【Memoirs of Construction Engineering Research Institute Vol.61(papers) Nov.2019】

STUDY OF CHARACTERISTICS OF FLOW THROUGH TYPICAL STORM WATER DRAIN INLETS BY SIMULATION

Akihiko Nakayama Jing Kuan Tan Kok Weng Tan

ABSTRACT

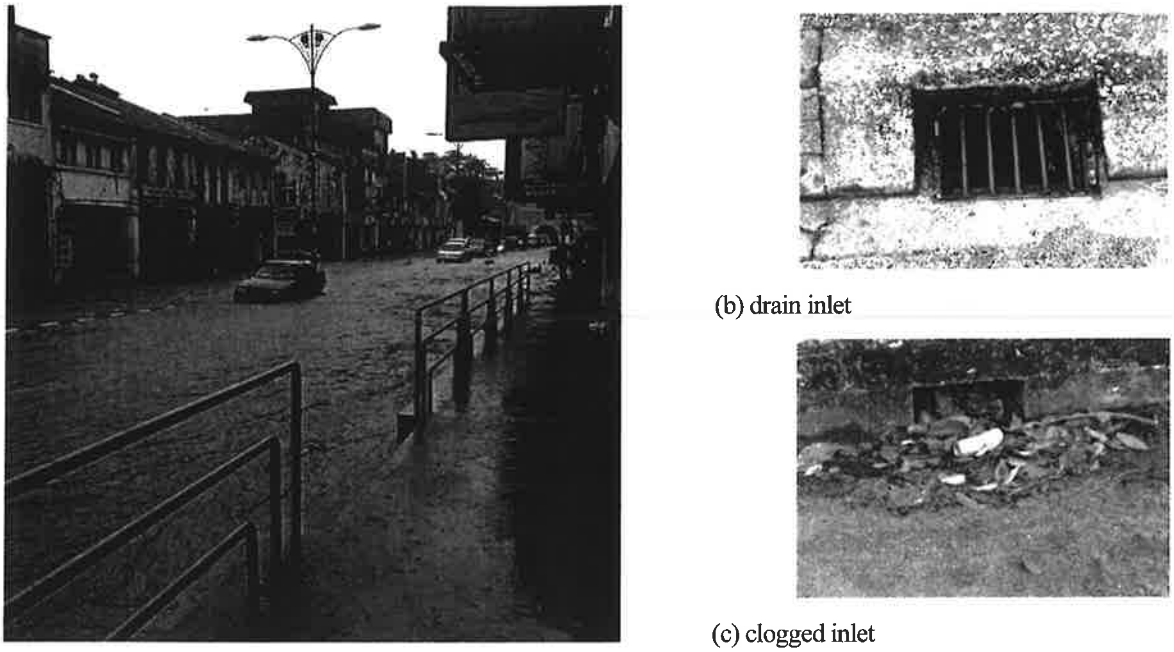
A smoothed particle hydrodynamics (SPH) method has been developed to simulate flows through storm water drainage inlet system. Conventional hydraulic analysis methods do not exist for such complex mixed modes of flow pattern The present work is an attempt to make use of recently developed mesh-free particle method (Smoothed Particle Hydrodynamics, SPH) to simulate the storm water flow over ground in the drainage gutter and into the drainage inlets. It treats the effects of turbulence by a method similar to the large eddy simulation representing the unresolved scales of motion by eddy viscosity. Trial computation indicates that the flow pastover ground, through gutters, inlet grid and into vertical shafts including the effects of falling rain can be simulated well for the purpose of evaluation and improvement of drain inlets.

1. INTRODUCTION

In spite of continued and stepped-up efforts to reduce water hazards, occurrence of floods has not reduced but even increased and the extent of the effects and damages are at the record levels not only within Japan but many other places in the world. There are different types of floods and new types are seen due possibly to the global warming. Inundation of low-altitude areas due to excessive rainfall over the catchment is one aspect, but recently localized heavy downpours are causing flash floods in various areas (Ref 1). In low-latitude areas surrounded by the sea in South East Asia, strong but relatively short-duration rainfalls cause frequent inundation of streets affecting the traffic. Though these inundations are not as serious as large-scale floods that submerge buildings and houses and may even wash them but they occur very frequent and even daily. The drainage of storm water in urban areas of Malaysia, for example, is designed for precipitation with 100 year recurrence probability (Ref 2) and the drain systems are sufficient to take the expected volume of water and the drain channel is rarely or never full. However, streets and parking lots are inundated frequently. The inundation of these streets and parking lots are usually caused by insufficient capacity of drain inlets. Figure 1 is a typical street flash flood caused by insufficient or clogged drain inlets.

Storm water drainage inlets are designed based on empirical estimate that a constant fraction of water coming towards the inlet goes through it[3]. The flow through drainage inlets installed along streets and parking lots are complex and the volume flow through them are conventionally estimated by a gross factor, 'capture rate' and the verification of the actual drain system is depended on a model test or observation of real structure. When the inlet is not submerged above the inlet and below, the flow is a free fall under gravity. If the drain shaft below the inlet is submerged, the flow through the inlet is controlled by the downstream water level. If too much water floods the area above the inlet, the inlet is choked and the flow rate reduces significantly compared with the ventilated flows.

Among various hydraulic analysis methods, there is none at present that can handle the complex water flow through the drain inlets[3]. The mesh-free particle methods that have been applied to free-surface flows may be one that can be adjusted to handle this type of flow. One of the present authors has applied Smoothed Particle Hydrodynamics (SPH) methods to high-speed air-water mixture past spill ways and stepped channels[4]. This method can cope with water flows with splashing droplets and the air entrained in water. One necessary improvement that is needed is to handle complex three-dimensional geometry of disconnected non-smooth boundary surfaces. SPH methods cope with these boundaries by placing particles called boundary particles that behave like water particles but at fixed on solid walls. The boundary



(a) flash flood inundating part of a street

(b) drain inlet

(c) clogged inlet

Figure 1. Flooding of streets due to ineffective storm water drain inlets.

particles can simulate the effects of solid walls by exerting repelling forces on the moving water particles coming near the solid surface. This method can represent the non-penetration condition very well but the frictional effects of high Reynolds number flows cannot be represented correctly.

The present work extends the SPH method developed for open-channel flows to complex flow through drain inlets. It is extended to simulate falling rain on the surface as well. The developed method is applied to a flow through model inlet and flows with some modifications and with incidental clogging.

2. CALCULATION METHOD AND BASIC EQUATIONS

2.1 Basic equations of SPH method

The basic method used here is very close to the method described earlier (Nakayama et al.[4]). In SPH method the basic equations of motion are written for discrete particles representing the flow variables at points that move with the flow. In the present formulation the momentum equations are the same as those used by Kajtar & Monaghan[5] except the stresses due to the unresolved turbulence are included as the sub-particle stresses. For particle a located at $\boldsymbol{r}_a$ with velocity $\boldsymbol{v}_a$ it is

$$\frac{d\boldsymbol{v}_a}{dt} = -\sum_b \frac{m_b}{\rho_a}\left(\frac{p_a}{\rho_a} + \frac{p_b}{\rho_b} + \Pi_{ab} + R_{ab}\right)\nabla_a W_{ab} + \boldsymbol{g} \tag{1}$$

where t is time, p_a and ρ_a are the pressure and the density of fluid at $\boldsymbol{r}_a$, i.e. those of particle a, $\nabla_a W_{ab}$ is the gradient of kernel function $W(\boldsymbol{r}_a - \boldsymbol{r}_b, h)$ with respect to the position $\boldsymbol{r}_a$, m_b and ρ_b are mass and the density of particle b located in the neighborhood of particle a so that the ratio m_b/ρ_b represents the volume that particle b occupies and $\boldsymbol{g}$ is the gravitational acceleration. Π_{ab} and R_{ab} are the terms to represent the viscous and the unresolved turbulence stresses. If the effects of unresolved small-scale turbulent motion are represented by the effective viscosity as many LES methods do, they can be expressed as the additional stress with the eddy viscosity coefficients μ_{ta}, μ_{tb} of particles a and b in addition to the coefficients of the molecular viscosity μ_a and μ_b. Then the expression proposed by Kajtar & Monaghan[5] is used for the total of the viscous and the turbulent stresses

$$\Pi_{ab} = -\frac{C_\mu(\mu_a+\mu_b)}{\varrho_b(\mu_a+\mu_b)}\frac{(\boldsymbol{v}_a-\boldsymbol{v}_b)\cdot(\boldsymbol{r}_a-\boldsymbol{r}_b)}{h\sqrt{|\boldsymbol{r}_a-\boldsymbol{r}_b|^2+0.01h^2}} \tag{2}$$

$$R_{ab} = -\frac{C_\mu(\mu_{ta}+\mu_{tb})}{\varrho_b(\mu_{ta}+\mu_{tb})}\frac{(\boldsymbol{v}_a-\boldsymbol{v}_b)\cdot(\boldsymbol{r}_a-\boldsymbol{r}_b)}{h\sqrt{|\boldsymbol{r}_a-\boldsymbol{r}_b|^2+0.01h^2}} \tag{3}$$

In these equations h is the smoothing length and is taken as 1.3 times the average particle separation as in most previous work (Monaghan[6], Liu & Liu[7]). The summation over all b in the neighborhood approximates the integration over the volume with the radius of the order of h surrounding point $\boldsymbol{r}_a$. The eddy viscosity μ_{ta} representing the unresolved turbulence is modeled following the Smagorinsky model used in grid-based LES methods. The SPH form with the Smagorinsky constant C_S is

$$\frac{\mu_{ta}}{\rho_a} = (C_S 2h)^2 |D_{ij}|,\ D_{ij} = \sum_b \frac{m_b}{\rho_b} v_{bi} \nabla_{aj} W_{ab} \tag{4}$$

where v_{bi} is the ith component of the velocity vector v_b, ∇_{aj} is the jth component of the differential operator ∇_a. The continuity equation takes the following form

$$\frac{d\rho_a}{dt} = -\sum_b m_b v_b \cdot \nabla_a W_{ab}\ . \tag{5}$$

For the kernel function W_{ab} the following cubic spline fluctuation is used.

$$W_{ab} = W(\boldsymbol{r}_a - \boldsymbol{r}_b, h) = \begin{cases} \frac{1}{6}[(2-q)^3 - 4(1-q)^3],\ 0 \le q \le 1 \\ \frac{1}{6}(2-q)^3,\quad 0 \le q \le 1 \\ 0,\quad q > 2 \end{cases} \tag{6}$$

where $q = |\boldsymbol{r}_a - \boldsymbol{r}_b|/h$.

2.2 Boundary conditions

The boundary conditions for the SPH equations (1) and (5) are somewhat different from boundary conditions for partial differential equations. Eqs. (1) and (5) are not valid all the way up to the boundaries since the region of influence lies outside the boundary where there are no particles. Conventional methods of dealing with this situation are either the consideration of the contribution from the surface integral that arises from particle interpolation over fine region (Violeau & Issa[8]) or to place artificial particles that give correct effects of boundaries[7),9)-11)]. Solid boundaries require the velocity to be zero and slip boundaries require the velocity component normal to the wall to be zero. These conditions can be satisfied by placing boundary particles and/or image particles on and outside the boundary. When the flow is turbulent, however, none of these conditions are satisfied unless the smallest scale of motion near solid wall is resolved. Most of the practical calculations are done with number of particles not sufficiently large to resolve this scale. This is the same situation as the large eddy simulation that does not resolve wall viscous scale. Therefore, in the present formulation we treat the particles that fall near the solid boundary separetely and require them to satisfy the wall condition as well as the conservation conditions.

In the present approach, therefore, the equations of motion are modified or replaced with those that follow the restrictions imposed by the solid boundaries. First, the velocity components near the solid boundary $n<h_b$ are resolved into tangential and the normal velocity components

$$\boldsymbol{v}_a = v_t\boldsymbol{t} + v_n\boldsymbol{n} \tag{7}$$

where $\boldsymbol{t}$ and $\boldsymbol{n}$ are the unit vectors tangent to and normal to the solid wall close to particle a. The tangential velocity is determined from the pressure gradient and the shear stresses from the nearby particles, and the shear stress due to the wall so that it is computed by

$$\frac{dv_t}{dt} = -\sum_b \frac{m_b}{\rho_a}\left(\frac{p_a}{\rho_a} + \frac{p_b}{\rho_b} + \Pi_{ab} + R_{ab}\right)\nabla_{at} W_{ab} + T_B + \boldsymbol{g}_t \tag{8}$$

where ∇_{at} is the component of the differential operator ∇_a in $\boldsymbol{t}$ direction, $\boldsymbol{g}_t$ is the component of the gravitational acceleration in $\boldsymbol{t}$ direction. T_B is the force due to the wall resistance

$$T_B = -\tau_w/r_n \tag{9}$$

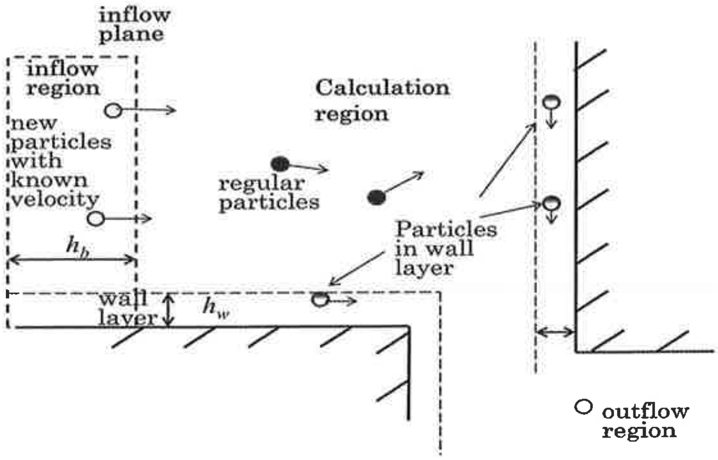


Figure 2. Boundary conditions

where τ_w is the wall shear stress given by the standard wall law corresponding to the velocity v_t at the distance r_n from the wall. The determination of τ_w is done by an iterative method when the wall law is given with logarithmic function similar to the way it is done in fixed grid LES (Nakayama & Yokojima[12]).

Once the tangential velocity components are determined by the above equations with the effects of the wall stress, the normal velocity component is determined from the continuity in terms of the tangential velocity. Although the present formulation assumes weak compressibility, incompressible relation is used here. Then

$$v_n = -\left(\frac{\partial v_{t1}}{\partial t_1} + \frac{\partial v_{t2}}{\partial t_2}\right) r_n \tag{10}$$

where t_1 and t_2 are the two orthogonal directions in the plane tangent to the boundary and r_n is the distance fro m the wall to particle a.

The other boundary conditions are applied as shown in Figure 2. In the inflow plane on the left boundary, new particle with prescribed velocity and pressure is introduced when a particle in the inflow region goes into t he main calculation region.

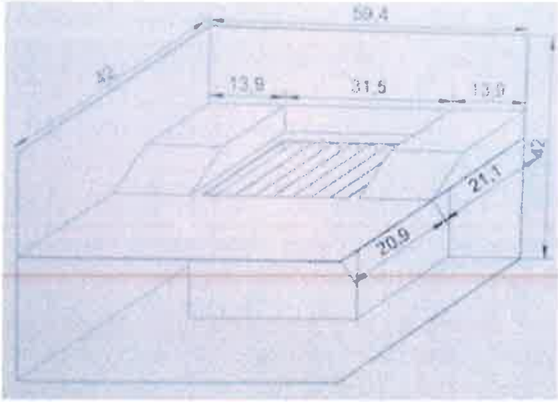
2.3 Method of finding distance to the closest boundary and the surface normal direction

The boundary conditions implemented as above requires determination of the distance from all moving particles to the closest solid boundary and its normal direction. While finding these geometric quantities is easy when the solid surfaces are simple planes, but if the boundary geometry is complex such as those in the drain inlet which may have complex shapes and grids a quick determination is not easy. In the present implementation of the method, we use the link cell technique to search for neighboring particles. To find the nearest boundary, we give the position and the normal direction of the solid boundary surfaces at finite number of points, which may be called boundary points. If we place these points at sufficiently closely space positions, the task of finding the closest boundary is done by finding the closest point, which can be done fairly easily by the same link-cell technique.

3. SIMULATION OF FLOW THROUGH MODEL INLET

Simulation has been conducted first for the case tested in a laboratory (Kong[13]). The drawing and the photograph of this inlet are shown in Figure 3. This is one of standard drain inlets used in Malaysia. The opening is covered by bars that are aligned perpendicular to the curb. The elevation of the pavement on both sides of the inlet is elevated by 5cm. The

(a) drawing of inlet model



(b) photograph of the laboratory model

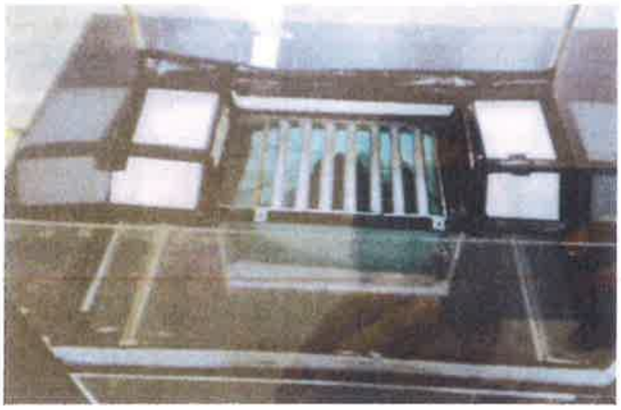


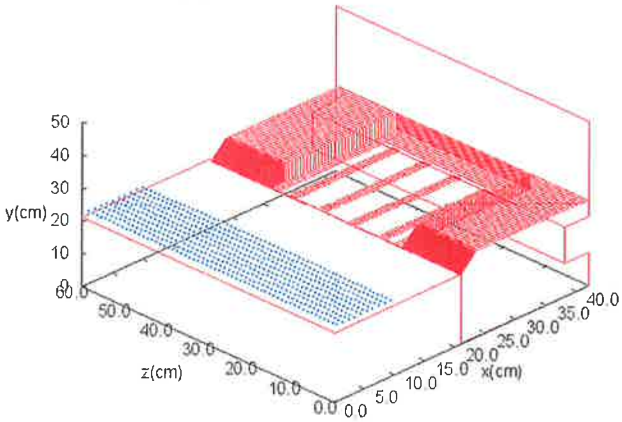
Figure 3 Stormwater drain inlet model

vertical shaft below the inlet bars is of rectangular cross section. In the real drain system there is a drainage channel below the drop shaft which is not modeled since the flow we are considering is assumed not too large to fill this channel. The experiment was conducted for the total flow rate of 40 l/min corresponding to the runoff from a typical small parking lot of about 100 m^2. The drain channel under the drop shaft is assumed to be sufficiently large that it does not influence the flow through the inlet.

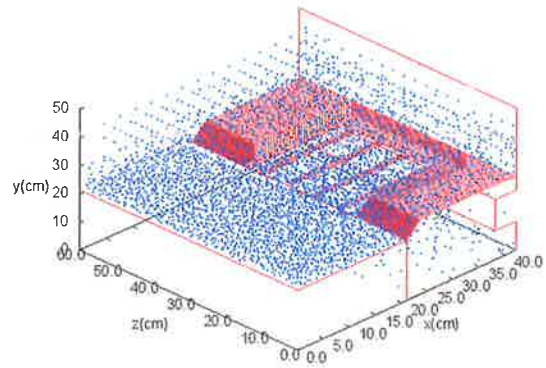
4. SIMULATION RESULTS

Figure 4 shows the geometry of the simulated inlet, the initial condition and the results of the baseline case that has been observed in the laboratory experiment. Hence the inflow through the left boundary is hence adjusted to be 20 l/min and

(a) initial condition at t=0



(b) t=0.75s



(c) t=1.50 s

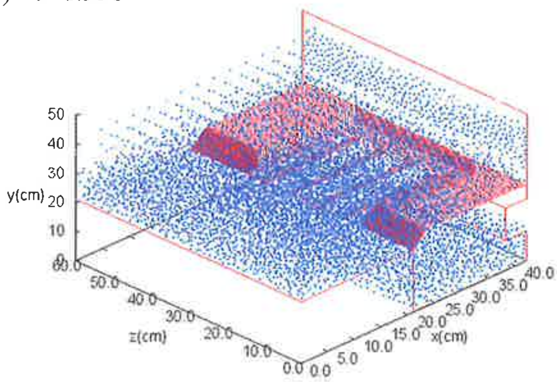


Figure 4. Simulation of rain flow through model inlet, baseline Case1. Blue dots indicate water particles and red dots and red lines indicate solid boundaries.

the rainfall on the inlet is 1mm/min. Figure 4(a) is the assumed initial state at $t = 0$ when the inlet is dry and half of the ground surface is assumed to be covered by 1cm deep water at rest. The boundaries defined by the flat 2-d surfaces are shown in red lines and the surfaces represented by the wall particles are shown in red dots (38000 particles for this configuration). The vertical wall on the back of the inlet is part of the curb of the side walk and its height is taken 30cm high enough not to overtop it. The initial separation of particles is 1cm so there is only one layer of 600 water particles (shown in blue dots in the figure) wetting the ground surface. The particle separation is smaller than the smallest dimension of the inlet grid.

Figure 4(b) and 4(c) show the water flow and the rain at subsequent times. At t =0.75s the water coming from the left boundary, where the storm water from the street is coming in, already flows over the inlet grid plane and impinges on the wall of the curb. The flow through the inlet is very small fraction of the inflow. At t =1.5 s the flow starts to drain through the inlet as seen in Figure 4(c). It is noted that the water of depth of the order of 10cm accumulates on top of the inlet. It is also seen that more flow goes through the inlet near the back wall of the curb. The flow at subsequent time approaches the steady state. In the present simulation the drain channel below the inlet is assumed to be deep and large enough and all the incoming flow drains out of the inlet system. The downstream structure can be and need be modeled to evaluate their influence.

To help understand the flow features shown in the three-dimensional view of Figure 4, the side views of the velocity vectors of the water particles going through the inlet are shown in Figure 5. It shows the magnitudes and the directions of the individual water particles projected in z-direction. It shows the magnitudes and the directions of the individual water particles. It is seen that the water tends to drop through the inlet near the back side that is close to the curb and the region directly below the entrance there are less number of particles and some are flowing backwards. This flow was observed in a laboratory test conducted by Kong[10], which shows a similar trend.

Figure 6 shows the similar result when the inlet grid is partially blocked by trash or fallen leaves which is often seen as in Figure 1. This is one of the causes of street flash floods and the way the inlet flow is influenced by the partially or fully blocking objects. Some leave like objects are placed that blocks 20% of the passage as shown in Figure 6(a). The simulated flow is shown in the subsequent figures. It is seen that the flow through the inlet is about the same as that without the blockage at t=0.75s when the water above fills the passages between the grid bars. However as more flow approaches at t=1.5s, the water fills the passages between the grid bars, the flow passages get restricted. Then the depth of water above the inlet plane increases and the flow becomes like orifice flow rather than a free falling flow through the gaps.

The present simulation method allows examination of the existing inlet system but can examine possible modifications as well. Since the flow of the baseline design indicated that the flow just below the inlet surface reverses just downstream of the entrance. So an additional simulation was conducted for the flow with tapered entrance. Figure 7 shows the

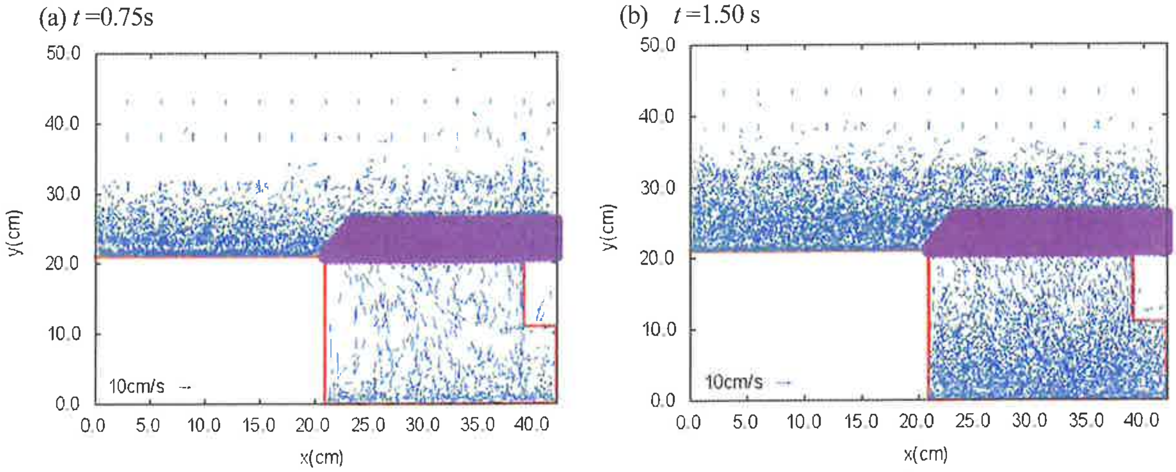


Figure 5. Side view of velocity vectors, Case1.

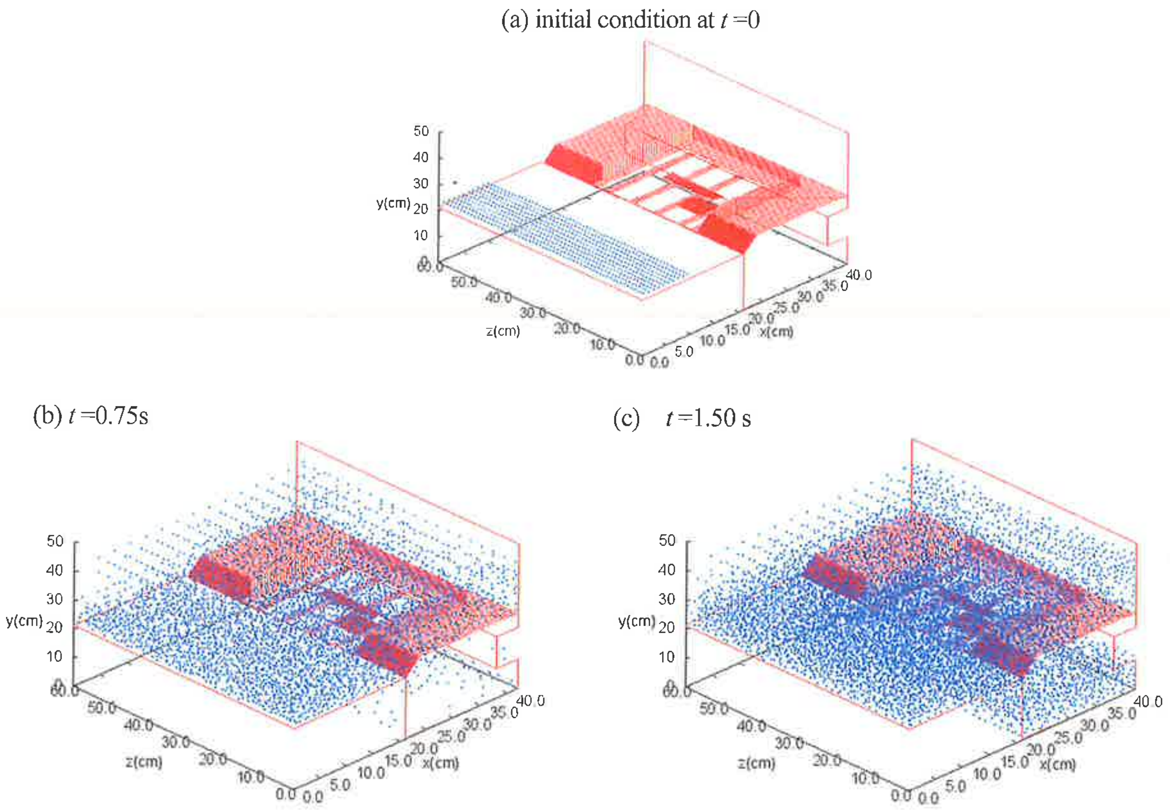


Figure 6. Simulation of rain flow through model inlet, Case 2 with blocking obstacles. Blue dots indicate water particles and red dots and red lines indicate solid boundaries.

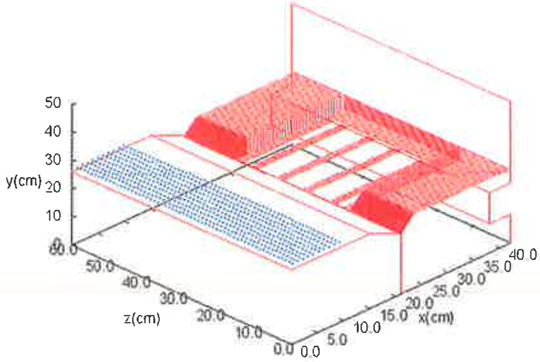
simulation results for Case 3, in which the entrance from the street pavement into the inlet is inclined. This is motivated by the flow pattern of baseline Case 1, in which most of the water flows over the inlet and impinges on the back wall and only after that it drops through the opening of the inlet. It is seen from Figures 7(b) and 7(c) that the flow becomes smoother and more flow goes through the front part of the entrance.

5. CONCLUSION

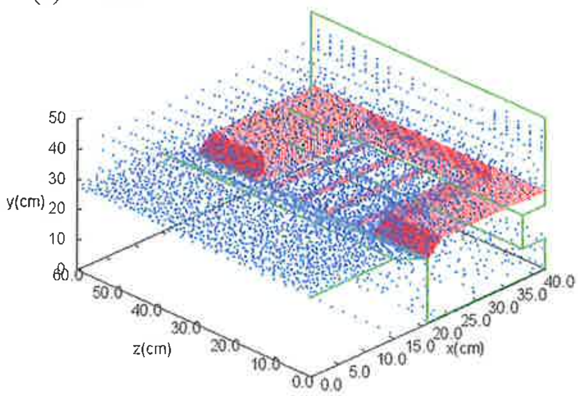
A weakly Compressible SPH method has been developed to simulate complex flows of surface runoff approaching and going through an urban stormwater drain inlet so that the performance and its improvement may be evaluated. The complex three-dimensional boundary is represented by boundary points placed on solid boundaries which contain information of the position and the direction of the boundary surface. With this technique the complex grid bars as well as complex inlet shapes can be represented fairly easily and reflected in the motion of the nearby water particles. Also the falling rain is modeled by introducing water particles at the top of the simulation region and letting them free fall. The results of the baseline test case appear to reproduce laboratory flow through the model inlet.

In addition, further simulations with particle blockage due to fallen objects and with modified geometry were conducted which indicate that the method does represent the detailed changes of the flow characteristics. The change in the entrance part of the inlet is found to change the flow characteristics and the total flow rate. The simulation cases conducted in the present work is nowhere sufficient to evaluate the performance of the existing inlet nor is it enough to suggest the improvement. However, methodology itself is enough to be exploited for investigation of various conditions which may lead to improvement and reduction of flash floodings of many streets and parking lots.

(a) initial condition at t=0



(b) t=0.75s



(c) t=1.50 s

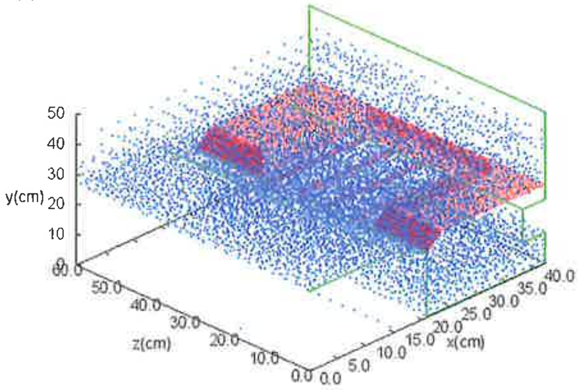


Figure 7. Simulation of rain flow through model inlet Case3 tapered approach. Blue dots indicate water particles and red dots and red lines indicate solid boundaries.

Further work that should be conducted is to simulate the flow in larger approach area and the downstream structures. The way the surface flow approaches an inlet influences the way it goes into the inlet. When the depth of approaching water increases, the three dimensionality of the flow becomes significant and inlet vortex may appear.

REFERENCES

1) Kundzewicz, Z. W., Kanae, S., Seneviratne, S. I., Handmer, J., Nicholls, N. Peduzzi, P., Mechler, R., Bouwer, L.M.、Arnell, N., Mach, K., Muir-Wood, R., Brakenridge, R., Kron, W., Benito, G., Honda, Y., Takahashi, K. and Sherstyukov, B.:Flood risk and climate change: global and regional perspectives, Hydrological Sciences Journal, Vol. 59, issue 1, doi:https://doi.org/10.1080/02626667.2013.857411, (2014).

2) Mays, L.W.: Stormwater Collection Systems Design Handbook, McGraw-Hill, pp.6.65-6.84, (2001).

3) HEC22 Urban Drainage Design Manual, http://www.fhwa.dot.gov/bridge/hec22.pdf, (2001).

4) Nakayama, A., Leong, L.Y. and Kong, W.S.: Simulation of turbulent free surface flow with two-phase SPH method, Journal of Japan Society of Civil Engineers Ser.B1(Hydraulic Engineering) Vol.72, No.4, I_523-I_528, (2016).

5) Kajtar, J. and Monaghan, J. J. : SPH simulation of swimming linked bodies, *J. Comput. Phys.*, Vol. 227, pp.8568-8587, (2008).

6) Monaghan, J. J. : Simulating free surface flows with SPH, *J. Comput. Phys*. Vol.110. pp.399-406, (1994).

7) Liu, G..R. and Liu, M. B.: *Smoothed Particle Hydrodynamics: A Meshfree Particle Method*, World Scientific Pub. Co. Inc., Singapore, (2003).

8) Violeau, D. and Issa, R. : Numerical modelling of complex turbulent free-surface flows with the SPH method: an overview, *Int. J. Numer. Meth. Fluids*, Vol. 83, pp.277-304, (2007).

9) Colagrossi, A. and Landrini, M. : Numerical simulation of interfacial flows by smoothed particle hydrodynamics, *J. Comput.*

Phys. Vol.191, pp.448-475, (2003).

10) Gotoh, H., Sibahara, T. and Sakai, T. : Sub-particle-scale turbulence model for the MPS method – Lagrangian flow model for hydraulic engineering, *Computational Fluid Dynamics Journal* Vol. 9, pp.339-347, (2001).

11) Monaghan, J. J. : Smoothed particle hydrodynamics and its diverse applications, Annu. Rev. Fluid Mech. Vol.44, pp.326-346, (2012).

12) Nakayama, A. and Yokojima, S. : Modeling free-surface fluctuation effects for calculation of turbulent open-channel flows, Environmental Fluid Mechanics, Vol.3, pp.1-21, (2003)

13) Kong Y. H. : Sustainable urban stormwater management in Malaysia, A study of innovative drainage system in Universiti Tunku Abdul Rahman, Final Year Project Thesis, Department of Environmental Engineering, Universiti Tunku Abdul Rahma . (2019)

AUTHORS

Akihiko Nakayama	Ph.D., Fluid Mechanics, Hydraulics
Tan Jing Kuan	Student, Department of Environmental Engineering, Universiti Tunku Abdul Rahman
Tan Kok Weng	PhD, Department of Environmental Engineering, Universiti Tunku Abdul Rahman