Proceedings of the World Congress on Engineering 2013 Vol III,
WCE 2013, July 3 - 5, 2013, London, U.K.

Simulating Dam-Break Flooding with Floating Objects through Intricate City Layouts Using GPU-based SPH Method

Jiansong Wu, *Member, IAENG,* Hui Zhang, and Robert A. Dalrymple

***Abstract*—For the fast transient dam break flooding with floating bodies presented through intricate city layouts, the traditional grid-based method based on solving two dimensional (2D) Shallow Water Equations or three dimensional (3D) Reynolds-averaged Navier-Stokes equations have difficulty in modelling the 3D unsteady flow features and the moving objects in the flow, causing inaccuracies. In this paper, the fully Lagrangian meshfree Smoothed Particle Hydrodynamics (SPH) method with the graphical processing unit parallel computing technique employed (GPUSPH) is therefore applied to simulate the dam-break flood through the intricate urban district and to implement floating objects in the flow. Taking advantage of GPUs parallel computing techniques, simulations involving millions of particles (computational nodes) can be achieved. Numerical results identify the complex 3D flow features, such as hydraulic jumps, wave vortices and flow discontinuities, which indicates SPH method is well-suited for the modeling of dam-break flood through urban areas and the complicated flooding flow involving fluids with interacting floating objects.**

***Index Terms*— city layouts, flooding, floating objects, SPH**

I. Introduction

In the past decades, some catastrophic flooding events which were caused by the breaking of a dam or a dike, or a flash flood after an exceptional rainfall were really dramatic: Nîmes, France (1988); New-Orleans, USA (2005); Jinan, China (2007); Vicksburg, USA (2011); Quebec Canada (2011); Bangkok, Thailand (2011); Beijing, China (2012); New York City, USA (2012). As a result, more and more researchers around the world have been working on numerical modeling of urban flooding by taking advantage of the rapid development of advanced computing technology these years.

In the numerical modeling of dam-break flooding, two dimensional (2D) depth-averaged shallow water equations (SWEs) models have been playing a dominant role (Zoppou et al. (1999) [1], Ishigaki et al. (2004) [2], Mignot et al. (2006a,2006b) [3,4], McMillan et al. (2007) [5], Soares-Frazão et al. (2008) [6], Abderrezzak et al. (2009) [7], Delelegn et al. (2011) [8], Schubert et al. (2012) [9]). The SWEs are derived through the depth-averaged integration under the underlying assumption of hydrostatic pressure distribution. These simplified 2D models are reasonable to simulate flood propagation in straight streets and some near-street intersections. However, in the city layouts with extremely intricate network of streets, these 2D models will cause inaccuracies, especially for the fast transient dam break flooding flows (demonstrated by Soares-Frazão et al. (2008) [6] with the computed unreasonable velocity field at crossroads in an idealized city layout, and also verified by Abderrezzak et al. (2009) [7] finding unreasonable water depth near an isolated building), because the intricate layout of buildings and infrastructural constructions in urban areas can induce more complex flow features, such as hydraulic jumps and flow discontinuities. These local 3D features make the underlying assumptions of the 2D depth-averaged models invalid. Moreover, it is not straightforward to simulate extreme fluid flows motion with body interaction by Eulerian 2D SWEs or 3D Reynolds-averaged Navier-Stokes methods as explicit special treatment of the free surface locations is required(Watts (2000) [10]; Monaghan et al. (2003) [11]; Yim et al. (2008) [12]).

SPH is a fully Lagrangian technique for Computational Fluid Dynamics simulations originally developed to solve astrophysical problems in 3D open space by Lucy (1977) [13] and Gingold and Monaghan (1977) [14]. In SPH method, each particle carries its own physical quantity, such as mass, position, velocity and internal energy. The most attractive advantage of using SPH over Eulerian meshed methods is that no special treatment of the free surface is required which is quite difficult for simulating highly unsteady and nonlinear flows with splashing around structures and interacting with moving bodies. Monaghan et al. (2003) [11] firstly applied SPH method to simulate the impacts between water and a rigid body. Manenti et al. (2008) [15] and Rogers et al. (2009) [16] investigated the behavior of a floating box in periodic waves and in a surf zone, respectively. Omidvar et al. (2012) [17] investigated wave body interaction problems using a variable particle mass distribution based on an arbitrary Lagrange-Eulerian formulation. All the research work above on SPH modeling of the fluid flows with interacting floating objects is

Manuscript received March 23, 2013; revised July April 13, 2013. This work partially supported by the National Basic Research Program of China (973 Program No: 2012CB719705), National Natural Science Foundation of China (Grant No. 70833003, 91024032).

Jiansong Wu is with the Institute of Public Safety Research, Engineering Physics Department, Tsinghua University, Beijing, 100084 CHINA (phone: (+86-10)6279-2894; fax: (+86-10)6278-2863; e-mail: jiansongwu@hotmail.com).

Hui Zhang was with the Institute of Public Safety Research, Department of Engineering Physics, Tsinghua University, Beijing, 100084 CHINA (e-mail: zhhui@tsinghua.edu.cn).

Robert A. Dalrymple is with the Civil Engineering Department, Johns Hopkins University, Baltimore, MD 21218 USA(e-mail: rad@jhu.edu).

satisfactory. Therefore, in this paper, SPH method is used to simulate the dam-break flood through the intricate urban district and to implement floating objects in the flooding flow. Moreover, in order to conduct high-fidelity modeling of local features of dam break flow and accelerate the speed of relatively expensive SPH computation, we employ the graphical processing unit (GPU) parallel computing technique available in the open source model GPUSPH (www.ce.jhu.edu/dalrymple/GPUSPH). Fast transient dam break flows in two different city layouts are examined: (1) an idealized square city layout of 5×5 buildings in both horizontal and oblique direction, and (2) an intricate urban district with complex buildings and infrastructural constructions.

II. Methodology

A. SPH Fundamentals

The SPH formulations of governing Partial Differential Equations are generally derived through two key steps: the integral representation of field functions and the particle approximation [18]. The first integral representation of field functions is represented as follows

$$f(\vec{x}) = \int_{\Omega} f(\vec{x}')W(\vec{x}-\vec{x}',h)d\vec{x}' \tag{1}$$

$$\nabla \cdot f(\vec{x}) = -\int_{\Omega} f(\vec{x}')\cdot \nabla W(\vec{x}-\vec{x}',h)d\vec{x}' \tag{2}$$

where $f(\vec{x})$ is a function of the position vector $\vec{x}$, $W(\vec{x}-\vec{x}',h)$ is the smoothing function which is approximated to the Dirac delta function $\delta(\vec{x}-\vec{x}')$ when h is close to zero, h is the smoothing length determining the influence area of the smoothing function, and $\nabla \cdot f(\vec{x})$ is the spatial derivative of field function.

The second particle approximation is to convert the continuous integral functions to discretized forms of summations with a finite number of particles that carry individual physical quantity:

$$f(\vec{x}_i) = \sum_{j=1}^{N} \frac{m_j}{\rho_j} f(\vec{x}_j)\cdot W_{ij} \tag{3}$$

$$\nabla \cdot f(\vec{x}_i) = \sum_{j=1}^{N} \frac{m_j}{\rho_j} f(\vec{x}_j)\cdot \nabla_i W_{ij} \tag{4}$$

with $W_{ij} = W(\vec{x}_i - \vec{x}_j, h) = W(|\vec{x}_i - \vec{x}_j|, h)$,

$$\nabla_i W_{ij} = \frac{\vec{x}_i - \vec{x}_j}{r_{ij}} \frac{\partial W_{ij}}{\partial r_{ij}} = \frac{\vec{x}_{ij}}{r_{ij}} \frac{\partial W_{ij}}{\partial r_{ij}}$$

where N is the number of particles in the support domain of particle i, r_{ij} is the distance between particle i and j, ρ_j is the density associated with particle j, and m_j is the mass associated with particle j. W_{ij} is the smoothing function of particle i evaluated at particle j, and is closely related to the smoothing length h.

B. SPH Formulations

For Newtonian fluid flows, the viscous shear stress should be proportional to the shear strain rate. The governing Navier-Stokes equations with laminar viscous stresses can be written in Lagrangian description as follows:

$$\frac{D\rho}{Dt} = -\rho \nabla \cdot \vec{v} \tag{5}$$

$$\frac{D\vec{v}}{Dt} = -\frac{1}{\rho}\nabla p + \vec{g} + \nu_0 \nabla^2 \vec{v} \tag{6}$$

where $\vec{v}$ is the velocity vector, p is the pressure, $\vec{g}$ is the gravitational acceleration. While in order to handle and capture the turbulence effects in the flow, the Large Eddy Simulation turbulence scheme for the sub-particle scale(SPS) flow is obtained by adding a flattop spatial filter on equation (5) and (6) by Lo and Shao (2002) [19]

$$\frac{D\rho}{Dt} = -\rho \nabla \cdot \overline{\vec{v}} \tag{7}$$

$$\frac{D\overline{\vec{v}}}{Dt} = -\frac{1}{\rho}\nabla \overline{p} + \vec{g} + \nu_0 \nabla^2 \overline{\vec{v}} + \frac{1}{\rho}\nabla \cdot \vec{\tau} \tag{8}$$

The SPS approach above to model turbulence was originally presented by Lo and Shao (2002) [19] in their incompressible SPH scheme. Dalrymple and Rogers (2006) [20] firstly introduced SPS model to the weakly compressible SPH scheme with the derived discretized forms based on the SPH approximations mentioned in Section 2.1 as follows:

$$\frac{d\rho_i}{dt} = \sum_{j=1}^{N} m_j (\vec{v}_i - \vec{v}_j)\cdot \nabla_i W_{ij} \tag{9}$$

$$\frac{d\vec{v}_i}{dt} = -\sum_{j=1}^{N} m_j \left(\frac{p_i}{\rho_i^2} + \frac{p_j}{\rho_j^2}\right)\nabla_i W_{ij} + \vec{g} + \sum_{j=1}^{N} m_j \left(\frac{4\upsilon_0 \vec{x}_{ij}\cdot \nabla_i W_{ij}}{(\rho_i + \rho_j) r_{ij}}\right)\vec{v}_{ij} + \sum_{j=1}^{N} m_j \left(\frac{\tau_i}{\rho_i^2} + \frac{\tau_j}{\rho_j^2}\right)\nabla_i W_{ij} \tag{10}$$

The particles are moved based on the velocity:

$$\frac{d\vec{x}_i}{dt} = \vec{v}_i \tag{11}$$

The SPH method introduced an artificial compressibility to use a quasi-incompressible equation of state to implement the incompressible flow. The following equation of state for water to model free surface flows presented by Monaghan (1994) [21] has been widely used by many applications of SPH.

$$p = B\left\{\left(\frac{\rho}{\rho_0}\right)^{\gamma} - 1\right\} \tag{12}$$

where $\gamma = 7$ is used in most circumstances, ρ_0 is the reference density. B is a problem dependent parameter, which is always taken as the initial pressure p_0.

C. Motion of Floating Objects

The scheme to implement the floating objects in dam-break flooding is referred to the motion equations of moving rigid bodies proposed by Monaghan et al. (2003) [11] and Monaghan (2005) [22]. The motion of rigid objects is determined through the motion and the rotation of the center of mass. The equations of the motion and rotation of the center of mass is given by

$$M\frac{d\vec{V}}{dt} = \vec{F} \tag{13}$$

$$I\frac{d\vec{\Omega}}{dt} = \vec{\mathrm{T}} \tag{14}$$

ISBN: 978-988-19252-9-9
ISSN: 2078-0958 (Print); ISSN: 2078-0966 (Online)

where M is the mass of rigid objects, $\vec{V}$ is the velocity of the center of mass, $\vec{F}$ is the total force adding on the rigid bodies, I is the moment of inertia, $\vec{\Omega}$ is the angular velocity of the center of mass, $\vec{T}$ is the total torque about the center of mass.

The rigid object is represented by a number of boundary particles, and thus the movement of each boundary particle is part of the motion of the rigid object. These boundary particles interact with fluid particles, and the summation of the forces adding on each boundary particle by fluid particles is the total forces adding on the rigid object, so equations (13) and (14) become

$$M\frac{d\vec{V}}{dt}=\sum_{k=1}^{N_{BP}}m_k\vec{f}_k \tag{15}$$

$$I\frac{d\vec{\Omega}}{dt}=\sum_{k=1}^{N_{BP}}m_k(\vec{x}_k-\vec{x}_0)\times\vec{f}_k \tag{16}$$

where N_{BP} is the total boundary particles of the rigid body, $\vec{f}_k$ is the force per unit mass on boundary particle k, m_k is the mass of boundary particle k, $\vec{x}_0$ is the displacement of the center of mass. The velocity of boundary particles is given by

$$\vec{v}_k=\frac{d\vec{x}_k}{dt}=\vec{V}+\vec{\Omega}\times(\vec{x}_k-\vec{x}_0) \tag{17}$$

The force adding on each boundary particle is calculated by summing up the contribution from all the surrounding fluid particles within the support kernel, which is given by

$$\vec{f}_k=\sum_{i=1}^{N_{WP}}\vec{f}_{ki} \tag{18}$$

where N_{WP} is the total surrounding fluid particles of boundary particle k, $\vec{f}_{ki}$ is the force per unit mass exerted by fluid particle i on boundary particle k. To ensure that linear and angular momentum conservation of the entire particle system in the absence of external forces, the force on boundary particle k due to fluid particle i must be equal and opposite to the force on i due to k, so

$$m_k\vec{f}_{ki}=-m_i\vec{f}_{ik} \tag{19}$$

This is very useful as during the computation we only actually compute repulsive force, which is calculated based on the modified version of boundary repulsive force by Monaghan and Kos (1999) [23].

The integrations of Equations (15) ~ (17) with time can track and predict the motion of floating bodies.

D. GPUSPH Program

SPH is computationally expensive to compute the numerous particle interactions for some large-scale applications. So the acceleration of the SPH using parallel computing technique is really necessary for achieving the widely-used engineering practice. The powerful parallel nature of GPUs makes them pretty well-suited tools for advanced scientific modeling. After the CUDA programming language introduced by Nvidia in the spring of 2007, it is really readily accessible for engineers and scientists to use the GPU by extending the C++ language to handle the computing operations of the GPU and its interfacing with the CPU host. GPUSPH, developed by Herault and Dalrymple (2010) [24], is programmed in CUDA, C++ and OpenGL based on the SPH methodologies mentioned above. With OpenGL, GPUSPH can display computing results real-time. Now it is an open source code and updated to Version 2.0. For more details you can download the GPUSPH code and the guide from this link: www.ce.jhu/dalrymple/GPUSPH.

III. Results and Discussion

In order to validate the ability of SPH method to deal with complex free surface evolution with extreme interface motion and fully analyze the effects for simulating the 3D local hydraulic features, we examined the dam break flow through an idealized city layout based on the laboratory experiments which was conducted by Soares-Frazão and Zech (2008) [6]: a square city layout of 5×5 buildings in both horizontal direction and oblique direction.

Our numerical testing platform is a workstaion rack mounting 4×Tesla C2075 cards on as many 2nd generation PCI-Express slots. The system is based on a 2×Intel Xeon X5675 processor with 12 total cores totally (3.06 GHz, 12 MB cache) and 96 GB RAM. Each Tesla C2075 has 480 CUDA cores grouped in 15 multiprocessors, 48KB shared memory per MP, 6.0 GB global memory. The operating system is CentOS 6.3 with CUDA runtime 4.2 installed.

A. Dam-break Flooding through An Idealized City Layout

The set-up of the experiment is shown in Fig. 1. The channel with a horizontal concrete bed was 35.8 m long and 3.6 m wide. A rectangular gate of 1.0 m width was located between two fixed impervious blocks. The idealized square city layout was consisted of 5×5 buildings, and located in two arrangements: parallel to or an orientation turned 22.5° relative to the approach flow direction. The size of wooden blocks representing “building” was 0.30 m×0.30 m, and the width of “streets” was 0.10 m. The initial reservoir water level was 0.40 m, and the downstream reach after the gate was wetted with a thin layer of 0.011m water.

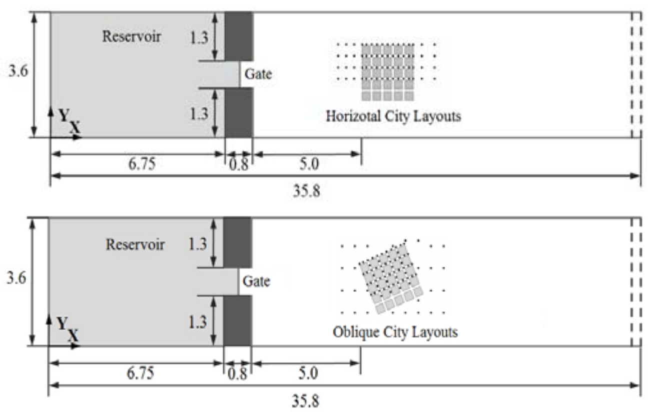
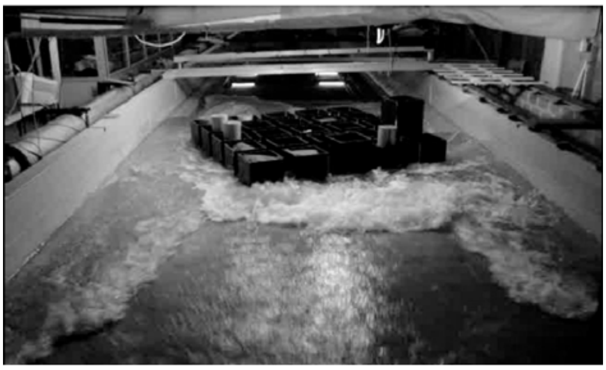


Fig. 1 Set-up of the experiment by Soares-Frazão and Zech (2008) [6]

Our numerical simulation used 1 GPU. The particle spacing that we initialized the two arrangements of city layouts with was the same as 0.018m, and the arrangement for the horizontal building layout achieved water particles and obstacle particles totally up to 2,352,864 and the arrangement for the oblique building layout achieved total particles up to 2,334,155. The total particles of two arrangements of city layouts are different as wet bottom particles in the oblique city layout are fewer because of its unstructured initialization. We used the variable time step in SPH scheme with an initial time step (dt = 0.00003 s).

We mainly discuss the typical instances of the flow propagation that presents the most complex hydraulic features. Observations from the snapshot of the lab experiments (see Fig. 2) showed that the flow rises at the city front after the wave impact against the buildings, and a hydraulic jump formed in front of the buildings with the water level locally higher than without the presence of the buildings. And also, hydraulic jumps formed near the sidewall due to the wave impact against the sidewall and reflection by the sidewall.

(a) Horizontal layouts

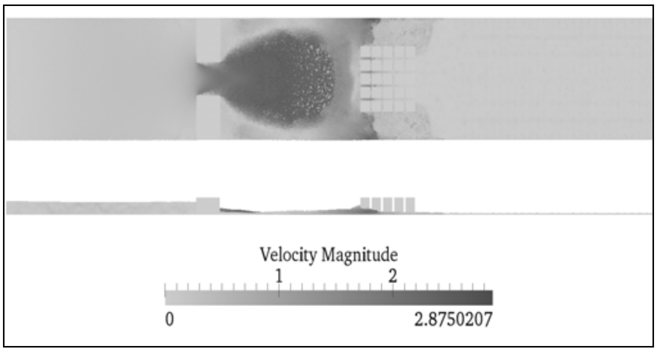


(b) Oblique layouts

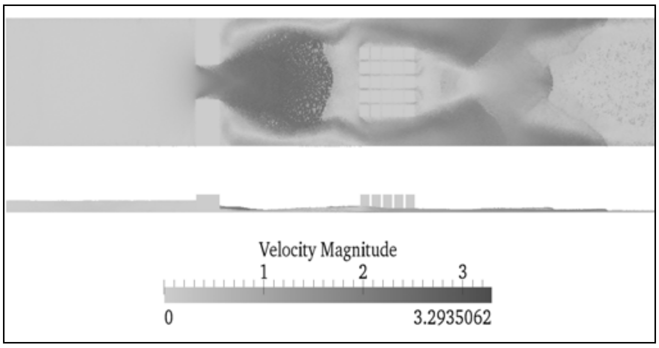
Fig. 2 Upstream hydraulic jump in the experiment by Soares-Frazão and Zech (2008) [6]

From the top view and side view, Fig. 3 and Fig. 4 present the computed velocity fields of the horizontal city layout and the computed velocity fields of the oblique city layout by SPH method at t = 4.0 s and t =8.0 s after the dam break, respectively. At t = 4.0 s, the flood flow has entered the streets, and a hydraulic jump forms in front of the buildings of both horizontal and oblique city layouts. Since the oblique city layout is not aligned with the flow approach direction, the hydraulic jump is not symmetric as shown in Fig. 2 (b). From the side view, we can obviously see the water level of the hydraulic jump is locally higher than without the presence of the buildings. Following the impact and reflection against the buildings, the speed of the flood wave in front of the buildings significantly decreases. At t = 8.0 s, the flooding fronts have crossed the urban district and a noticeable wave zone is identified just downstream from the building, surrounded by cross waves. Due to the repeated reflection of the flooding flow against the building and the sidewalls of the channel, velocities are reduced and more complex flow patterns can be observed in the area in front of the city layout.

Comparisons with Fig. 2, Fig. 3 and Fig. 4 indicate that the computed results of SPH method agree qualitatively well with the lab measurement, and also achieve good 3D local hydraulic features of the dam break flow through urban districts.

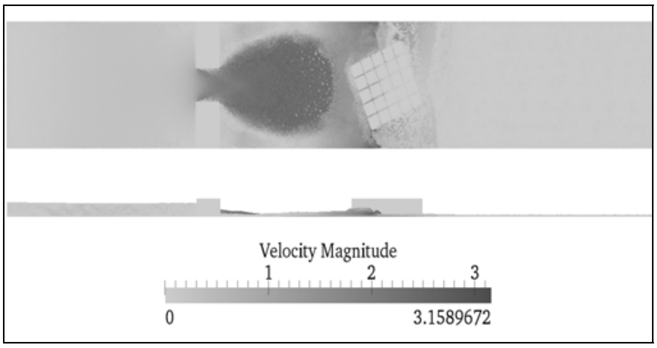


(a) t = 4.0 s

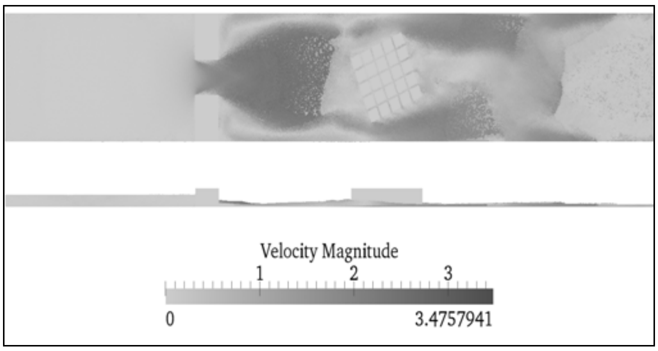


(b) t = 8.0 s

Fig. 3 Velocity field computed by SPH model (t = 4.0, 8.0 s)



(a) t = 4.0 s



(b) t = 8.0 s

Fig. 4 Velocity field computed by SPH model (t = 4.0, 8.0 s)

ISBN: 978-988-19252-9-9
ISSN: 2078-0958 (Print); ISSN: 2078-0966 (Online)

B. Dam-break Flooding with Floating Objects through the Intricate City Layout

The intricate network of city layout is consisted of several kinds of buildings, infrastructural constructions, idealized obstacles and three floating objects (see Fig. 5(a)). The city layout is in a channel which is horizontally 12.0 m long and 5.4 m wide. The dam locates at the origin of the channel with a size of 5.40 m × 1.0 m ×1.0 m. The first row of the city layout is 2.0 m away from the dam. The distance between the first two rows of buildings is 0.8 m. The width of the central street is 0.6 m. The height of the buildings in the first row is 2.0 m, and the heights of buildings in the second row are between 2.5 m and 3.0 m. The tallest tower downstream is high of 5.4 m. The floating objects of cone, cube and sphere shape locate upstream in black color in Fig. 5(a).

There are three different foundations of the buildings in the first row of the city layout. The first one is established with four cube cement piles and four cylinder cement piles to elevate parts of buildings above the foundation. The second one is constructed with a two-step staircase, and the buildings on the stairs have a 0.15 m free distance between each other. The third is established with a stair of only one step, and an isolated big building stands on it. In the second row and third row of the city layout, the buildings are established in different shapes, i.e. cube buildings, cylinder buildings.

Our numerical simulation used only one GPU. The particle spacing that we initialize the intricate city layout with was 0.02 m, and achieved total particles up to 1,079,334. We use the variable time step in SPH scheme with an initial time step (dt = 0.00003 s). Fig. 5 shows the propagation of the dam-break flooding in the intricate city layout, which is evaluated by the displacement of particles by SPH method at t = 0, 0.5, 1.0, 2.5 s after the dam break.

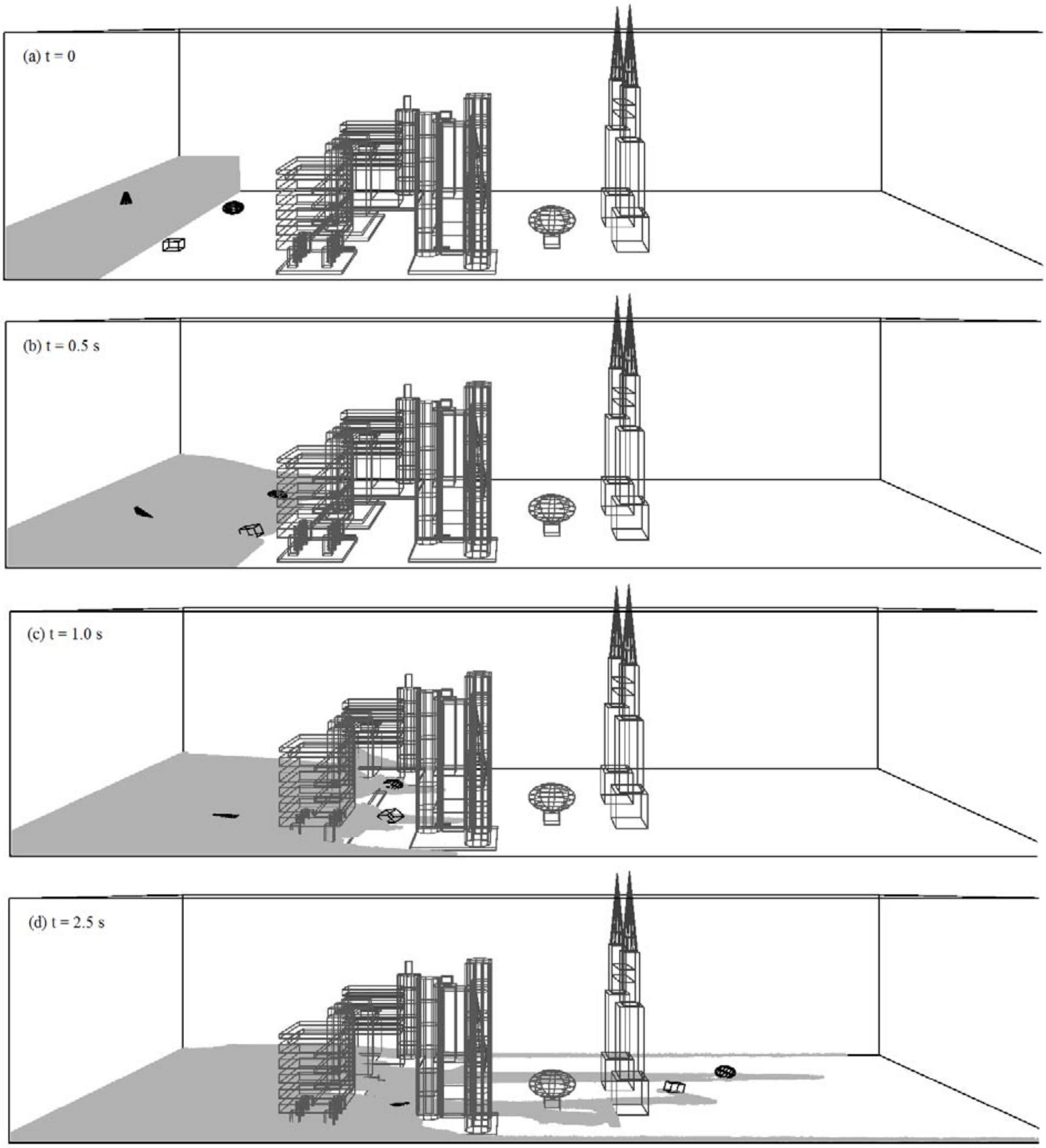


Fig. 5 Displacement of flooding flow and floating objects computed by SPH model (t = 0, 0.5, 1.0, 2.5 s)

Like the dam break flood through the idealized city layout, the upstream hydraulic jumps form in front of the buildings due to the impact and reflection of the flood flow against the buildings and sidewalls, and wave zones are produced right downstream of every building. However, more flow patterns occur in the complex urban district case. The flooding flow passing through the building with piles foundation moves faster than the buildings built on the staircases even though leaving small free distance between the buildings above the staircases.

From Fig. 5, we can observe that all three floating objects move into the city layout with the flooding flow, and the floating objects of cube and sphere shape interact with the buildings and infrastructures when travelling. Thus, we may conclude that SPH method can handle the interactions between flooding flow and moving objects and also the interactions between floating objects and buildings and infrastructures in the city layout. Meanwhile, from tracking the movement of tree floating objects, we can found that: (1) the flooding flow taking the sphere object moves faster which indicates that buildings in that street have a slight effect on the flooding flow there; (2) although the flooding flow taking the cube object passes through the piles foundation quickly, it has got significant resistance by the extreme building layout in the second row of the city layout, and therefore the velocity of the flooding flow significantly decreases and the velocity of moving objects decreases as well.

Incorporating the floating objects in the flooding flow is useful for investigating the flow features and flow propagation paths which is important to enhance the quality of urbanized flood hazard assessment and well forecast the downstream inundation mapping.

IV. Conclusion

SPH method indeed brings advantages on modeling highly unsteady dam-break flooding through city layouts. The computed results of GPUSPH model qualitatively show the typical and good 3D regimes of the urbanized dam-break flooding as shown in real experiments, and reasonably agreed with experimental data. Numerical results of simulating flooding with floating objects through intricate city layout identify complex 3D flow features, such as hydraulic jumps and flow discontinuities, which indicates SPH method is well-suited for modeling and handling the interactions between flooding flow and moving objects and also the interactions between floating objects and buildings and infrastructures in the city layout.

Although some preliminary reasonable results are achieved, the SPH model for handling dam break flow through urban districts is required further improvements. More quantitative comparison with other numerical methods and experimental data will be conducted.

References

[1] C. Zoppou, S. Roberts. "Catastrophic collapse of water supply reservoirs in urban areas," *Journal of Hydraulic Engineering*, vol. 125, no. 7, pp. 686–695, Jul. 1999.

[2] T. Ishigaki, H. Nakagawa, Y. Baba. "Hydraulic model test and calculation of flood in urban area with underground space," *Environmental Hydraulics and Sustainable Water Management- Lee & Lam(eds)*, Vol.2: Sustainable Water Management in the Asia-Pacific Region, A.A. Balkema Publishers 2004

[3] E. Mignot, A. Paquier, "Comparison of numerical and experimental simulations of a flood in a dense urban area". *Water Sci. Technol*, vol. 54, no. 6-7, pp. 65-73, Feb. 2006a.

[4] E. Mignot, A. Paquier, S. Haider, "Modeling floods in a dense urban area using 2D shallow water equations," *J. Hydrol (Amst)*, vol. 327, no. 1-2, pp. 186-199, Jul. 2006b.

[5] H.K. McMillan, J. Brasington, "Reduced complexity strategies for modeling urban flood plain inundation," *Geomorphology*, vol. 90, pp. 226-243, Oct. 2007.

[6] S. Soares-Frazão, Y. Zech, "Dam-break flow through an idealised city," *Journal of Hydraulic Research*, vol. 46, no. 5, pp. 648-658, Apr. 2008.

[7] Kamal El Kadi Abderrezzak, A. Paquier, E. Mignot, "Modelling flash flood propagation in urban areas using a two-dimensional numerical model," *Nat Hazards*, vol. 50 no. 3, pp. 433-460, 2009.

[8] S.W. Delelegn, A. Pathirana, "Multi-objective optimisation of cost–benefit of urban flood management using a 1D2D coupled model," *Water Science & Technology*, vol. 63, no. 5, pp. 1054-1060, Mar. 2011.

[9] Jochen E. Schubert, Brett F. Sanders, "Building treatments for urban flood inundation models and implications for predictive skill and modeling efficiency," *Advances in Water Resources*, vol. 41, pp. 49-64, Jun. 2012.

[10] P. Watts, "Tsunami features of solid block underwater landslides," *J. Waterway, Port, Coastal, Ocean Eng.*, vol. 126, no. 3, pp. 144-152, Jun. 2000.

[11] J. J. Monaghan, A. Kos, and N. Issa, "Fluid Motion Generated by Impact," *J. Waterway, Port, Coastal, Ocean Eng*., vol. 129, no. 6, pp. 250-259, Nov. 2003.

[12] S. C. Yim, D. Yuk, A. Panizzo, M. Di Risio, and P. L.-F. Liu, "Numerical Simulations of Wave Generation by a Vertical Plunger Using RANS and SPH Models," *J. Waterway, Port, Coastal, Ocean Eng.*, vol. 134, no. 3, pp. 143-159, Jun. 2008.

[13] L. B. Lucy, "Numerical approach to testing the fission hypothesis," *Astronomical Journal*, vol. 82, pp. 1013-1024, Dec. 1977.

[14] R. A. Gingold and J. J. Monaghan, "Smoothed Particle Hydrodynamics: Theory and Application to Non-spherical stars," *Monthly Notices of the Royal Astronomical Society*, vol. 181, pp. 375-389, Nov. 1977.

[15] S. Manenti, A. Panizzo, P. Ruol, L. Martinelli, "SPH simulation of a floating body forced by regular waves," 2008 *Proceedings of Third International SPHERIC Workshop*, pp. 38-41.

[16] B. D. Rogers, R. A. Dalrymple, P. K. Stansby, "SPH modeling of floating bodies in the surf zone," 2009 *Proceedings of 31st International Conference on Coastal Engineering*, pp. 204-215.

[17] Pourya Omidvar, Peter K. Stansby and Benedict D. Rogers, "Wave body interaction in 2D using smoothed particle hydrodynamics (SPH) with variable particle mass," *Int. J. Numer. Meth. Fluids*, vol. 68, no. 6, pp. 686-705, Feb. 2012.

[18] G. R. Liu, M.B. Liu, *Smoothed particle hydrodynamics: a meshfree particle method*. Singapore, World Scientific, 2003.

[19] E. Lo, and S. Shao, "Simulation of near-shore solitary wave mechanics by an incompressible SPH method," *Applied Ocean Research*, vol. 24, no. 5, pp. 275-286, Oct. 2002.

[20] R.A. Dalrymple, B.D. Rogers, "Numerical modeling of water waves with the SPH method," *J Coast. Eng.*, vol. 53, no. 2-3, pp. 141-147, Feb. 2006.

[21] J.J. Monaghan, "Simulating free surface flow with SPH," *Journal of Computational Physics*, vol. 110, no. 2, pp. 399-406, 1994.

[22] J.J. Monaghan, "Smoothed Particle Hydrodynamics," *Rep. Prog. Phys.* ,vol. 68, pp. 1703-1759, Jul. 2005.

[23] J.J. Monaghan, A. Kos, "Solitary waves on a Cretan Beach," *J. Waterway, Port, Coastal, Ocean Eng.*, vol. 125, no. 3, pp. 145-154, Jun.1999.

[24] A. Hérault, G. Bilotta, R. A. Dalrymple, "SPH on GPU with CUDA," *Journal of Hydraulic Research*, vol. 48(extra issue), pp. 74-79, Jun. 2010.

ISBN: 978-988-19252-9-9
ISSN: 2078-0958 (Print); ISSN: 2078-0966 (Online)