Particle Model for Simulating Flow over Large Areas

By Guangqian Wang,[1] Songdong Shao,[2] and Xiangjun Fei[3]

Abstract: On the basis of traditional particle-in-cell methods, a particle model has been developed to simulate flow over large areas. Under the assumption that the fluid medium is an assembly of many small, independent fluid particles, the momentum equation for a particle is derived for shallow-flow conditions. In the formulation used, only two forces are involved. One is the hydrostatic force arising from the accumulation of different numbers of particles at different locations. The other is a friction force that varies inversely with flow depth and quadratically with particle velocity and bed roughness. The velocity and spatial positions of all particles are averaged at fixed grid points to obtain the overall flow behavior. The particle model is demonstrated through an application to a documented 1954 flood in the Jingjiang River flood diversion area in Hubei, China. The flood lasted 300 h, with the total discharge volume being 4 billion m^3. Good agreement between computed and observed water levels was obtained. Convergence of the method is demonstrated by repeatedly doubling the number of particles employed in the computation until there is little change between simulations.

INTRODUCTION

Combinations of Lagrangian and Eulerian computing methods provide a useful tool for the study of complex flow problems. The established techniques include the ALE (Hirt et al. 1974), MAC (Harlow and Welch 1965), and PIC (Harlow 1964) methods. Because of the Lagrangian aspects of these methods they are applicable for computing flows with free surfaces or those having material interfaces. However, these methods maintain Eulerian aspects that help to overcome undesirable grid distortions often associated with purely Lagrangian techniques. The Lagrangian-Eulerian methods have been applied to simulate a variety of flow behaviors, e.g., the flow behind a broken dam or under a sluice gate, the flow around or over an obstacle, free splashing-drop flow, two-fluid flow with surface tension, and the flow of blood through flexible arteries. With regard to water flow in two horizontal dimensions, Lagrangian-Eulerian computations have been employed in estuary hydrodynamics (Cheng 1983). Moreover, observed sedimentary features have been reproduced by the 2D marker-in-cell techniques (Tetzlaff and Harbaugh 1989).

Flow problems over large areas, such as the calculation of inundation regions and the resulting water levels, are of great engineering interest once a dam break or flood diversion occurs. Although many approaches have been employed to solve such problems, 2D L-E methods are attractive owing to their simplicity and the effectiveness of the numerical procedures employed. In addition, the revival of such old techniques for the solution of new problems is interesting.

MODEL PRINCIPLES

The particle model is based on the assumption that the fluid medium is composed of many small, identical fluid particles. Assuming that the flow occurs only in the two horizontal dimensions, the motion of each particle is computed from the equation of motion. This equation is solved through a step-by-step numerical integration in the time domain. Consequently, the velocity and spatial positions of all particles are obtained and averaged at grid points to give the overall flow behavior.

Variables including the topographic elevation, the bed roughness, the flow depth, and the velocity are represented at fixed points on a 2D grid (square grid cells are employed in this paper) covering the modeled area. Variables including the velocity and spatial coordinates of particles are represented at points whose positions move with the fluid. Flow parameters at grid points represent averages within a square region surrounding each grid point. For example, the flow depth at grid points is calculated by

$$h_{ij} = \frac{N_{ij}\Delta V}{A} \tag{1}$$

where h_{ij} = flow depth at grid point ij; N_{ij} = number of particles near the grid point; ΔV = volume of one fluid particle; and A = area of the square region near the grid point, i.e., area of a grid cell.

Flow velocity at grid points is calculated as the average velocity of all particles near each grid point, i.e.,

$$\mathbf{v}_{ij} = \frac{1}{N_{ij}} \sum_{p=1}^{N_{ij}} \mathbf{v}_p \tag{2}$$

where $\mathbf{v}_{ij}$ = flow velocity at the grid point; and $\mathbf{v}_p$ = velocity of particle p.

EQUATION OF MOTION

Flow conditions are characterized as shallow when the problem possesses much larger horizontal than vertical scales. Thus the flow can be assumed to only occur in the horizontal plane. Therefore, the following momentum conservation equation in two horizontal dimensions is employed:

$$\frac{\partial \mathbf{v}}{\partial t} + (\mathbf{v}\cdot\nabla)\mathbf{v} = -g\nabla(Z + h) - g\mathbf{S}_f \tag{3}$$

where $\mathbf{v}$ = depth-averaged velocity vector; t = time; g = acceleration from gravity; Z = topographical elevation; h = flow depth; and $\mathbf{S}_f$ = hydraulic friction slope, formulated by Manning's representation

$$\mathbf{S}_f = n^2 \frac{\mathbf{v}|\mathbf{v}|}{h^{4/3}} \tag{4}$$

where n = Manning roughness of the bed.

Substituting (4) into (3) and writing the result in Lagrangian form gives

$$\frac{D\mathbf{v}}{Dt} = -g\nabla(Z + h) - gn^2 \frac{\mathbf{v}|\mathbf{v}|}{h^{4/3}} \tag{5}$$

Writing (5) algebraically (Tetzlaff and Harbaugh 1989), the

[1]Prof., Dept. of Hydr. Engrg., Tsinghua Univ., Beijing 100084, China.
[2]Grad. Student, Dept. of Hydr. Engrg., Tsinghua Univ., Beijing 100084, China.
[3]Prof., Dept. of Hydr. Engrg., Tsinghua Univ., Beijing 100084, China.
Note. Discussion open until October 1, 1998. To extend the closing date one month, a written request must be filed with the ASCE Manager of Journals. The manuscript for this technical note was submitted for review and possible publication on January 11, 1996. This technical note is part of the ***Journal of Hydraulic Engineering***, Vol. 124, No. 5, May, 1998. ©ASCE, ISSN 0733-9429/98/0005-0554–0557/$8.00 + $.50 per page. Technical Note No. 12335.

momentum equation for a fluid particle is developed as follows:

$$\frac{\mathbf{v}_{p,t+1} - \mathbf{v}_{p,t}}{\Delta t} = g\left(\mathbf{S}_{p,t} - n_{p,t}^2 \frac{\mathbf{v}_{p,t}|\mathbf{v}_{p,t}|}{h_{p,t}^{4/3}}\right) \quad (6)$$

where $\mathbf{v}_{p,t+1}$ and $\mathbf{v}_{p,t}$ = velocity of a particle at time $t + \Delta t$ and t, respectively; Δt = time step; and $\mathbf{S}_{p,t}$, $h_{p,t}$, and $n_{p,t}$ = water surface slope, flow depth, and bed roughness, respectively, at the location of a particle. The evaluation of these parameters requires interpolation between their values at grid points. A biquadratic interpolation function was used in the model applications presented below.

Let $\mathbf{a}$ be the sum of the right side of (6). The velocity of a particle is then given by

$$\mathbf{v}_{p,t+1} = \mathbf{v}_{p,t} + \mathbf{a}\Delta t \quad (7)$$

The spatial position of a particle can then be computed from

$$\mathbf{X}_{p,t+1} = \mathbf{X}_{p,t} + \frac{\mathbf{v}_{p,t+1} + \mathbf{v}_{p,t}}{2}\Delta t \quad (8)$$

where $\mathbf{X}_{p,t+1}$ and $\mathbf{X}_{p,t}$ are the spatial coordinates of a particle.

INITIAL AND BOUNDARY CONDITIONS

The topographic elevation and bed roughness at all grid points must be specified. In addition, the velocity and spatial positions of existing fluid particles, as well as the flow velocity and depth at grid points, must be specified if the modeled domain initially contains flow.

For flow calculations in flood diversion areas, two kinds of boundary conditions are required. Flow boundary conditions provide the input/output of fluid particles whose spatial positions are those of the flow inlets/outlets. The number and velocity of these particles are determined from available discharge information.

Wall boundary conditions must be prescribed at the land boundaries surrounding the flood diversion area. Fluid particles will change their directions after colliding with these boundaries. The collision process is assumed to be elastic, except that the velocity of particles after collision is reduced by half to approximate the energy loss resulting from the collision.

NUMERICAL CONVERGENCE

Because individual particles are discrete points and cannot deform like a real fluid, the number of particles employed in the calculation must be sufficiently large to give a realistic flow simulation. This requirement is essential for the numerical convergence of the particle model. Convergence is achieved by repeatedly doubling the number of particles until numerical solutions from two consecutive doubling simulations are essentially the same. This convergence testing is demonstrated below in reproducing the 1954 flood of the Jingjiang River. After computation parameters such as the horizontal grid spacing, time step, and bed roughness are specified, the model is operated by doubling the number of particles from one simulation to the next (i.e., reducing the particle volume ΔV by half) to determine the effect on the solution. The computed water levels were not stable until the number of particles in the simulation system totaled 80,000, corresponding to a ΔV of 50,000 m^3. An additional doubling of the number resulted in only slight computational differences.

MODEL APPLICATIONS

The Jingjiang River flood diversion area (Fig. 1) extends over 960 km^2, with the surrounding land boundaries being about 200 km long. The maximum length is 70 km and the maximum width is 30 km. The total drop in the topographic elevation is 10 m, with an average longitudinal slope of 1.5/10,000. The flood diversion area was initially built to store floodwaters from the well-known Yangtze River and was put into use in 1954. For the flow event modeled, the inlet flow lasted 300 h, with the total discharge volume being 4 billion m^3. The inlet discharge (Li and Huang 1993), given in Table 1, defines the flow boundary conditions. Three gauging stations, at Ma Jiazui, Huang Jinkou, and Huang Tianhu (see Fig. 1), were established in the flood diversion area in 1954 to monitor the arrival of the flood as well as water level variations. These data were used for comparison with the particle model results.

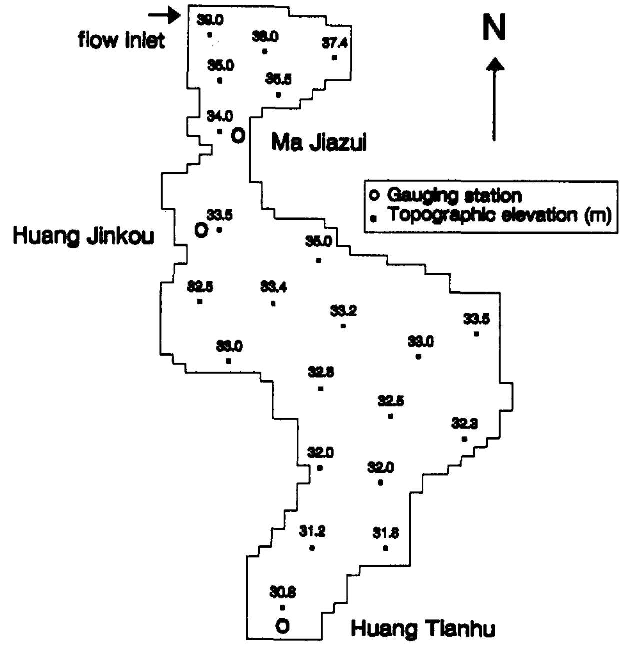

FIG. 1. Plan View of Jingjiang River Flood Diversion Area, with Total Area = 960 km^2

TABLE 1. Discharge Data at Flow Inlet

Time (h) (1)	Discharge (m^3/s) (2)	Time (h) (3)	Discharge (m^3/s) (4)
0	0	192	6,600
24	6,400	216	6,700
48	5,900	240	5,500
72	5,600	254	0
96	4,200	258	0
144	0	264	2,600
171	0	288	5,100
180	5,600	300	3,700

The computational domain is divided into 960 square grid cells with a horizontal grid spacing ΔS = 1,000 m. The evaluation of the time step Δt follows the rule that the maximum translation distance of a particle during each step should be less than the grid spacing. A value of 10 s met this requirement. Bed roughness n depends closely on the type of terrain. From the literature (Li and Huang 1993), n = 0.025 for low-resistance regions composed of rivers and ground surfaces; n = 0.05 for moderate-resistance regions composed of trees and crops; and n = 0.07 for high-resistance regions composed mainly of farmhouses. The particle volumes were taken to be ΔV = 50,000 m^3 and ΔV = 25,000 m^3, corresponding to total numbers of particles of 80,000 and 160,000, respectively.

Computed and measured water levels at the three gauging stations are shown in Figs. 2(a–c), respectively. Good agree-

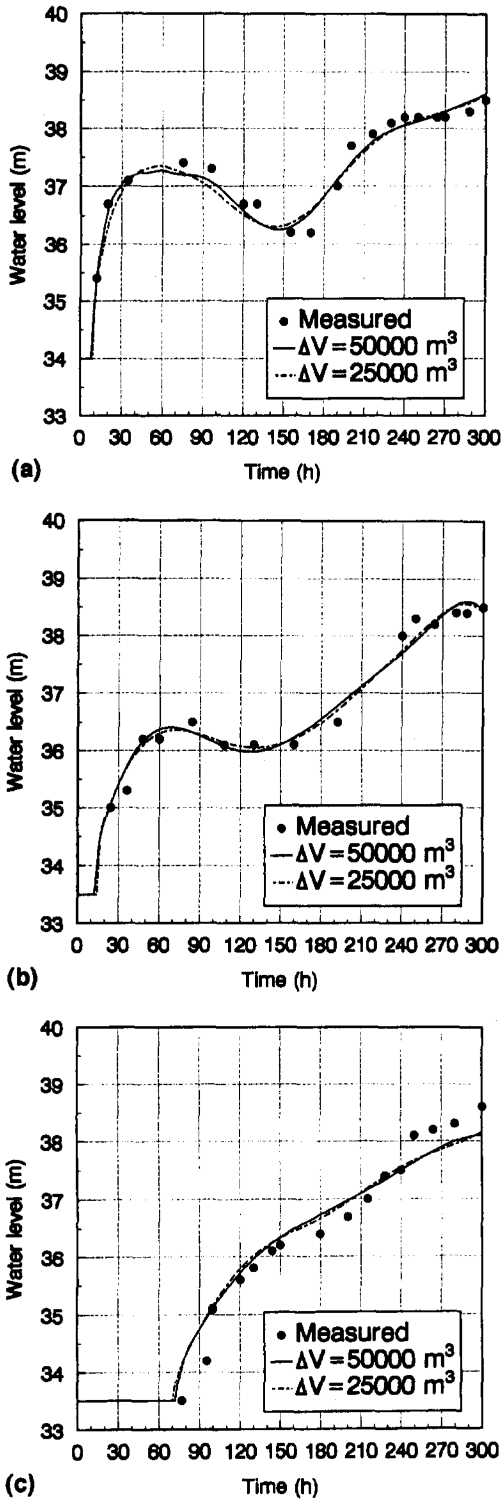

FIG. 2. Measured and Computed Water Levels at Gauging Station, with Solid Line Corresponding to ΔV = 50,000 m^3 and Dashed Line to ΔV = 25,000 m^3: (a) Ma Jiazui; (b) Huang Jinkou; (c) Huang Tianhu

ment is realized, verifying that the model behaves well. Convergence of the method is demonstrated by the small difference between the two computations. The velocity distribution at t = 102 h is shown in Fig. 3.

CONCLUSIONS

A simple and convenient method to predict the overall flow behavior in downstream flooded areas is indispensable for reducing loss of life and destruction of property. The simple

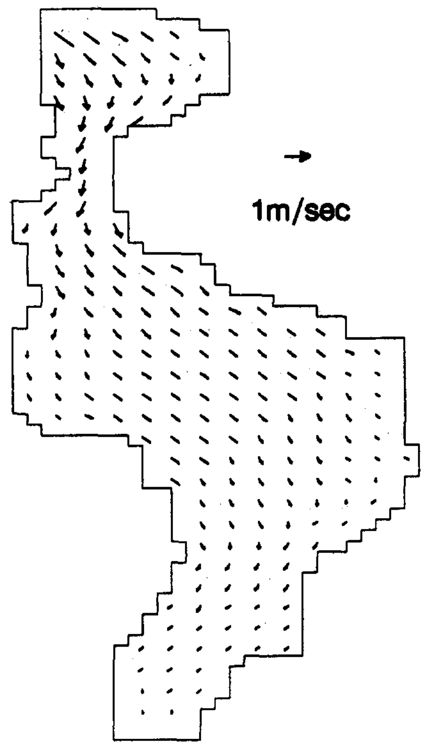

FIG. 3. Velocity Distribution at t = 102 h

structure and reliable performance of the particle model make it ideal for meeting this demand. By reproducing a documented flood, it has been demonstrated that the model provides a useful tool for accurately calculating flow over large flooded areas.

Additional research is needed on model convergence and error analysis. In addition, the possibility of improving the model's efficiency will be investigated in future work.

ACKNOWLEDGMENTS

The work was supported by the National Natural Science Foundation of China and the Fok Yingtung Funds.

APPENDIX I. REFERENCES

Cheng, R. T. (1983). "Eulerian-Lagrangian computations in estuarine hydrodynamics." *Proc., 3rd Int. Conf. on Numerical Methods in Laminar and Turbulent Flow*, Pineridge Press, Swansea, U.K., 341–352.

Harlow, F. H. (1964). "The particle-in-cell computer method for fluid dynamics." *J. Comp. Phys.*, 3, 319–343.

Harlow, F. H., and Welch, J. E. (1965). "Numerical calculation of time-dependent viscous incompressible flow of fluid with free surface." *Phys. Fluids*, 8(12), 322–329.

Hirt, C. W., Amsden, A. A., and Cook, J. L. (1974). "An arbitrary Lagrangian-Eulerian computing method for all flow speeds." *J. Comp. Phys.*, 14, 227–232.

Li, W., and Huang, J. G. (1993). "Numerical simulations of the flood development in the Jingjiang River flood-diversion area." *Res. Rep.*, Wuhan Hydraulic and Electric Univ., Wuhan, China, 1–19 (in Chinese).

Tetzlaff, D. M., and Harbaugh, J. W. (1989). *Simulating clastic sedimentation.* Van Nostrand Reinhold, New York, N.Y., 1–64.

APPENDIX II. NOTATION

The following symbols are used in this paper:

A = area of square region near grid point;
g = acceleration due to gravity;
h = flow depth;
h_{ij} = flow depth at a grid point;
$h_{p,i}$ = flow depth at the location of a particle;

N_{ij} = number of particles near a grid point;
n = Manning roughness of the bed;
$n_{p,t}$ = bed roughness at the location of a particle;
$\mathbf{S}_f$ = hydraulic friction slope;
$\mathbf{S}_{p,t}$ = water surface slope at the location of a particle;
$\mathbf{v}$ = depth-averaged velocity vector;
$\mathbf{v}_{ij}$ = flow velocity at a grid point;
$\mathbf{v}_{p,t}$ = velocity of a particle;
$\mathbf{X}_{p,t}$ = spatial position of a particle;
Z = topographic elevation;
ΔS = horizontal grid spacing in both the x and y directions;
Δt = time step; and
ΔV = volume of one particle.

Subscripts

ij = grid point;
p = fluid particle; and
t = time.