A STUDY OF BREAKING WAVE METHOD USING SMOOTHE PARTICLE HYDRODYNAMICS

José Gustavo Coelho, josegustavo@icte.uftm.edu.br
Federal University of Triângulo Mineiro, Uberaba, Brazil

Taygoara Felamingo de Oliveira, taygoara@unb.br
Antonio Cesar Pinho Brasil Junior, brasiljr@unb.br
University of Brasília, Brasília, Brazil

Pierre Maruzewski, pierre.maruzewski@edf.fr
EDF Group, Le Bourget du Lac, France

Abstract. *The flow in conditions of free surface has characteristics that make it difficult to simulate using the conventional numerical methods, like Finite Volume Method (FVM) or Finite Element Method (FEM). In this context, the Smoothed Particle Hydrodynamics (SPH) method is used to simulate the breaking waves. This is a Lagrangian and meshless method and the flow is divided into volume parts of fluid, using the concept of particles. The code was validated with experimental data and in the future, we intend to study the phenomenon of erosion by coupling the results obtained from the impacts of waves in coastal regions.*

Keywords: *SPH, meshless, wave, Lagrangian referencial*

1. INTRODUCTION

The Smoothed Particle Hydrodynamics method is comparatively new and was created by Lucy and Monaghan in 1977 for modeling of Astrophysics (Lucy, 1977) and later extended to other fields, such as the material deformation (Gray *et al.*, 2001), the multiphase flow (Hu and Adms, 2007), the fluid dynamics (Monaghan, 1994), (Monaghan and Kos, 1999) and other. In this numerical method, the domain is represented by discrete volumetric parts (particles), and each part has properties individual and they move in accordance with the governing equations. The SPH is a Lagrangian meshfree method. The properties of each particle are determined by the information of other neighbors particles. This neighbor is determined by a process of space average, or smoothing, which is defined by a finite radius of influence. This is fulfilled by an interpolation or weight function which is often called the interpolation kernel (Hosseini *et al.*, 2007).

There are various approaches to incorporate the incompressibility in a numerical model. This study uses an approximation termed as "weakly compressible" where the pressure equation is solved explicitly by an equation of state. This approach has acceptable results in flows with a very small Mach number, in addition, reduces the computational effort (Colagrossi and Landrini, 2003).

This work is the initial part of the study of the erosion caused by waves. In this problem, the behavior of the free surface wave justifies the effort to develop a method capable of obtaining good results, even under the complicated situations how large deformation and detection of discontinuity. So, to confirm the robustness of the code, in this work the dam break and the irregular wave are analyzed.

2. SMOOTHED PARTICLE HYDRODYNAMICS FORMULATION

In the formulation of the SPH every particle has physical properties, such as mass, volume, pressure, velocity and density. All properties vary during evolution, except the mass of the particle. These properties represent a spatial average centered on a point of the domain. Considering a generic property A (scalar, vector or tensor) value of the particle at a position **x** is given by

$$A(\mathbf{x}) = \int_{\Omega} A(\mathbf{x}')W(\mathbf{x} - \mathbf{x}')d\mathbf{x}' \tag{1}$$

where W is the kernel, and Ω is the domain. This integration is performed only in a portion of volume and this defines the neighboring particles. In this study, we use a function known as cubic spline (Monaghan, 2002).

Thus, considering the particle position a in **x**, Eq. (1) can be rewritten as

$$A(\mathbf{x}_a) = \sum_b \frac{m_b}{\rho_b} A_b W_{ab} \tag{2}$$

where $\sum_b$ is the sum of all particles that are within the radius of influence of the particle a, m is the mass and x_{ab} is the distance between particles a and b.

2.1 FLUID MECHANICS EQUATIONS IN SPH FORMULATION

The conservation of mass in SPH formalism can be written as

$$\frac{D\rho_a}{Dt} = \sum_b m_b \mathbf{u}_{ab} \cdot \nabla_a W_{ab} \tag{3}$$

where D/Dt is the material derivative, $\mathbf{u}_{ab} = \mathbf{u}_a - \mathbf{u}_b$ and $\nabla_a W_{ab}$ corresponds to the spatial partial derivative of the kernel.

In a Lagrangian form, the moment equation is written like

$$\frac{D\mathbf{u}}{Dt} = -\frac{1}{\rho}\nabla P + \nu \nabla^2 \mathbf{u} + \mathbf{F} \tag{4}$$

where ρ and P denote respectively the density and mechanical pressure, ν is the kinematic viscosity of the fluid, ∇^2 is the Laplacian operator and **F** external forces such as gravity.

Using Eq. (3), Liu (Liu and Liu, 2003) suggests that the pressure gradient can be written as

$$\nabla P_a = \rho_a \sum_b m_b \left(\frac{P_b}{\rho_b^2} + \frac{P_a}{\rho_a^2} \right) \nabla_a W_{ab} \tag{5}$$

where P_a is the pressure of the particle a.

The viscous term used was proposed by Monaghan (Monaghan, 1992) according to

$$\nu \nabla^2 \mathbf{u}_a = 8 \frac{\nu_a + \nu_b}{\rho_a + \rho_b} \frac{\mathbf{u}_{ab} \cdot \mathbf{x}_{ab}}{x_{ab}^2 + 0.01h^2} \nabla_a W_{ab}, \tag{6}$$

where h is the smoothing length.

The pressure is calculated explicitly, using a state equation given by

$$P = \frac{\rho_0 c^2}{\gamma} \left[\left(\frac{\rho}{\rho_0} \right)^{\gamma} - 1 \right], \tag{7}$$

where ρ_0 = 1000 and $\gamma = 7$. This approximation for the pressure was proposed by Batchelor (Batchelor, 1967) and for liquids presents a good approximation, since the variation the density is small (Monaghan, 2005). This approach depends on the numerical speed of sound, c. The appropriate value of c is one in which the maximum variance the density is below 1% (Morris *et al.*, 1997). In general, to simulate a nearly incompressible flow, c must be at least ten times superior than the maximal velocity of the flow.

The Larde Eddy Simulation (LES) and SPH are spatial filters. So, we use a LES approach. Filtering the moment equation, Eq. (4), we obtain (Violeau and Issa, 2006)

$$\frac{D\bar{\boldsymbol{u}}}{Dt} = -\frac{1}{\rho}\nabla\bar{P} + \nu\nabla^2\bar{\boldsymbol{u}} + \mathbf{F} + \frac{1}{\rho}\nabla\cdot\boldsymbol{\tau} \tag{8}$$

where $\boldsymbol{\tau}$ is the sub-grid Reynolds tensor. Each component of this tensor is defined as $\tau_{ij} = \rho\left(\bar{u}_i\bar{u}_j - \overline{u_i u_j}\right)$ (Gotoh *et al.*, 2004).

Using the Boussinesq hypothesis,

$$\frac{\tau_{ij}}{\rho} = \left(2\nu_t\bar{S}_{ij} - \frac{2}{3}k\delta_{ij}\right) \tag{9}$$

where $\bar{S}_{ij}$ is the filtered strain rate tensor and k the turbulent sub-grid kinetic energy that can be incorporated into the pressure term. The strain rate tensor filtered, $\bar{S}_{ij}$, is determined by

$$\bar{S}_{ij} = -\frac{1}{2}\left(\frac{\partial\bar{u}_i}{\partial x_j} + \frac{\partial\bar{u}_j}{\partial x_i}\right). \tag{10}$$

To model the eddy viscosity, we use the Smagorinsky model, $\nu_t = (Cs\Delta)^2|\bar{S}|$, where Cs is Smagorinsky constant, $Cs = 0,12$ (Violeau and Issa, 2006), Δ is the length scale which is equal to the initial distance between the particles (Dalrymple and Rogers, 2006). The local value of the tensor $|\bar{S}| = (2\bar{S}_{ij}\bar{S}_{ij})^{1/2}$.

Thus, the sub-grid Reynolds tensor in relation to the particle a is determine by (Gotoh *et al.*, 2004)

$$\left(\frac{1}{\rho}\nabla\cdot\boldsymbol{\tau}\right)_a = \sum_{b=1}^{N} m_b\left(\frac{\boldsymbol{\tau}_a}{\rho_a^2} + \frac{\boldsymbol{\tau}_b}{\rho_b^2}\right)\cdot\nabla_a W_{ab}. \tag{11}$$

For the time integration several time step criteria must be satisfied,

$$\Delta t = min\left\{0,25\frac{h}{c}; 0,25\min_a\sqrt{\frac{h}{f_a}}; 0,125\min_a\frac{h^2}{\nu}; \iota\frac{h_w}{u_w}\right\} \tag{12}$$

where $\iota = 0.005$, f_a is the magnitude of particle accelerations, h_w and u_w are the smoothing and velocity of the wall. In cases where there is a moving boundary, the latter condition in Eq. (12) is most restrictive.

The particles are moved with the following equation:

$$\frac{D\mathbf{x}_a}{Dt} = \mathbf{u}_a - \epsilon\sum_{b=1}^{N}\frac{m_b}{\rho_b}\mathbf{u}_{ab}W_{ab} \tag{13}$$

where the constant ϵ is 0.03. This is a correction to the particle velocity a and was proposed by Monaghan (Monaghan,

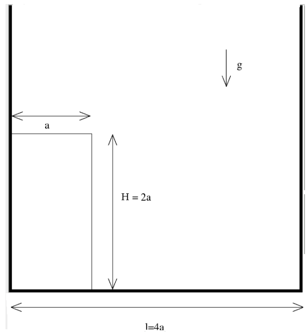


Figure 1. Geometry of the dam break.

2005) where the neighbors particles move with approximately the same velocity.

Finally, for the boundary conditions, we use the repulsive force exerted by the wall particles proposed by Monaghan (Monaghan, 1992). This force is given by

$$FP_{ab} = \begin{cases} D\left[\left(\frac{r_o}{x_{ab}}\right)^{12} - \left(\frac{r_o}{x_{ab}}\right)^{4}\right]\frac{x_{ab}}{r^2{}_{ab}} & \text{if } \frac{r_o}{x_{ab}} \leq 1 \\ 0 & \text{if } \frac{r_o}{x_{ab}} > 1 \end{cases} \tag{14}$$

where the constant D is the square of the magnitude higher flow velocity. The distance r_0 is approximately equal to the initial distance between the particles. Both play a key role for the stability of the method (Liu and Liu, 2003).

3. RESULTS

The first problem studied is the dam break. This example is frequently used to validate numerical methodologies, particularly with the free surface condition. In the Fig. 1 there is an illustration of the domain, where a=1 m. The active force on this problem is the gravitational and the dam break is performed at time $t = 0$, when the dam is removed. In this case we use the discretization of a/100, that result in 20000 fluid particles. For the wall, we use 4 layers of the particles. The smoothing length is $1.3\triangle x$. Using gravity as negative in the direction y, Monaghan Monaghan (1994) suggests that the initial density must be determined by

$$\rho = \rho_0 \left(1 + \frac{7\rho_o g(h-y)}{\rho_o c^2}\right)^{1/7} \tag{15}$$

where ρ_0 = 1000, H is the height of the water. Other point in focus is the value of the numerical speed of sound. The Suggestion is $c = \sqrt{200gH} \approx 63$, but this value results in a variation of density greater than 1%. For the variation of density smaller than this limit, $c = 100$.

Figure 2 shows a comparison of the fluid obtained using numerical and experimental (Idelsonh *et al.*, 2004) results, where there is the detection of the tube which is difficult to simulate with conventional numerical method. This shows the robustness of the SPH method. Another comparison is made with (Koshizuka and Oka, 1996), Fig. 3, where there is the greater variation of the axis x and of the height. The results obtained are satisfactory.

The other problem studied is the irregular wave propagation. The domain used is shows in Fig. 4. The left wall is responsible for generating irregular waves. To move the wall we use a signal, found in (Cox and Ortega, 2002), where an experimental study of this sample was performed.

The smoothing length used is $h = 1,3\Delta x$ for all particles, except the moving boundary. For these particles we use

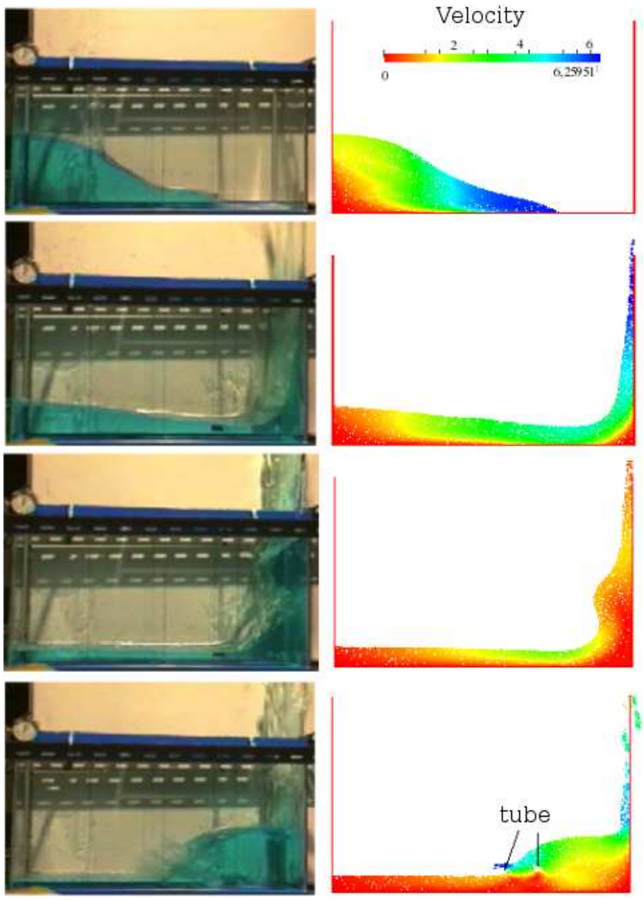
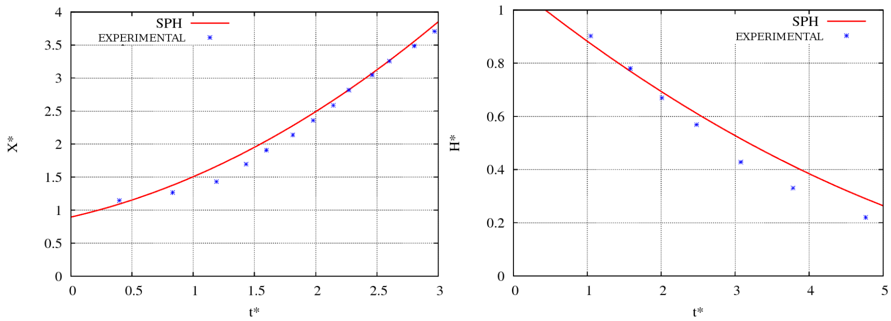


Figure 2. Comparison of numerical and experimental(Idelsonh *et al.*, 2004) results.



(a) (b)

Figure 3. Comparison with experimental simulation (Koshizuka and Oka, 1996) where $X^* = x/a$, $t^* = t\sqrt{(2g/a)}$, $H^* = H/(2a)$ and $t^* = t\sqrt{(2g/a)}$.

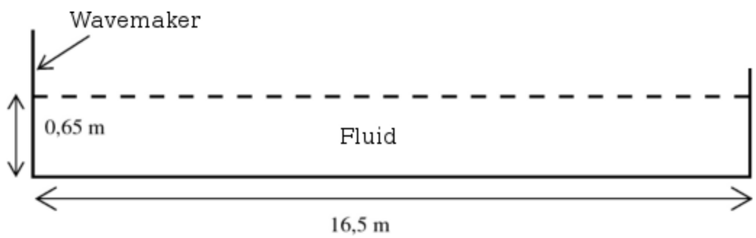
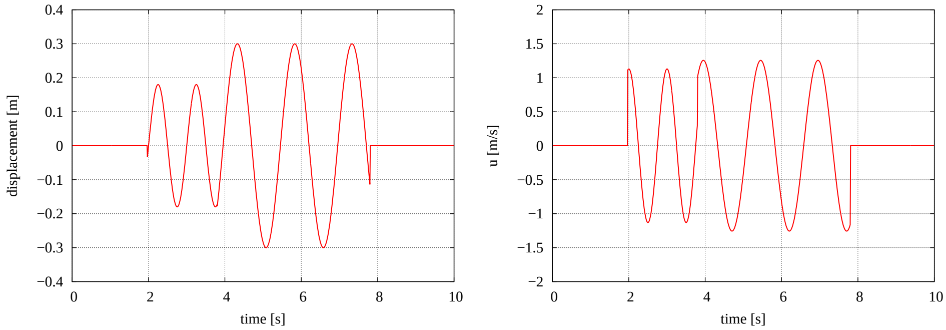


Figure 4. Initial configuration of the fluid and particles of the wall to reproduce the irregular waves.



(a) (b)

Figure 5. Displacement and velocity of the wavemaker in the horizontal axis.

$h = 0,3\Delta x$ to prevent numerical instabilities. The amount of particles used is 43236 (4000 fluid and wall 3236). The numerical speed of sound is $35,7$ m/s and $.pdfilon = 0.03$.

To reproduce the moving boundary the position and velocity of the particles are externally imposed. Changes in the velocity value may generate high accelerations of the fluid particles causing instabilities. To prevent a high acceleration we use the smoothing function proposed for (Gómez-Gesteira *et al.*, 2005). Thus, if the wavemaker moves in amplitude A_i and frequency f_i in the interval $t \in [t_i, t_{i+1}]$ and amplitude A_{i+1} and frequency f_{i+1} in the interval $t \in [t_{i+1}, t_{i+2}]$ then for any t between $(t_i + t_{i+1})/2$ and $(t_{i+1} + t_{1+2})/2$, the oscillatory movement of the movemaker in the direction x is determined by the functions

$$\begin{aligned} x_p(t) &= s_1(t)A_i sen\left(f_i(t-t_i)\right) + s_2(t)A_{i+1} sen\left(f_{i+1}(t-t_{i+1})\right) \\ u_p(t) &= s_1(t)A_i f_i cos\left(f_i(t-t_i)\right) + s_2(t)A_{i+1} f_{i+1} cos\left(f_{i+1}(t-t_{i+1})\right) \end{aligned} \quad (16)$$

where $s_1(t) = 0,5(-tanh\left((t-t_i)k\right)+1)$, $s_2(t) = 0,5(tanh\left((t-t_i)k\right)+1)$ are smoothing functions and $k = max(f_i, f_{t+1})$.

Figure 5 shows the displacement of the wavemaker and the velocity. It is observed that no discontinuity in the velocity is found in the transition between the different cycles.

The progress of flow in different moments is illustrated in Fig. 6, where it can be oberserved which breaking waves are detected.

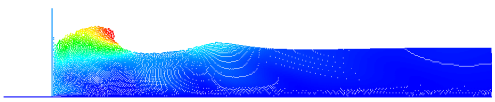
The results are compared to experimental data found in (Cox and Ortega, 2002), where Fig. 7 shows the elevation at 7, 8, 9 and 10 m of the origin, respectively. As noted in the comparison, the numerical result are satisfactory in phase and amplitude. It can be oberserved that more distant of the origin, greater the difference in amplitude. This occurs because the SPH is a dissipative method. This dissipative process was also reported by (Zheng and Duan, 2010).

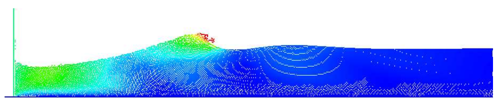
4. CONCLUSION

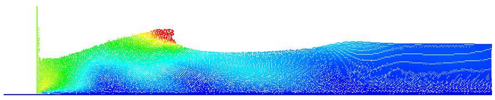
This work is the initial part of a study to determine erosion. In this moment, the code was used to study the free surface which has characteristics that make it difficult to simulate using the conventional numerical methods. The cases studied was the dam break and irregular wave. In both the code presented satisfactory results. It shows wich the Smoothed Particle Hydrodynamics (SPH) method gives good results to simulate flows with free surface feature.

5. REFERENCES

Batchelor, G.K., 1967. *An Introduction to Fuid Dynamics*. Cambridge University, Cambridge.


(a) 2,41 s


(b) 2,96 s


(c) 4,16 s

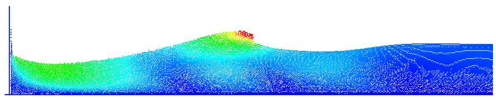
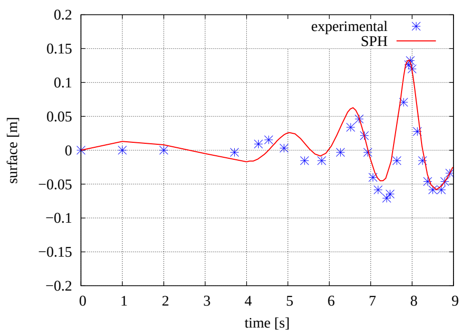
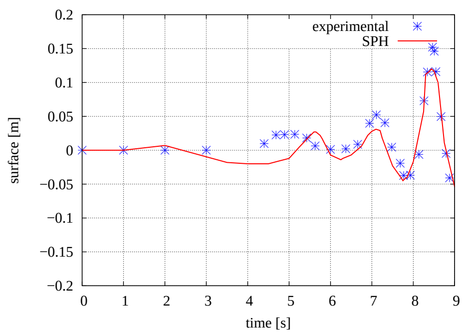

(d) 6,35 s

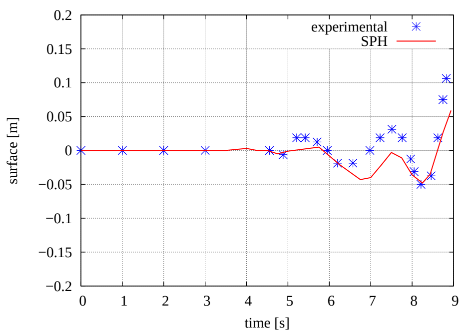
Figure 6. Flow at different instants.



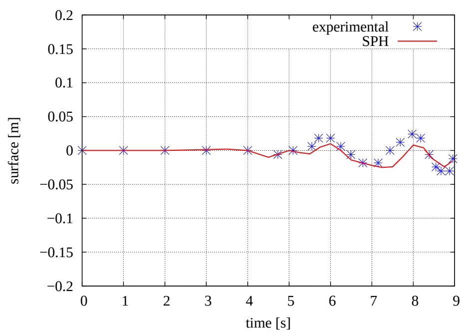
(a) 7 m



(b) 8 m



(c) 9 m



(d) 10 m

Figure 7. Surface elevation at 7, 8, 9 and 10 m of the origin.

Colagrossi, A. and Landrini, M., 2003. "Numerical simulation of interfacial flows fy particle hydrodynamics". *Journal of Computational Phusics*, , No. 191, pp. 448–475.

Cox, D.T. and Ortega, J.A., 2002. "Laboratory observations of green water overtoppin a fixed deck". *Journal of Ocean Engieering*, , No. 29, pp. 1827–1840.

Dalrymple, R.A. and Rogers, B.D., 2006. "Numerical modeling of water waves with the sph method". *Coastal Engineering*, Vol. 53, pp. 141–147.

Gotoh, H., Shao, S. and Memita, T., 2004. "Sph-les model for numerical investigation of wave interaction with partially immersed breakwater". *Coastal Engineering Journal*, Vol. 46, No. 1, p. 36.63.

Gray, J.P., Monaghan, J.J. and Swift, R.P., 2001. "Sph elastic dynamics." *Computer methods in applied mechanics and engineering*, Vol. 190, pp. 6641–6662.

Gómez-Gesteira, M., Cerqueiroa, D., Crespo, C. and Dalrymple, R.A., 2005. "Green water overtopping analyzed with a sph model". *Ocean Engieering*, Vol. 32, pp. 223–238.

Hosseini, S.M., Manzari, M.T. and Hannani, S.K., 2007. "A fully explicit three-step sph algorithm for simulating of non-newtonian fluid flow". *International Journal of Numerical Methods for Heat and Fluid Flow*, Vol. 17, No. 7, pp. 715–735.

Hu, X. and Adms, N.A., 2007. "An incompressible multi-phase sph method". *Journal of Computational Physics*, Vol. 227, No. 1, pp. 264–278.

Idelsonh, R.S., Oñate, E. and Del Fin, F., 2004. "The particle finite element method: a powerful tool to solve incompressible flows with free-surface and breaking waves". *International Journal for Numerical Methods in Engineering*, , No. 61, pp. 964–989.

Koshizuka, S. and Oka, Y., 1996. "Moving paricle semi-implicit method for fragmentation for compressible fluid". *Nuclear Science Enginnering*, , No. 123, pp. 421–434.

Liu, G.R. and Liu, M.B., 2003. *Smoothed Particle Hydrodynamics - a meshfree particle method*. World Scientific, New Jersey.

Lucy, L.B., 1977. "Numerical approach to testing the fission hypothesis". *Astronomical Journal*, , No. 82, pp. 1013–1024.

Monaghan, J.J., 1992. "Smoothed particle hydrodymaics". *Annual Review of Astonomical and Astrophysics*, Vol. 30, pp. 543–574.

Monaghan, J.J., 1994. "Simulating free surface flows with sph". *Journal of Computational Physics*, , No. 110, pp. 399–406.

Monaghan, J.J., 2002. "Sph compressible turbulence". *Royal Astronomical Society*, Vol. 335, pp. 843–852.

Monaghan, J.J., 2005. "Smoothed particle hydrodymaics". *Reports on Progress in Physics*, , No. 68, pp. 1703–1759.

Monaghan, J.J. and Kos, A., 1999. "Solitary waves on a cretan beach". *Journal Waterway, Por, Coastal and Ocean Engineering*, Vol. 125, pp. 145–154.

Morris, J.P., Fox, L.P. and Zhu, Y., 1997. "Modeling low reynolds number incompressible flows using sph". *Journal of Computational Physics*, Vol. 136, pp. 214–226.

Violeau, D. and Issa, R., 2006. "Numerical modelling of complex turbulent free-surface flows with the sph method: an overview". *International Journal for Numerical Methods in Fluids*, , No. 53, pp. 277–304.

Zheng, X. and Duan, W., 2010. "Numerical simulation of dam breaking using smoothed particle hydrodynamics and viscosity behavoir". *Journal of Marine Science and Apllication*, Vol. 9, pp. 34–41.

6. RESPONSIBILITY NOTICE

The authors are the only responsible for the printed material included in this paper.