,1\}^3$. $\boldsymbol{\xi}$ is the speed of the distribution function defined as $\boldsymbol{\xi}_{ijk}=(ic,jc,kc)^T$ using $c=\Delta x/\Delta t$. The fluid density and the fluid velocity are then calculated from the distribution functions as

$$\rho=\sum_{ijk} f_{ijk}, \quad \mathbf{u}=\frac{1}{\rho}\sum_{ijk}\xi_{ijk} f_{ijk} \tag{4}$$

To treat high Reynolds number flows such as dam-breaking problems, the cumulant model [5] is applied to the collision term $\Omega_{ijk}(\boldsymbol{x},t)$, in which the velocity distribution function is non-linearly transformed into the statistic cumulant, and the collision term is calculated using the cumulant.

Variation of the free surface is solved by a phase field method. The Allen-Cahn equation is used for the time evolution of ϕ.

$$\frac{\partial \phi}{\partial t}+\nabla\cdot(\mathbf{u}\phi)\rho=\nabla\cdot\left[M\left(\nabla\phi-\frac{1-4(\phi-0.5)^2}{W}\mathbf{n}\right)\right] \tag{5}$$

where, ϕ is the phase field function, W the interface thickness, and M the mobility parameter. $\mathbf{n}$ is the unit normal vector of the interface, which is calculated from the signed distance function of the interface. $\phi=1$ means the liquid phase; $\phi=0$ the gas phase; and $0<\phi<1$ denotes the interface.

4 RESULTS

Numerical simulation on the experiment cases has been carried out by using the above described two CFD solvers. The performance of the numerical methods is studied by comparing the numerical results with the experiments. In this section only a couple of results are presented and detailed discussion on the numerical simulations will be presented at the workshop.

In the FVM simulation of the dam break experiment, a simplified gate model has been developed to consider the effect of the gate motion. The numerical solutions are compared with the experiments and fairly good agreement is obtained in both pressure and free surface. The effect of turbulence models has been investigated for the case of circular cylinder. It is demonstrated that a proper choice of turbulent model has very important effect in obtaining accurate predictions for separated flows. As an example, in Fig. 3 a comparison of the pressure on the cylinder surface between the FVM result and the experimental measurement is shown.

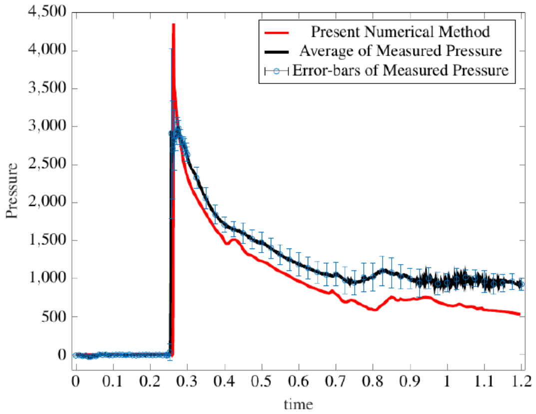


Fig. 3 Comparison of pressure on the cylinder

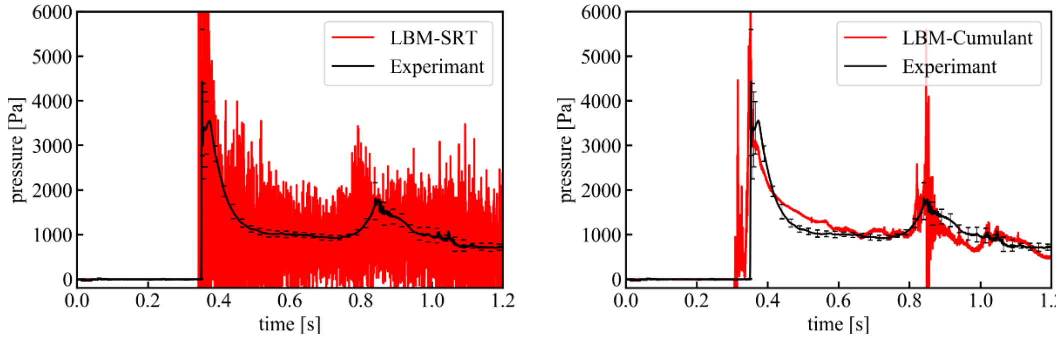


Fig.4 Pressure on the downstream vertical wall (left: LBM-SRT, right: LBM-cumulant).

In the numerical simulation with the LBM solver, extensive investigations on the effect of collision models, turbulence models, and mesh resolutions have been carried out. As an example of the results, we compare the LBM results with the experiment of the pressure on the downstream vertical wall in Fig. 4. By comparing to the experiment, it is found that the pressure oscillation can be largely suppressed by the cumulant model. By further improvement it is possible to apply the LBM solver with the cumulant model to prediction of free surface impact problems.

REFERENCES

[1] Mohamed M. Kamra, Development of an Unstructured Grid Solver for Complex Wave Impact Problems, PhD Thesis, Kyushu University 2018

[2] Seiya Watanabe, Implementation of AMR method to lattice Boltzmann method for large-scale GPU simulation of multiphase flow, PhD Thesis, Tokyo Institute of Technology, 2019

[3] Mohamed M. Kamra, Jabir Al Salami, Makoto Sueyoshi, Changhong Hu, Experimental study of the interaction of dambreak with a vertical cylinder, Journal of Fluids and Structures, V. 86, pp. 185-199 (2019)

[4] S. Ii, B. Xie, F. Xiao, An interface capturing method with a continuous function: The THINC method on unstructured triangular and tetrahedral meshes, Journal of Computational Physics 259 (2014) 260–269.

[5] Geier, M., Schönherr, M., Pasquali, A., & Krafczyk, M., The cumulant lattice Boltzmann equation in three dimensions: Theory and validation. Computers &Mathematics with Applications, Vol. 70(4) (2015) pp. 507-547.