Journal of Hydroscience and Hydraulic Engineering
Vol. 20, No.1 May, 2002, 95-102

LAGRANGIAN MODEL OF DRIFT-TIMBERS INDUCED FLOOD BY USING MOVING PARTICLE SEMI-IMPLICIT METHOD

By

Hitoshi GOTOH

Associate Professor, Department of Civil Engineering, Kyoto University,
Yoshida Hon-machi, Sakyo-ku, Kyoto, 606-8501, Japan,
e-mail: gotoh@coast.kuciv.kyoto-u.ac.jp

Tetsuo SAKAI

Professor, Department of Civil Engineering, Kyoto University, Japan,
e-mail: sakai@coast.kuciv.kyoto-u.ac.jp

and

Minoru HAYASHI

Engineer, Dept. of Construction, Nishimuro Promotions Bureau, Wakayama Prefectural Government,
Asahigaoka 23-1, Tanabe-shi, Wakayama, 646-8580, Japan,
e-mail: hayashi_m0007@office.wakayama.go.jp

ABSTRACT

A flood caused by a dam-up of drift timbers at a small bridge in a mountain stream is simulated by attaching the subroutine for tracking the motion of rigid body as a model of drift timbers to the MPS method. The mass conservation of fluid-timber mixture, the fluid-timber interaction, the inter-timber collision, and the fragmentation and coalescence of fluid including water surface waves and splash are described clearly by the present model. The complicated time-dependent process of the dam-up induced flood on a small bridge is simulated effectively.

INTRODUCTION

A dam-up process of drift timbers and a resultant flood are one of the primary events of disasters in mountain streams. Recently, in Japan, a potential of the generation of drift timbers increases due to the increase of the fallen trees on mountainsides. Uncared artificial coniferous forests with Japanese cedars and cypresses, which cover even steep slopes on high mountainsides, also contribute to the increasing potential of the generation of drift timbers. Fallen trees, which were once deposited on a mountain streams, are picked up by a flood flow after a heavy rainfall, and flow down to rivers as drift timbers. An appropriate management of artificial forests is an essential factor in decreasing a generation of drift timbers. By looking at the present situation of many uncared artificial forests, the change of flow field by containing drift timbers and the resultant change of drag force exerted on hydraulic structures should be estimated with sufficient accuracy.

In spite of the fact that studies of the flow field with drift timbers have been necessary, few studies have been made on the modeling of flow with drift timbers. Nakagawa et al. (1,2) treated the drift-timber transport by floods and dam-up process of drift timbers by buildings in an inundated area. Nakagawa et al. tracked the motion of individual timbers based on the equation of motion of timbers without considering the timber-timber interaction, or a collision, in the flow field solved by the Eulerian model. They estimated the time-dependent changes of the timber distribution in an inundated area and compared them with the experiment.

In this study, a series of the pre-flooding events, such as an interaction between timbers and fluid flow, and a dammed-up timbers induced overflow at a small bridge in a mountain stream, are simulated. Nakagawa et al. (1,2) analyzed the dispersion of drift timbers in the horizontal 2D field, while, in this study, the vertical 2D field should be the computational domain. To describe the drift-timber induced dam-up and flooding process at a small bridge in a mountain stream, the required sub-models are as follows: (i) the mass conservation of fluid-timber mixture; (ii) the fluid-timber interaction; (iii) the inter-timber collision; (iv) behavior of free surface including wave breaking; and (v) the trapping of timbers by a bridge.

The ordinary Eulerian approaches have a difficulty in keeping the resolution of water surface with sufficient accuracy, especially where there is fragmentation and coalescence of water. One of the most useful tools which overcome this difficulty is the particle method. The particle method is categorized as a gridless analysis, which is free from the numerical diffusion due to the discretization of advection terms in the Navier-Stokes equation; hence the complicated shape of a water surface can be tracked with sufficient accuracy. The application of the particle method to the hydrodynamic phenomena began when Koshizuka et al.(3) proposed the MPS (Moving Particle Semi-implicit) method as a solver of the Navier-Stokes equation. Koshizuka et al.(4) and Gotoh and Sakai(5) conducted the simulation of the wave breaking on a uniform slope. Hayashi et al.(6) carried out the simulation of the wave breaking and the resultant wave overtopping at a vertical seawall. Gotoh and Fredsøe(7) simulated the dispersion of sediment dumped into water by the two-phase flow model on the basis of the MPS method. Gotoh et al.(8) proposed the SPS, or Sub-Particle Scale, turbulence model for the MPS method and simulated the mixing process around a jet.

In this paper, the module for tracking a single passively moving solids, which has been proposed by Koshizuka et al.(4), is extended to multiple solids. The extended MPS method with the model of multiple moving solids is applied to the flow with drift timbers.

SIMULATION MODEL

Governing Equations

Here the MPS method for discretizing the governing equations, or the continuity equation and the momentum equation, is explained briefly (for the details of the MPS method, see Koshizuka et al.(3)). A fluid, or water, is modeled as an assembly of the particles, around which the interaction zone is assumed. All of the terms in governing equations are described as the interaction of neighboring particles; hence the computational grid is not required.

The continuity equation,

$$\nabla \cdot \mathbf{u} = 0 \tag{1}$$

in which $\mathbf{u}$=velocity vector, is satisfied by keeping the total number of particles, the mass of which is a constant, to be a constant. The incompressibility of fluid is satisfied by keeping the number density of fluid particle constant n_0, because the density of the fluid is proportional to the number density of fluid particles.

The Navier-Stokes equation,

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

in which p=pressure; ρ=density of fluid; $\mathbf{g}$=gravitational acceleration vector; and ν=kinematic viscosity, is modeled based on the information of the neighboring particles without introducing computational grids.

Gradient model and Laplacian model

The advection terms are calculated automatically in the tracking procedure of the particles. This simple process of the advection terms calculation can effectively suppress the numerical diffusion, which causes a serious problems when calculating water-surface profiles by the Eulerian methods.

The pressure gradient term and the viscous term, or Laplacian, of the particle i are modeled as follows:

$$-\frac{1}{\rho}\langle \nabla p \rangle_i = -\frac{1}{\rho}\frac{D_0}{n_0}\sum_{j \neq i}\left\{\frac{p_j - p_i}{\left|\mathbf{r}_j - \mathbf{r}_i\right|^2}\left(\mathbf{r}_j - \mathbf{r}_i\right) \cdot w\left(\left|\mathbf{r}_j - \mathbf{r}_i\right|\right)\right\} \tag{3}$$

$$\nu\langle \nabla^2 \mathbf{u} \rangle_i = \frac{2\nu D_0}{n_0 \lambda}\sum_{j \neq i}\left(\mathbf{u}_j - \mathbf{u}_i\right) w\left(\left|\mathbf{r}_j - \mathbf{r}_i\right|\right) \tag{4}$$

$$\lambda = \frac{\sum_{j \neq i} w(|\mathbf{r}_j - \mathbf{r}_i|)|\mathbf{r}_j - \mathbf{r}_i|^2}{\sum_{j \neq i} w(|\mathbf{r}_j - \mathbf{r}_i|)} \tag{5}$$

in which $\mathbf{r}$=coordinate vector of fluid particle; D_0=number of space dimensions; and $w(r)$=the kernel function. Koshizuka and Oka (9) expressed the kernel function as follows:

$$w(r) = \begin{cases} \frac{r_e}{r} - 1 & \text{for } r \leq r_e \\ 0 & \text{for } r > r_e \end{cases} \tag{6}$$

in which r=distance from the center of the target particle; and r_e= diameter of the kernel.

This function, which has the finite interacting zone, is effective in that it saves the computational memory. The number density is defined in the following form by using kernel function.

$$n_i = \sum_{j \neq i} w(|\mathbf{r}_j - \mathbf{r}_i|) \tag{7}$$

Procedure of computation and boundary conditions

The iterative process of the MPS method consists of two sub-processes. The first process is an explicit calculation performed to obtain the temporal velocities under the given sets of viscosity and gravity terms. In the temporal field calculated in the first process, the mass conservation, namely, the agreement of the number densities with the initially defined one n_0, is not satisfied. Therefore, the second process, in which the number densities are corrected to satisfy the mass conservation, is required. In the second process, the pressure term, which was omitted in the first process, is considered. Thus the following Poisson equation of pressure is obtained.

$$\nabla^2 p_{l+1} = -\frac{\rho}{(\Delta t)^2} \frac{n^*_l - n_0}{n_0} \tag{8}$$

in which n^*=temporal number density; Δt=time step of the calculation; and the subscript l shows the step of the l-th calculation. The pressure field is updated by solving this Poisson equation implicitly. Then the velocities of particles and the coordinates of particles are corrected.

The wall boundaries are constituted by the lines of the fixed particles having zero velocity. The particles on the inner first line of the wall, which are directly contacting with the fluid particles, are involved in the pressure-correction calculation. Few lines of particles are required behind the inner first lines of particles for the calculation of number density of particles on the first inner line. Without these dummy wall particles, the number density of the particle on the inner first line of the wall becomes small, and wall particles are recognized as the free surface. At the free surface, the boundary condition of the pressure should be applied. The free surface is assessed based on the number densities as follows:

$$n^*_i < \beta \cdot n_0 \tag{9}$$

Koshizuka and Oka (9) determined the constant β=0.97. This condition for the free surface assessment is very simple and stable even for the fragmentation and coalescence of water such as plunging breakers and splashing.

Model of passively moving solid

The drift timbers can be regarded as solids in comparison of their resistance to deformation to that of water. Here, the motion of timbers are tracked by using the model of passively moving solids proposed by Koshizuka et al.(4). At first, the same calculation procedure is applied to both of the fluid and solid particles. In other words, the conjunction among individual solid particles is not considered at this stage. As a result of this calculation, solid deforms, hence the relative locations of the solid particles are corrected in the following way.

The translational and rotational velocity vectors $\mathbf{T}_k$ and $\mathbf{R}_k$ of k-th timber are calculated as follows:

$$\mathbf{T}_k = \frac{1}{N_k}\sum_{i=1}^{N_k}\mathbf{u}_{ki} \tag{10}$$

$$\mathbf{R}_k = \frac{1}{I_k}\sum_{i=1}^{N_k}\mathbf{u}_{ki}\times\left(\mathbf{r}_{ki}-\mathbf{r}_{kg}\right) \tag{11}$$

in which, N_k= number of solid particles constituting the k-th timber. The gravity center $\mathbf{r}_{gk}$ and the moment of inertia I_k of k-th timber are given by

$$\mathbf{r}_{kg} = \frac{1}{N_k}\sum_{i=1}^{N_k}\mathbf{r}_{ki} \tag{12}$$

$$I_k = \sum_{i=1}^{N_k}\left|\mathbf{r}_{ki}-\mathbf{r}_{kg}\right|^2 \tag{13}$$

Then, the velocity vectors of timber-constituting particles are replaced by

$$\tilde{\mathbf{u}}_{ki} = \mathbf{T}_k + \left(\mathbf{r}_{ki}-\mathbf{r}_{kg}\right)\times\mathbf{R}_k \tag{14}$$

to satisfy the motion as a rigid body. By applying the above mentioned velocity-vector-correction procedure to only the timber-constituting particles, the motion of timbers can be tracked. The density of timber-constituting particles is replaced by the density σ.

Domain and conditions of calculation

The calculated domain is approximately 140 m long and the initial depth of the flow is 1.0 m, as shown in Fig. 1. The number of particles is 20,000: 15,000 of particles for water, and 5,000 of particles for solid wall and drift timbers. The diameter of particle is 0.1 m. Four of the drift timbers, which are 2.0 m in length and 0.4 m in thickness, are constituted by connecting 80 of particles with a specific gravity σ/ρ=0.7. The upstream boundary is the "soluble moving wall" proposed by Gotoh et al.(8), the moving velocity of which is 3.6 m/s. The "soluble moving wall", which is 40.0 m long initially, is located just upstream from the inflow boundary. With the downstreamwise motion of "soluble moving wall", some of particles at the front of "soluble moving wall" pass through the inflow boundary. Just after passing the inflow boundary, solid moving wall particles immediately change into fluid particles by changing the flag of the physical property of particles. The "soluble moving wall" is a technique to impose the constant inflow discharge at the upstream boundary. The downstream boundary is the free outflow boundary with small weir at the bottom, in the downstream of which the 30 m long water tank is located.

The slope of the channel is 1/30. The hydraulic condition with the bulk mean velocity 3.6 m/s and the averaged depth 1.0 m gives a supercritical flow with the Froude number 1.15. The Manning coefficient calculated from this hydraulic condition is 0.051, which is in the range of the flow on a gravel bottom in a mountain stream. The clearance between the water surface and the bottom of bridge is 0.1 m. A very severe situation, in which group of 0.4 m thick drift timbers rush at the bridge 0.1 m above the averaged water surface, dare to be treated in this paper, to ascertain the performance of the MPS method.

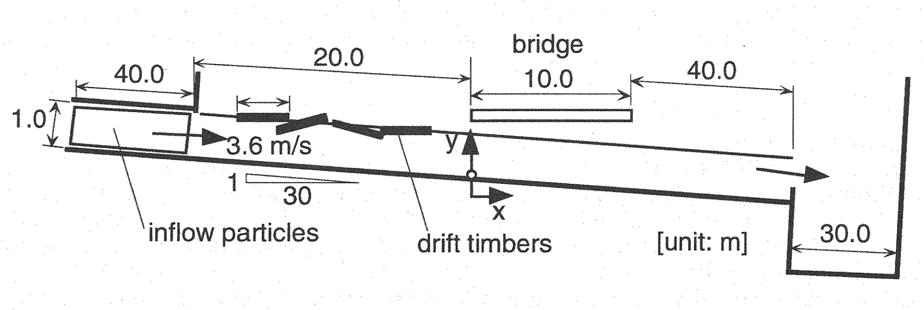

Fig. 1 Schematic expression of the calculating domain

A FLOW WITH DRIFT TIMBERS THROUGH A MOUNTAINSTREEM BRIDGE

The target of the calculation is the local flow around the upstream edge of a bridge during the passage of drift timbers, hence the closed-up figure of the instantaneous snapshot of particles in 16 m long section around the upstream edge of a bridge is shown for three cases of timber conditions.

Figure 2 shows the snapshot for the timbers with the specific gravity σ/ρ=0.7. At the time t=1.70 s, the increase of flow resistance due to existence of drift timbers causes the increase and decrease of water depth in the up-and down-stream of the drift timbers, respectively. At the time t=2.20 s, two timbers reach the edge of a bridge within a short time. The decrease of the water depth mentioned above keeps a sufficient clearance between the water surface and the bottom of bridge to pass timbers through without colliding with the bridge. Although one of the two timbers contacts with the bottom of the bridge, both of the two timbers slide through the bridge. The third and fourth timbers also slide through the bridge. Consequently, the trapping of timbers and the resultant jam do not occur in this case. At least, the drift timbers increase the flow resistance, which is found apparently as the increase of the local water surface gradient around the drift timbers. During the time t=4.20-6.20 s, some lump of water collides with the edge of the bridge and runs up onto the bridge. This may be caused by the increase of the water depth due to the backwater effect of drift timbers.

Figure 3 shows the snapshots for the jam of timbers and resultant flooding, by fixing the first and second timbers at the time t=2.20 s, when they reach the upstream edge of the bridge. The hydraulic condition in Fig. 3 is the same as that in Fig. 2. The trapping process of the drift timbers by the bridge is the 3D ones(see the report of Yosasa-gawa river drift timber disasters; URL= http:// www.aeng.tsukuba.ac.jp/ wsm_labo/yosasagawa/yosasagawa.html). The trigger of the trapping can be assumed to be a long timber trapped by the two bridge piers. To describe this situation accurately, the transverse motion, or rotation in the horizontal plane, of drift timbers should be considered; hence the 3D

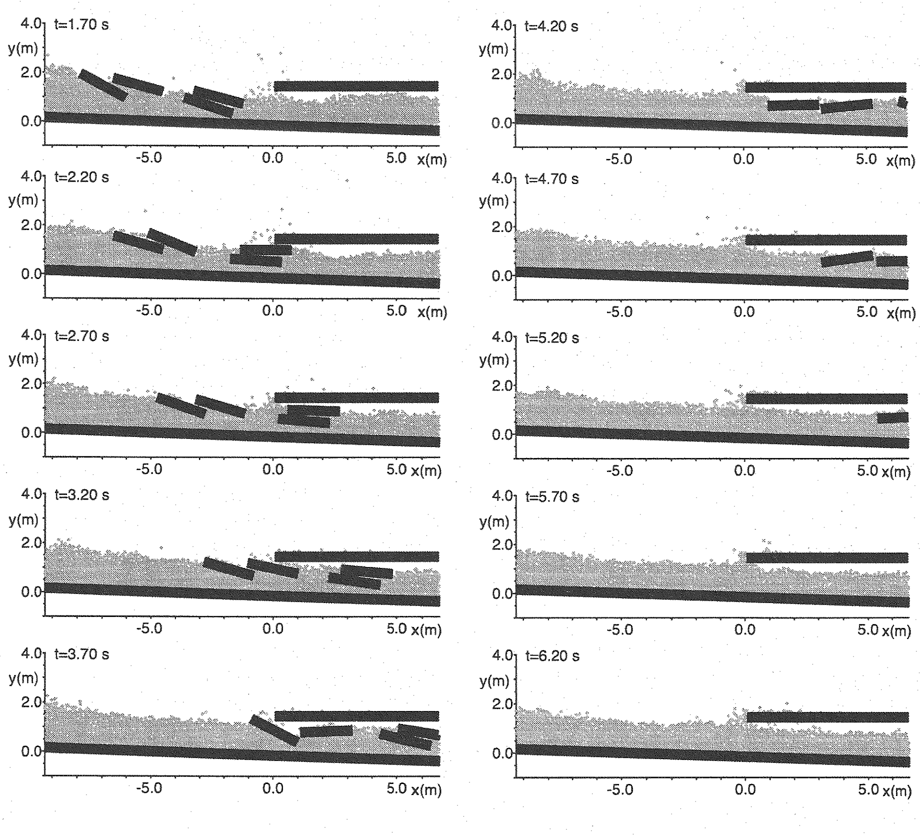

Fig. 2 Timbers passing through a bridge

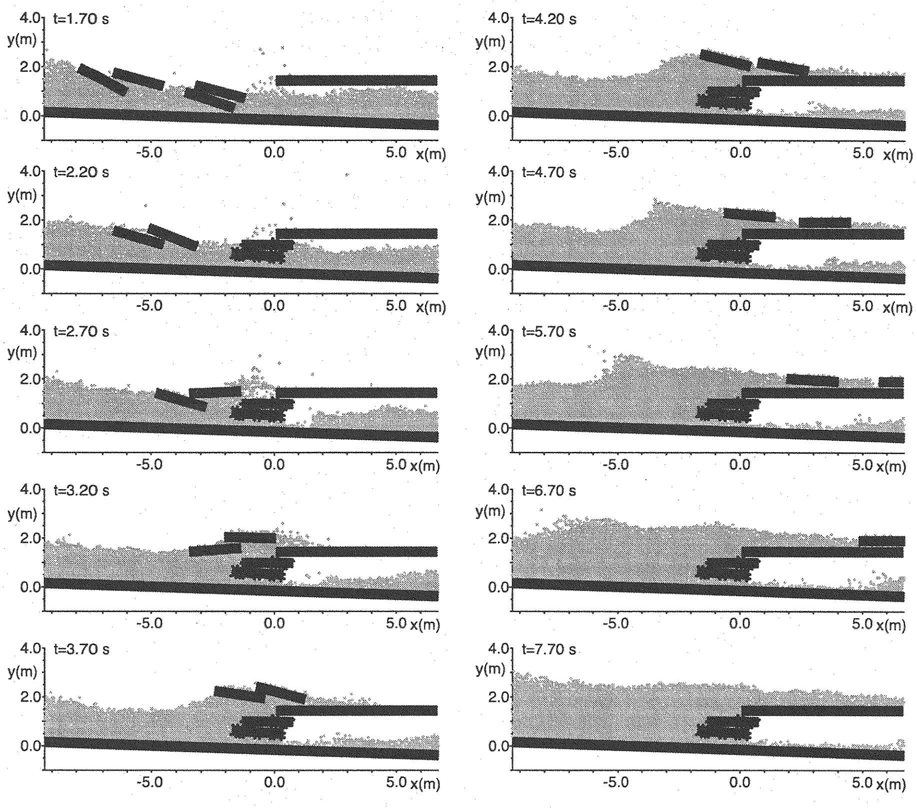

Fig. 3 Timber-jam induced flooding

model is essential. However, the limitations of the computational efficiency prevent the execution of full-3D simulation. At least, the one of the most essential factors in causing this phenomenon is the behavior of the free water surface; hence the vertical 2D MPS method is applied herein.

At the time t=2.70 s, the water depth below the bridge decreases rapidly due to the blocking of the timbers at the edge of the bridge. The jam of timbers also affects the increase in the water depth in the upstream section. The third timber reaches the jam and runs onto it with making a splash of water. After the time t=2.70 s, the increase of the water depth in the upstream section continues, then the third and fourth timbers reaching the timber jam under the upstream edge of bridge are transported by the overflow and run up to the top of the bridge. Due to the strong resistance of the timber jam, the main stream flows on the top of the bridge, transporting two timbers in a downstream direction. The snap shot after the time t=4.20 s show the upstreamwise transition of the backwater of timber jam.

Figure 4 shows the snapshot for the timbers with the specific gravity σ/ρ=0.5 under the same hydraulic condition in Fig. 2. Namely, the timbers in this case are lighter than that in Fig. 2. The small difference in the specific gravity of timbers promotes the pitching of the timber. Consequently, the timbers can be easily rotated. At the time t=2.70 s, the second timber slips under the bottom of the bridge with a high angle from the horizontal direction, and blocks the whole of the cross section below the bridge with contacting the bottom of channel and the edge of the bridge. The blocking of the cross section brings about a rapid increase of the water depth in the upstream section. Consequently, the third timber runs up to the top of the bridge. The same kind of the blocking of the entire cross section beneath the bridge can be found at the time t=5.50 s. It must be noted that the entire cross section blocking found in Fig. 4 should be the special solution of 2D model. In other words, in a real 3D field, the single timber never blocks the entire cross section beneath the bridge.

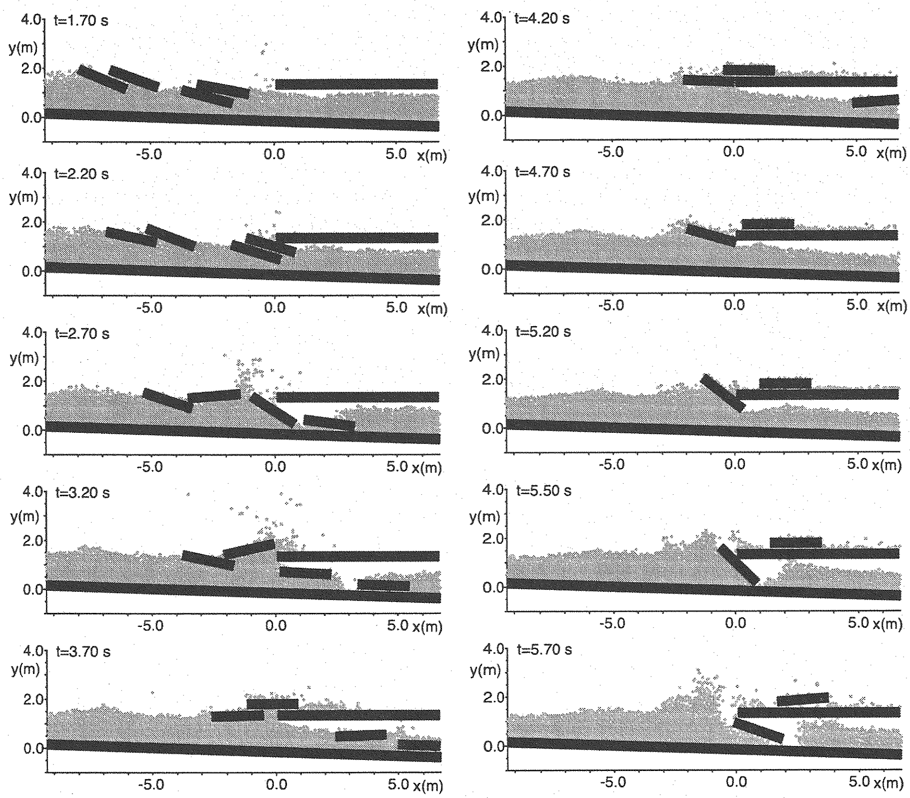

Fig. 4 Effect of specific gravity of timbers

CONCLUDING REMARKS

In this paper, the extended MPS method with the model of multiple moving solids is applied to the flow with drift timbers passing through a small bridge. Complicated behavior of a free surface and floating timbers are numerically simulated by the simple concept of Lagrangian particle method without any special sub-model of the timber-timber interaction. The robustness of this model ensures a wide range applicability of the MPS method to the hydrodynamic modeling.

Although the time series of the motion of drift timbers have been simulated, some of the contradictions in the simulated results, such as the process of the formation of timber jam and the existence of the instantaneous perfect blocking of flow by a single timber, are found in the snapshots of 2D model. These contradictions can be resolved by adopting the 3D extension of the present model, and by using the modeling of the sub-particle-scale turbulence proposed by Gotoh et al.(8). Further development of the computational technique is required to perform complete 3D turbulent flow calculations based on the MPS method.

REFERENCES

1. Nakagawa, H., Takahashi, T. and Ikeguchi M.: Drift wood diffusion by overland flood flow, *Proc. Hydraulic Eng.*, Vol. 37, JSCE, pp. 379-384, 1993(in Japanese).
2. Nakagawa, H., Inoue, K., Ikeguchi, M. and Tsubono T.: Behavior of drift wood and its dam up process, *Proc. Hydraulic Eng.*, Vol. 38, JSCE, pp. 543-550, 1994(in Japanese).
3. Koshizuka, S., Tamako, H. and Oka, Y.: A particle method for incompressible viscous flow with fluid fragmentation, *Computational Fluid Dynamics J.*, Vol. 4, No. 1, pp.29-46, 1995.

4. Koshizuka, S. Nobe, A. and Oka, Y.: Numerical analysis of breaking waves using the moving particle semi-implicit method, *Int. J. Numer. Mech. Fluids*, Vol. 26, pp. 751-769, 1998.
5. Gotoh, H. and Sakai, T.: Lagrangian simulation of breaking waves using Particle Method, *Coastal Eng. J.*, Vol. 41, Nos. 3 & 4, pp.303-326, 1999.
6. Hayashi, M., Gotoh, H., Memita, T. & Sakai, T.: Gridless numerical analysis of wave breaking and overtopping at upright seawall, *Proc. ICCE*, Sydny, Australia, pp.2100-2113, 2000.
7. Gotoh, H. & Fredsøe, J.: Lagrangian two-phase flow model of the settling behavior of fine sediment dumped into water, *Proc. ICCE*, Sydny, Australia, pp.3906-3919, 2000.
8. Gotoh, H., Shibahara, T. and Sakai, T.: Sub-Particle-Scale turbulence model for the MPS method -Lagrangian flow model for hydraulic engineering-, *Computational Fluid Dynamics J.*, Vol.9 No.4, pp.339-347, 2000.
9. Koshizuka, S. and Oka, Y.: Moving-particle semi-implicit method for fragmentation of incompressible fluid, *Nuclear Science and Engineering*, Vol. 123, pp.421-434, 1996.

APPENDIX - NOTATION

The following symbols are used in this paper:

D_0	= number of space dimensions;
d	= diameter of particle;
$\mathbf{F}$	= external force vector;
$\mathbf{g}$	= gravitational acceleration vector;
h	= water depth;
I_k	= moment of inertia of *k*-th timber;
N_k	= number of solid particles constituting the *k*-th timber;
n, n_0	= number density of fluid particle and its initial value;
n	= temporal number density of fluid particle;
p	= pressure;
$\mathbf{R}_k$	= rotational velocity vector of *k*-th timber;
$\mathbf{r}$	= coordinate vector of fluid particle;
$\mathbf{r}_{gk}$	= gravity center of *k*-th timber;
$\mathbf{T}_k$	= translational velocity vector of *k*-th timber;
t	= time;
$\mathbf{u}$	= velocity vector of fluid particle;
$w(r)$	= kernel function;
x, y	= streamwise and upward-vertical coordinates;
β	= constant for the pressure boundary condition at free surface;
Δt	= time step of calculation;
λ	= constant in Laplacian model;
ν	= kinematic viscosity;
ρ	= density of fluid; *and*
σ	= density of timber particle.

(Received August 20, 2001 ; revised January 20, 2002)