

The following paper presented by Vijay Kanth Ponnada is a part of course work ([seminar](#)) in the Masters Program, Computational Sciences in Engineering, Technical University, Braunschweig. Presented on 20 December, 2001.

# Moving Partice Semi-Implicit Method (MPS Method)

## 1 Introduction

The Moving Particle Semi-Implicit (MPS) method is introduced as a Lagrangian formulation for approximating the Navier-Stokes equations for incompressible flow. Particle interaction models are prepared for differential operators, such as gradient, divergence and Laplacian. Governing equations are transformed to interactions among moving particles. Hence the simulation does not require a mesh.

In the numerical studies of breaking waves, tracking the free-surface movement is difficult. In tracking the free surface in [xyz], marker-and-cell (MAC) [7] method works well for the breaking waves. In the MAC method, many markers which follow the flow velocities are introduced to track the free-surface movement. The cells that contain markers which have marker-empty cells as their neighbors define the free surface. To track the free-surface accurately, the MAC method requires many markers, more than 10 per each grid cell. Hence, Hirt and Nichols [8] proposed the volume of fluid (VOF) concept as a general extension of the MAC method by introducing the F-function to describe the fractional volume of fluid in the computational grid cells. The VOF concept has led to an improvement in the computational efficiency in tracking the free-surface compared to the MAC method.

In both the MAC method and VOF method, the Navier-Stokes equations are solved on a fixed Eulerian grid. Hence, advection terms arise, resulting in numerical diffusion. This is a severe problem especially for the simulation of the fragmentation and coalescence of the fluid at free surface like plunging breakers. Both the MAC method and the VOF method introduce Lagrangian concepts to overcome these difficulties.

As for the fluid dynamics, a Lagrangian method for the discretisation of the Navier-Stokes equations, so called *particle method*, has been proposed. The moving particle semi-implicit (MPS) method proposed by Koshizuka, Tamako and Oka [2] has a simple and straightforward

algorithm with sufficient numerical stability. Koshizuka, Nobe and Oka [3] applied the MPS method to the breaking waves on a uniform slope.

In the present paper, the governing equations are transformed to particle interactions among moving particles according to the Lagrangian formulation and then results from numerical experiments are shown: the collapse of a water column, breaking waves at several ground configurations, namely a uniform slope, and a permeable uniform slope.

## 2 The MPS Method

### 2.1 Governing equations

The governing equations are the continuity equation and the Navier-Stokes equation :

$$\nabla \cdot \mathbf{u} = 0, \quad (1)$$

$$\frac{\partial \mathbf{u}}{\partial t} + (\mathbf{u} \cdot \nabla) \mathbf{u} = -\frac{1}{\rho} \nabla p + \nu \nabla^2 \mathbf{u} + \mathbf{g} + \mathbf{F}, \quad (2)$$

where  $\mathbf{u}$  = velocity;  $p$  = pressure;  $\mathbf{F}$  = external force vector;  $\mathbf{g}$  = gravitational acceleration vector;  $\nu$  = kinematic viscosity;  $\rho$  = density.

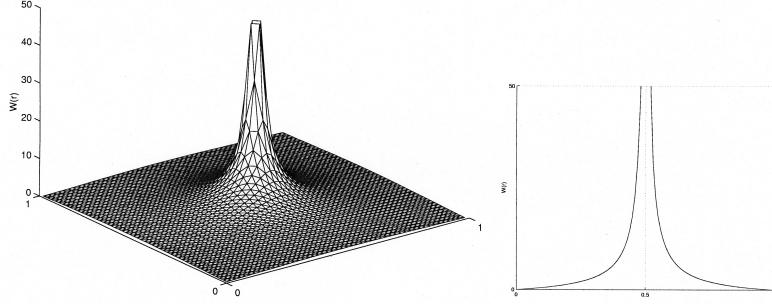
In the MPS method a fluid is represented by moving particles with constant mass. Convection results from the motion of these particles. Thus, numerical diffusion, which is a problem for Eulerian approaches, does not take place.

### 2.2 Particle Interaction Models

A *Kernel Function*  $w(r)$  is used for all the interaction models to restrict the particle interactions within a finite radius  $r_e$  :

$$w(r) = \begin{cases} \frac{r_e}{r} - 1 & \text{if } (0 \leq r < r_e) \\ 0 & \text{if } (r_e \leq r), \end{cases} \quad (3)$$

where  $r$  is the distance between two particles  $i$  and  $j$ .



*Fig 1. The Kernel function.* from [6]

The particle number density  $n$  for particle  $i$  is defined as

$$n_i = \sum_{j \neq i} w(|\vec{r}_j - \vec{r}_i|). \quad (4)$$

The incompressibility of the fluid is guaranteed by keeping the particle number density of the fluid constant. The continuity equation is also satisfied automatically by keeping the total number of particles and the mass of individual particles constant.

A gradient vector with respect to a scalar quantity  $\phi$  can be obtained between two particles  $i$  and  $j$  at coordinates  $r_i$  and  $r_j$  as

$$(\phi_j - \phi_i)(|\vec{r}_j - \vec{r}_i|)/|\vec{r}_j - \vec{r}_i|^2. \quad (5)$$

where  $\phi_i = \phi(r_i)$ .

The gradient vector at the particle  $i$  is computed as the weighted average of these gradient vectors

$$\langle \nabla \phi \rangle_i = \frac{d}{n_i} \sum_{j \neq i} \left[ \frac{\phi_j - \phi_i}{|\vec{r}_j - \vec{r}_i|^2} (\vec{r}_j - \vec{r}_i) w(|\vec{r}_j - \vec{r}_i|) \right]. \quad (6)$$

where  $d$  is the number of space dimensions.

In the MPS method, the Laplacian of  $\phi$  is discretised by distributing a part of  $\phi$  and then averaging from a particle  $i$  to its neighboring particles by the kernel function.

$$\langle \nabla^2 \phi \rangle_i = \frac{2d}{\lambda n^0} \sum_{j \neq i} (\phi_j - \phi_i) w(|\vec{r}_j - \vec{r}_i|), \quad (7)$$

where  $\lambda$  is a parameter which is chosen so that the variance increase is equal to that of the analytical solution.

$$\lambda = \frac{\int_V w(r) r^2 dv}{\int_V w(r) dv}. \quad (8)$$

Here,  $r_e = 2.1 \lambda_0$  is used for the gradient model and the particle number density and  $r_e = 4.0 \lambda_0$  for the Laplacian model [4].  $\lambda_0$  is the distance between two particles in the initial configuration.

### 3 Algorithm

A semi-implicit algorithm is used in the MPS method. Viscosity and external force terms in the Navier-Stokes equation are explicitly calculated to obtain the temporary velocities,  $\vec{u}_i^*$  and the temporary coordinates,  $\vec{r}_i^*$  in each time step and the particles are moved according to

$$\vec{u}_i^* = \vec{u}_i^n + \Delta t (\mathbf{v} \nabla^2 \vec{u}_i^n + \vec{g}), \quad (9)$$

$$\vec{r}_i^* = \vec{r}_i^n + \Delta t \vec{u}_i^*. \quad (10)$$

where  $\Delta t$  is the time step of the calculation and  $i$  shows the step of the  $i$  th calculation. The continuity equation is implicitly calculated with the pressure gradient term in the Navier-Stokes equation:

$$\nabla \cdot \vec{u}'_i = -\frac{1}{\Delta t} \frac{\rho_i^{n+1} - \rho_i^*}{\rho^0} = -\frac{1}{\Delta t} \frac{\langle n \rangle_i^{n+1} - \langle n \rangle_i^*}{n^0}, \quad (11)$$

Here, the fluid density is considered to be proportional to the particle number density.

$$\vec{u}'_i = -\frac{\Delta t}{\rho} (\nabla P_i^{n+1}), \quad (12)$$

where  $\vec{u}'_i = \vec{u}_i^{n+1} - \vec{u}_i^*$ . Combining equations (11) and (12), we have the poisson equation of pressure:

$$\langle \nabla^2 P \rangle_i^{n+1} = \frac{\rho}{\Delta t^2} \left\{ \frac{n_i^{n+1} - \langle n \rangle_i^*}{n^0} \right\}. \quad (13)$$

The right side of equation (13) represents by a deviation of the particle number density from the constant value, while usually velocity divergence is used in grid methods. The left-hand side is discretised by the Laplacian model. Finally we have simultaneous equations expressed by a linear symmetric matrix. These equations are solved by the conjugate gradient method preconditioned by a incomplete Cholesky decomposition. The pressure gradient terms are calculated from the gradient model, where the scalar  $\phi$  is substituted by  $P^{n+1}$  and is used to modify the velocities and coordinates of the particles.

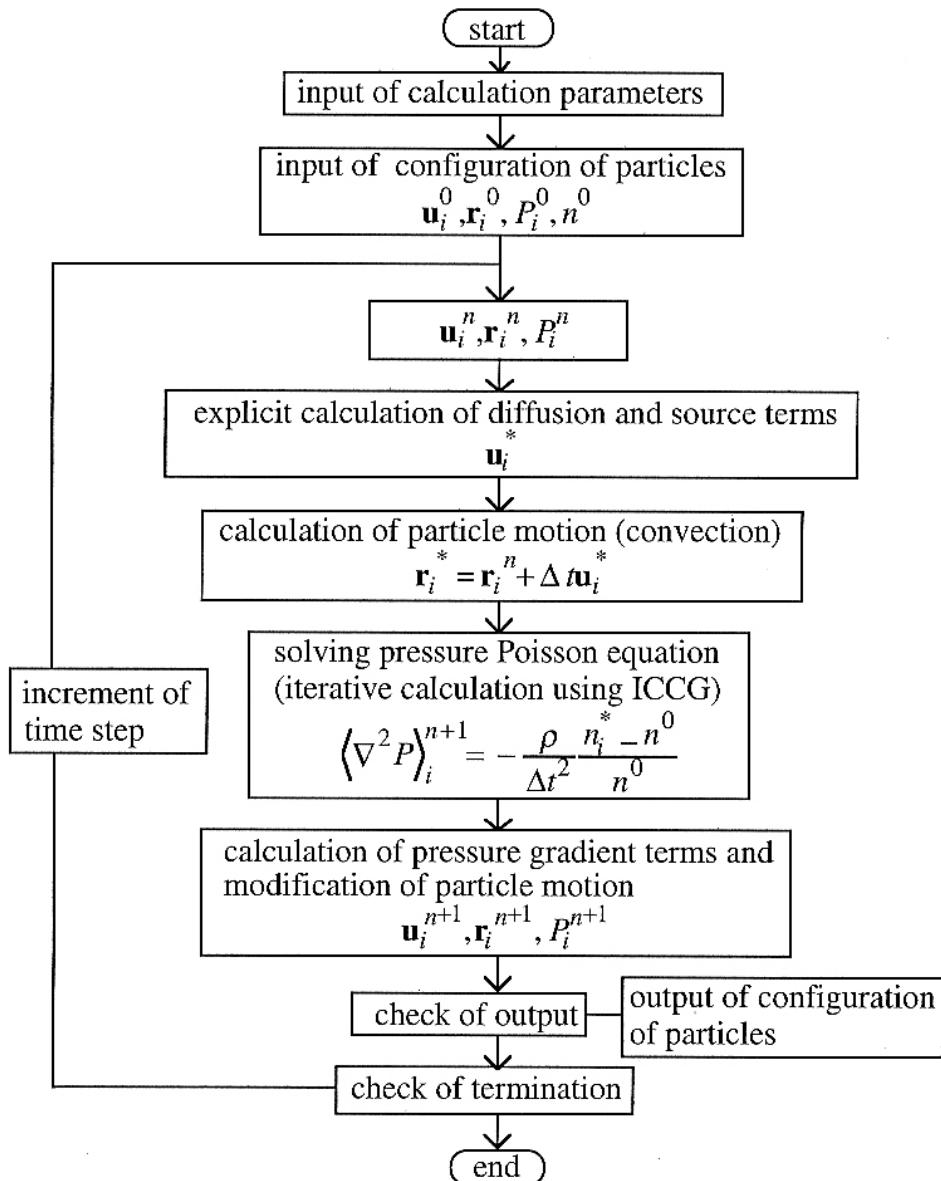
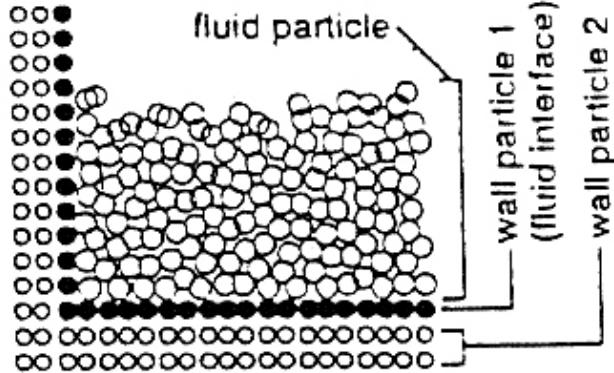


Fig 2. Algorithm of MPS method. from [1]

## 4 Boundary conditions



*Fig 3. Wall boundaries. from [5]*

The wall boundaries are the particles having zero velocity. The fixed-wall particles interfacing with the fluid are involved in the pressure-correction calculation. Also the number densities are calculated at these particles. For the calculation of the number density at the particles on the inner first wall of the wall, two other lines of particles are added outside. Without these dummy wall particles, the number density of the particles on the first inner line of the wall are small and are recognised as free surface. At the free surface, the boundary condition of the pressure,  $P=0$  is applied. Particle  $i$  is considered to be at the free surface when

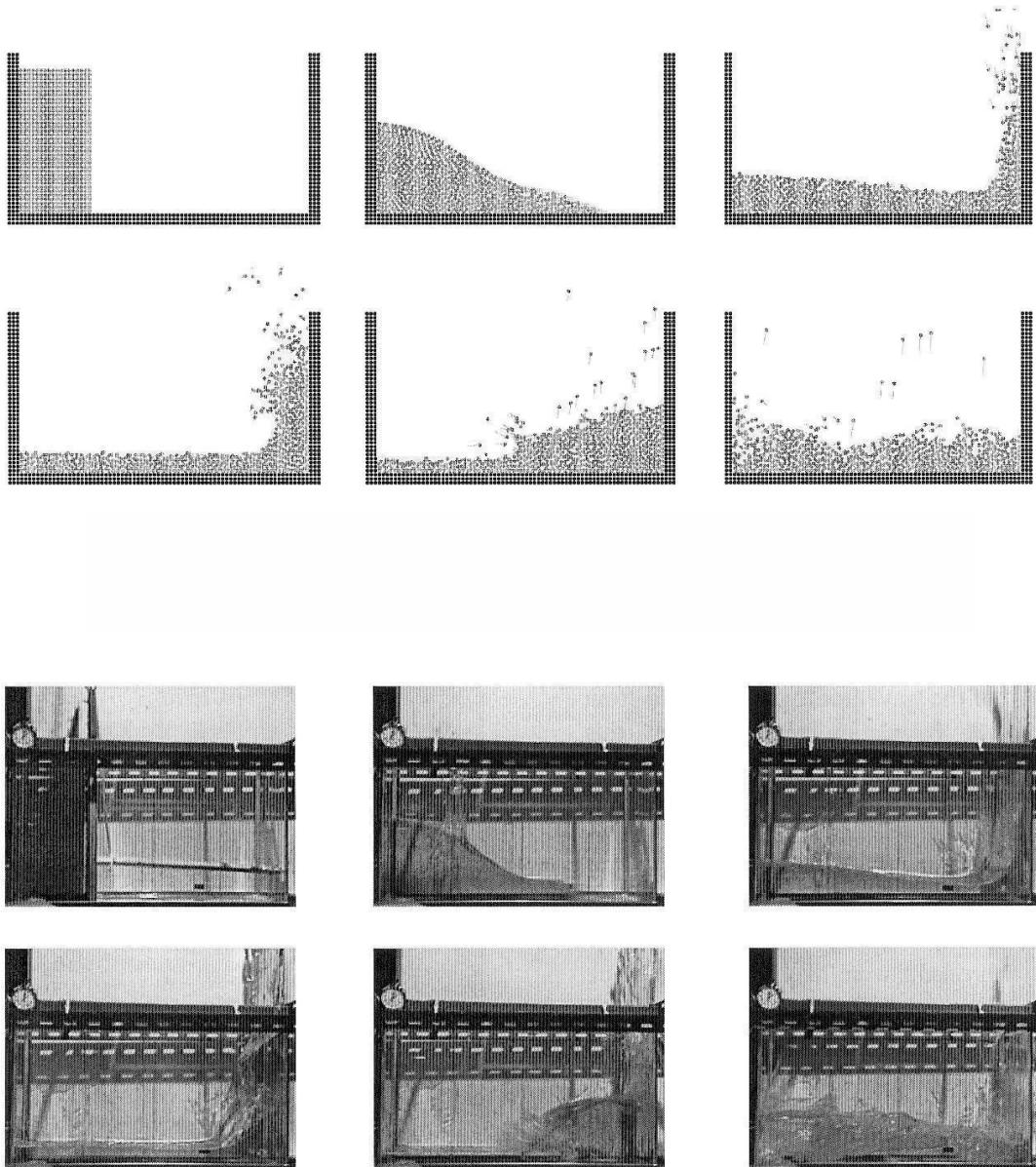
$$n_i^* < \beta \cdot n_0,$$

Koshizuka and Oka [4] determined the constant  $\beta = 0.97$ .

## 5 Examples

### 5.1 Collapse of water column

The collapse of water column is a classical example to test the validation of Lagrangian formulations in incompressible fluid flows. The calculation is carried out with the MPS method and compared with an experiment. The water is initially located in the left, supported by a removable card board. The water collapses (0.2 sec), splashes onto the right vertical wall (0.4 sec), falls down (0.6) and splashes to the left (0.8 sec) as shown in Fig.4. The results of the numerical simulation comply well with the physical experiment.

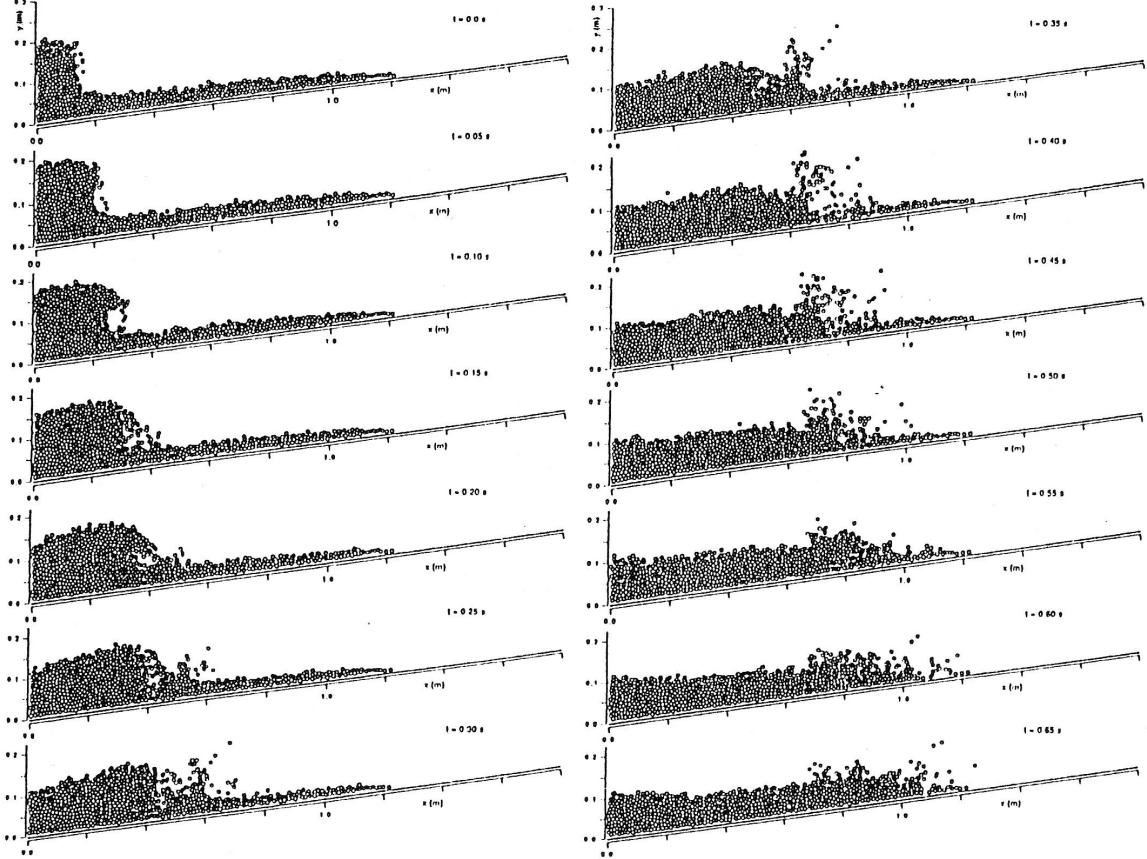


*Fig 4. Calculation and Experimental results of collapse of water column: time interval=0.2 sec. from [1]*

## 5.2 Plunging Breaker on a Uniform Slope

The calculated domain consists of 3200 particles with 1.0 cm in diameter. The vertical wall moves to generate a single cnoidal wave with 9.0 cm wave height and 1.5 s wave period. The

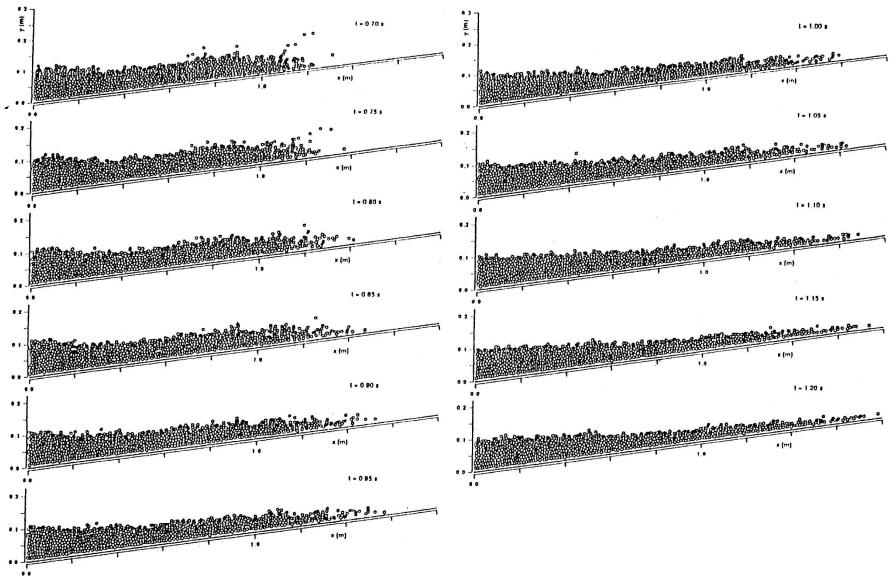
wave motion is tracked numerically on a uniform slope of 1:10.



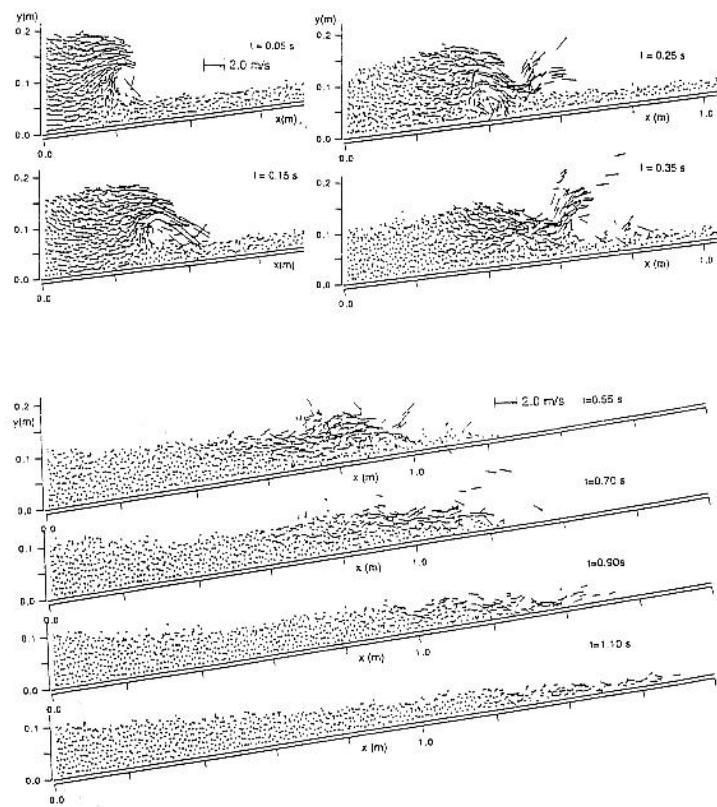
*Fig 5. Snapshots - instantaneous coordinates of particles (part1). from [5]*

Figures 7 and 8 show the instantaneous coordinates of the particles in the swash process on the slope. Figure 8 shows the instantaneous velocity vector fields to express the detailed behaviour of fluid particles for typical snapshots. At time  $t = 0.0$  s, the instance of jet generation at the wave crest is indicated.

At time  $t = 0.15$  s, the first jet falls down and at  $t = 0.25$  s, the secondary jet is induced by the swash of the first jet. At different time intervals the particles get isolated from the stream and the secondary jet hits the water surface. The height of splash and the scale of secondary wave are small compared to those of the first jet which is an indication of energy dissipation of wave.



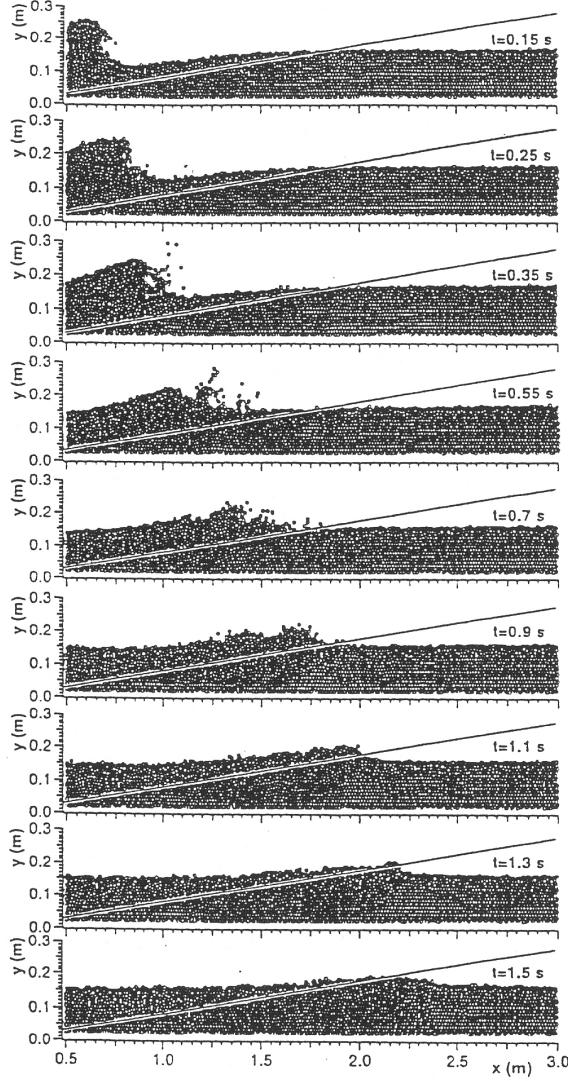
*Fig 6. Snapshots - instantaneous coordinates of particles (part2) from [5]*



*Fig 7. Instantaneous vector fields. from [5]*

### 5.3 Plunging Breaker on a Permeable Uniform Slope

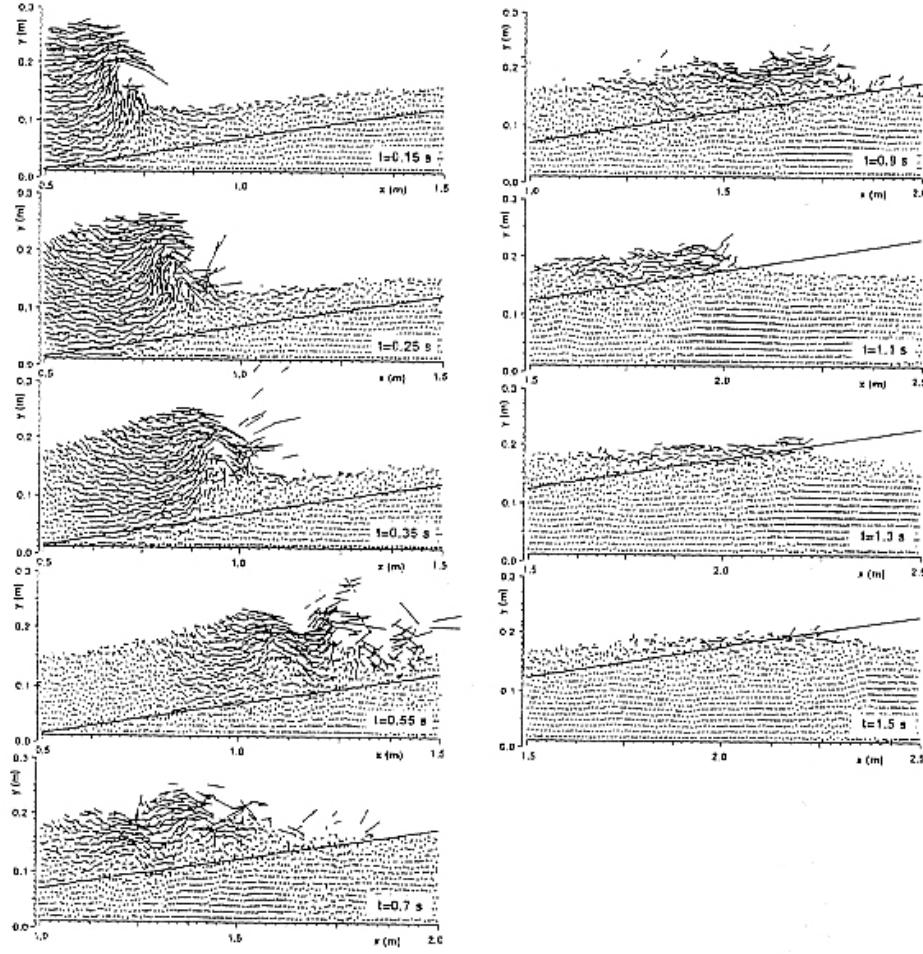
In order to introduce permeability, additional external forces, namely the drag forces are taken into account in the momentum equations [5]. The calculated domain consists of totally 7500 particles with 1.0 cm in diameter. The offshore boundary moves with a velocity which follows a single cnoidal wave with 11.8 cm wave height and 1.5 s wave period on the uniform permeable slope of 1:10.



*Fig 8. Snapshots - uniform permeable slope. from [5]*

Figures 8 and 9 show the calculated snapshots of the breaking waves on the permeable slope and the time series of the velocity fields. The intensity of the secondary jet at  $t = 0.55$  is

smaller compared to that on the impermeable slope. The scale of the wave front, which is found in  $t = 0.7$  s-1.1 s, is also smaller than that on the impermeable slope. The entrained water in the permeable layer contributes to the loss of the energy of the bore front after the wave breaking. The energy is rapidly lost when the front penetrates into the permeable slope when it comes to the elevation of the averaged water table in the permeable slope.



*Fig 9. Instantaneous vector fields - uniform permeable slope. from [5]*

## 6 Conclusions

We have considered the examples of collapse of water column and the simulation of breaking waves on a uniform slope and on a uniform permeable slope by solving the Navier-Stokes equation by the MPS method. The splash due to the wave breaking which is difficult to simulate by Eulerian methods due to numerical diffusion is simulated well by the MPS method.

The following conclusions can be made:

1. The MPS method is a useful tool in the numerical prediction of the wave breaking process.
2. This method is free from numerical diffusion due to advection terms in the Navier-Stokes equation, which is a common disadvantage in Eulerian methods.
3. The Lagrangian model takes much longer CPU time than the Eulerian model as it takes much time to generate a neighboring-particle table.

## References

- [1] Koshizuka, S., and Oka, Y. *Moving Particle Semi-implicit method : Fully Lagrangian analysis of Incompressible flows*, Proc. European Congress on Computational Methods in Applied Sciences and Engineering (ECCOMAS 2000), Barcelona, 11-14 September 2000.
- [2] Koshizuka, S., Tamako, H., and Oka, Y. (1995). *A Particle method for incompressible viscous flow with fluid fragmentation*, Comp. Fluid Dynamics J. 4, 1, pp. 29-46.
- [3] Koshizuka, S., Nobe, A. and Oka, Y (1998). *Numerical analysis of breaking waves using the Moving particle semi-implicit method* , Int. J. Numerical Meth. Fluids, 26:751-769.
- [4] Koshizuka, S., and Oka, Y (1996). *Moving particle semi-implicit method for fragmentation of incompressible fluid*, Nuclear Science Eng. 123: 421-434.
- [5] Gotoh, H., and Sakai, T.(1999) *Lagrangian Simulation of Breaking waves using particle method*, Coastal Eng J. Vol 41. Nos 3 & 4 (1999) 303-326.
- [6] Facundo DEL PIN (2000). *A Lagrangian Approach based on the Natural Neighbor Interpolation*, INRIA No. 4090.
- [7] Harlow, F. H. and Welch, J. E.(1965). *Numerical calculation of time- dependent viscous incompressible flow of fluid with free surface*, Phys. Fluid 8:2182-2189.
- [8] Hirt, C. and Nichols, B. D.(1981). *Volume of fluid (VOF) method for the dynamics of free boundaries*, J. Comput Phys. 39:201-225.