VNU Journal of Science, Earth Sciences 25 (2010) 168-176

Numerical simulations of overland floods in urban areas using a conservative Godunov-type scheme

Nguyen Tat Thang*

Institute of Mechanics, Vietnam Academy of Science and Technology, VAST

Received 14 September 2009; received in revised form 24 September 2009

Abstract. Floods in urban areas due to levee overtopping and/or breaking may cause a lot of severe damage of property and lost of human lives. In case of river dike and/or dam break, the problem is characterized by the overland propagation of discontinuity fronts or hydraulic jumps. It is of immense importance that urban planners and personnel have tools to assist in predicting and evaluating beforehand the flood process in such incidents. Recently, with the rapid development of computer resources and numerical methods, numerical models based on mathematical models for simulation of flood scenarios become highly useful. A model for the simulation of two dimensional (2D) overland floods in urban areas has therefore been developed. A finite volume Godunov-type numerical scheme is applied in the model. This numerical scheme has some important advantages. It is a conservative scheme and able to model more accurately hydraulic shockwave propagation. The scheme is based on unstructured computational meshes, in general, to deal with complicated urban geometries. The model has been applied to studying two experiments of overland floods. These experiments were carried out in research institutions in Japan and Italy. The computed results show general agreement with the measured ones. The model is prospective for analyzing overland flood process in practical cases.

Keywords: Numerical simulation; overland flood; godunov-type scheme; Un-structured meshes.

1. Introduction

Mathematical models for the numerical solution of the 2-D Saint Venant equations have long been developed. Applications of such models, which are based on advanced numerical techniques, to the simulations of overland floods in urban areas have attracted much attention recently [1, 2, 3, 4, 5]. These models are highly useful to urban planners to evaluate the impact of urban development to postulated flood events. Therefore numerical models for simulations of overland floods are urgently needed. The development of numerical methods for the solution of the 2D shallow water equations originally started with the traditional finite difference methods, then with the finite element methods and now with the finite volume ones [6]. Thanks to the rapid progress of the computer technology, computing ability increases incredibly. It enhances greatly the development of new, complicated 2D flood simulation models. Such advanced models usually based on the flexible irregular 2D computational meshes (unstructured mesh). In addition, the Godunov

* Tel.: (+44) 01224 273519
Email: thang.tat.nguyen@abdn.ac.uk

method, which is originated in aerodynamics and very efficient in dealing with problems with discontinuities, has recently been applied to fluid dynamics [7]. Moreover unstructured mesh generation techniques and models have recently been much developed and more powerful. Taking these advantages, we have developed a computer model to study 2D overland floods in urban areas. Such overland floods are typically 2D, and usually occur in very complicated geometries. The model uses unstructured meshes so that it can accurately deal with geometrically complex 2D domains. The unstructured meshes used consist of a set of connected-convex polygons with an arbitrary number of sides. In fact, due to the limited ability of mesh generation packages, the typical meshes used usually are triangular meshes. Our model is based on the Godunov method. This method is conservative and able to simulate unsteady flows with the presence of hydraulic discontinuities. One of the important difficulties arising in the implementation of the discretization scheme is the treatment of the wet-dry fronts [8]. Such fronts are inner boundaries, i.e. boundaries inside computational domains. They vary during the flood process. This situation is a very common in overland floods. A special technique has been applied based on the one mentioned in published literature [8]. The model is written in Compaq FORTRAN 6.6 programming language. Two experiments of the overland floods in urban areas [9, 10] have been studied numerically using the model. The computed results are compared with the measured ones. Acceptable agreements are obtained. The study shows that the model is able to deal well with wet-dry moving boundaries.

This paper briefly presents the numerical model in Part II. Computed results and comparisons for the experiments in Japan are given in Part III. Those of the experiment in Italy are presented in Part III. Conclusions are mentioned in Part IV. Finally a list of references is provided at the end of this paper.

2. Numerical model for the solution of the 2D shallow water equations

2.1. The system of equations

The model is based on the 2D system of the unsteady Saint Venant equations written in conservative form as shown below [4]:

$$\frac{\partial U}{\partial t}+\frac{\partial F(U)}{\partial x}+\frac{\partial G(U)}{\partial y}=S(x,y,U) \quad (1)$$

where $U=\begin{pmatrix} h \\ q_x \\ q_y \end{pmatrix}$ (conservative variable),

$$F=\begin{pmatrix} q_x \\ \frac{q_x^2}{h}+\frac{gh^2}{2} \\ \frac{q_x q_y}{h} \end{pmatrix},\quad G=\begin{pmatrix} q_y \\ \frac{q_x q_y}{h} \\ \frac{q_y^2}{h}+\frac{gh^2}{2} \end{pmatrix},\quad q_x=uh,$$

$q_y = vh$; h is the water depth; g is the gravity acceleration; (u,v) are the x and y components of the depth averaged velocity respectively; S is the source term. Equation (1) can be rewritten in the following form:

$$\frac{\partial U}{\partial t}+\nabla.E(U)=S(x,y,U) \quad (2)$$

where $E=\begin{pmatrix} F \\ G \end{pmatrix}$.

The unknowns that need to be computed are h, q_x and q_y or h, uh and vh.

2.2. Numerical technique

For a fixed control volume Ω as shown in Fig.1, the integral form of (2) is written as:

$$\int_\Omega \frac{\partial U}{\partial t} d\Omega + \int_\Omega \nabla.E(U) d\Omega = \int_\Omega S(x,y,U) d\Omega \quad (3)$$

Applying the Gauss's theorem, (3) can be rewritten in the following form

$$\frac{\partial}{\partial t}\int_\Omega U d\Omega + \oint_{\partial\Omega} (E.n) ds = \int_\Omega S d\Omega \quad (4)$$

where $\partial\Omega$ denotes the boundary surface of the 2D volume Ω, and n is the unit outward normal vector (Fig.1).

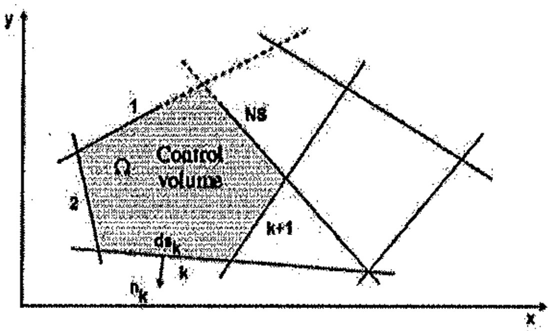

Fig. 1. A control volume (element or cell) in two dimensions (NS: number of sides; ds_k: the length of the side k).

Since equation (4) is written for each individual control volume (an element or cell of the computational meshes), the discretization technique is applied to each element. Denoting by U_i the average (or discrete) value of conservative variable over the volume Ω_i, using equation (4), the following conservation equation can be written for each cell i:

$$\frac{\partial U_i}{\partial t} A_i + \oint_{\partial\Omega_i} (E.n) ds = \int_{\Omega_i} S d\Omega \quad (5)$$

where A_i is the area of the 2D volume Ω_i [4].

Applying the mid-point rule to approximating the contour integral in (5) and a simple approximation for the time derivative, a finite difference like form of (5) is written as:

$$U_i^{n+1} = U_i^n - \frac{\Delta t}{A_i}\left(\sum_{k=1}^{NE} E_k^* . n_k . ds_k\right)_i^n + \Delta t S_i^{*n} \quad (6)$$

The ideas of the Godunov method and the Roe's approximate Riemann solver [11], which are originated in aerodynamics, are applied to the approximation of the E_k^* flux [7].

All details of the system of equations and discretization scheme should be referred to [4].

As for boundary conditions, the model uses three types of boundary conditions. Each of those is used where relevant. The first one is the condition of the river water discharges from river outlets flowing into the simulation domain. The second one is the reflective and no-slip boundary condition applied to rigid boundaries. And the last one is the free flow condition at open sea boundaries [4].

The numerical scheme shown here, for unstructured meshes in general, is highly efficient for the solution of the propagation of waves in spatial domains of complicated geometry [7]. Therefore it will be applied in this study.

3. Numerical study of the overland flood experiments

3.1. Experimental model of a dike break induced overland flood (Japan)

Experimental model description:

The experiment of a dike break induced overland flood in a city area was performed in DPRI (Disaster Prevention Research Institute), Kyoto University in Japan. The experiment aimed to simulate overland floods, which is caused by a water flow overtopping the river bank into the city (Fig.3), in a real site chosen as shown in Fig.2. This is a highly urbanized area of the ancient city of Kyoto, Japan. The site covers a square area of 1km x 2km. The experimental model site is reduced to a smaller scale of 10m x 20m [9]. Fig.3 shows positions numbered from 1 to 8 where the water depth was measured during the experiment. The Manning roughness coefficient determined in the experiment is calculated to be 0.01. The whole experimental site is dry just before the experiment begins.

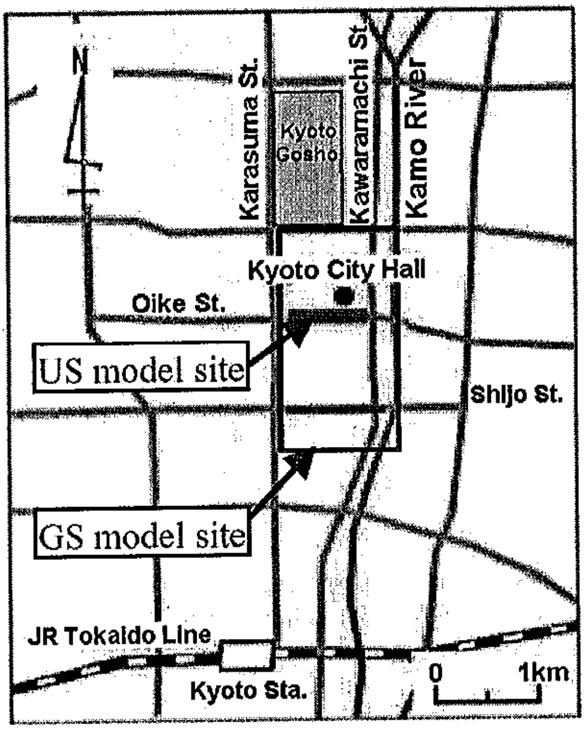

Fig. 2. The real experimental site.

The average slope of the site (downward to the South direction) is about 0.005. The experimental model assumes that there is no water invading into residential and building areas so that flood water only flows in the complicated street network in the modeled site (Fig.3). Fig.4 shows the experimental model set up in the Hydraulic Laboratory of DPRI. The discharge of the water flowing through the dike break point is computer controlled and shown in Fig.5.

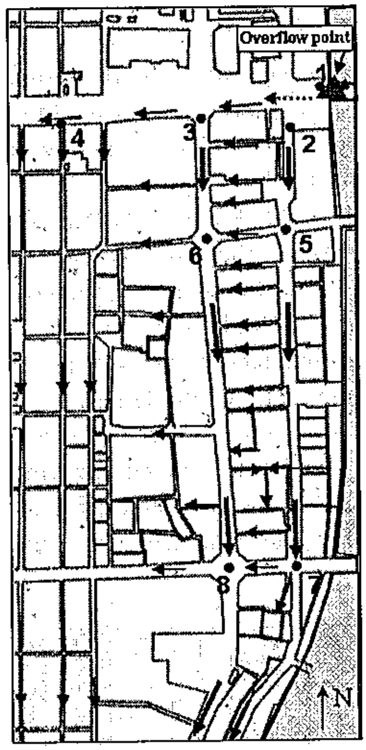

Fig. 3. The water depth measurement points.

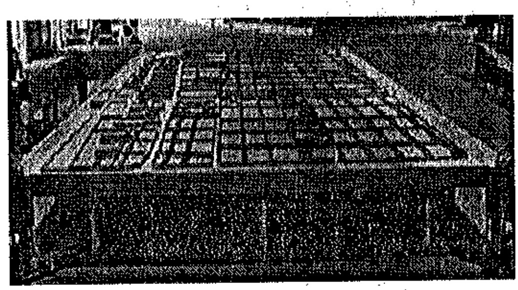

Fig. 4. The experimental model.

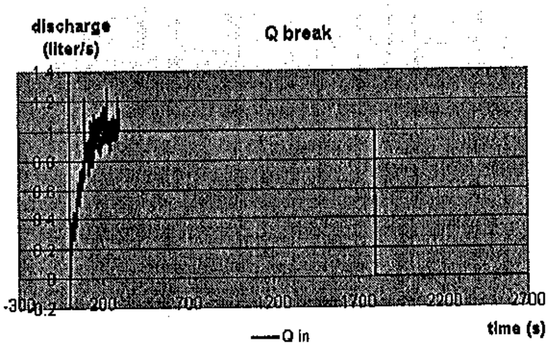

Fig. 5. The inflow discharge.

Numerical model:

The data structure of the computational meshes and geometry needed for the numerical model developed here is completely the same as the one described and used in the model mentioned in [5]. Some important features are abstracted here: the number of unstructured meshes is 4996; the meshes of streets are very fine but those of building blocks are kept coarse to save the time needed for mesh generation and for numerical simulation. This is straightforward since water does not penetrate into these blocks during the experiments. It is noted here that the computational meshes can be very flexible and irregular (unstructured meshes).

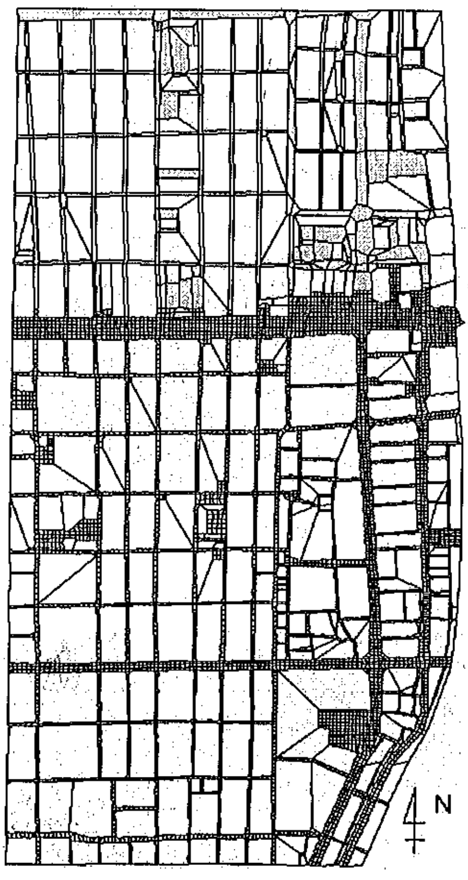

Fig. 6. The computational meshes.

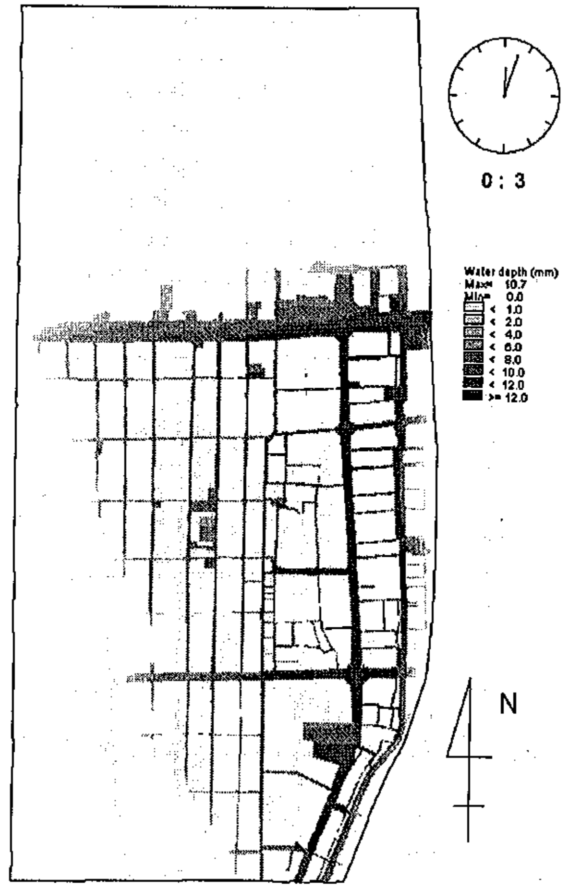

Fig. 7. The computed result of the water depth distribution after 5 minutes.

Fig.7 shows the distribution of the water depth computed in the area and the development of the overland flood in the area after 5 minutes.

Comparisons between the computed results and the measured ones:

Water depths are measured at the points (No.1 to No.8) mentioned in Fig.3. The data is provided by the Hydraulic Research Group in DPRI. These results are compared with the ones computed by the numerical model. The comparisons of the water depths are shown in from Fig.8 to Fig.11 below.

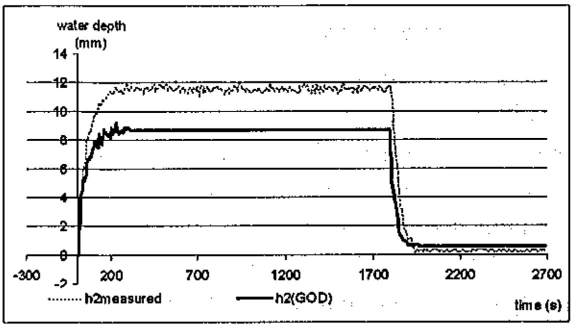

Fig. 8. Comparison of the water depth at point No.2.

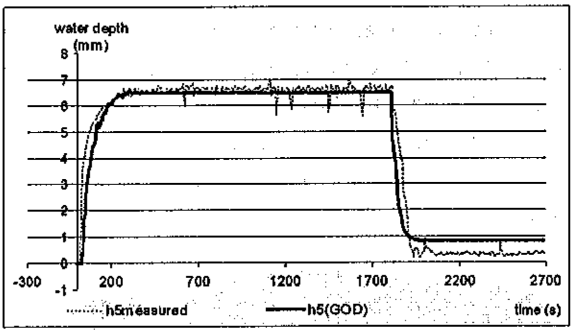

Fig. 9. Comparison of the water depth at point No.5.

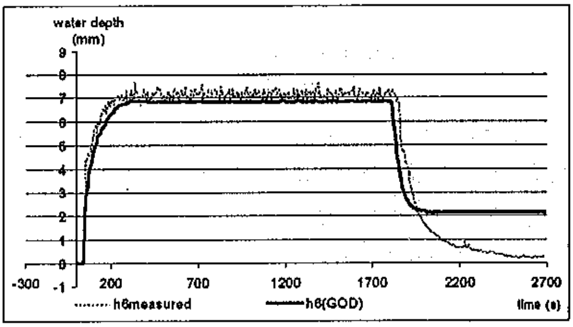

Fig. 10. Comparison of the water depth at point No.6.

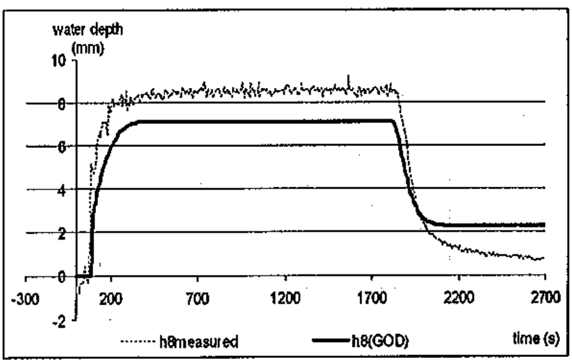

Fig. 11. Comparison of the water depth at point No.8.

Some remarks:

- The model developed in this study has been successfully applied to the simulation of the overland flood process in the experiment.
- The computed results show acceptable agreement with the measured ones. Some differences are assumed to be due to the nature of too shallow depth of the advancing fonts of water (wet-dry moving boundaries) in the experiment. The depth of those fronts is of the order of less than 1mm. Therefore the surface roughness would not be the same everywhere (a constant value of the roughness coefficient is used in the numerical simulation). This problem would need a theoretical treatment in the numerical model, or need to use different values of the Manning roughness coefficient at the advancing front. Proper treatment of the problem is the subject of further study.
- The development of the flood in the area during the experiment is also compared with the observed one. The extension of the flooded area in the numerical simulation agrees well with that in the experiment.
- The numerical model deals well with very irregular geometry and wet-dry moving/varying boundaries.

3.2. Experiment of a flood into a city area in the framework of the CADAM (European Concerted Action on Dam-Break Modeling) project (experiment performed in Italy)

Description of the experimental model:

The experimental model set up reproduces a 5km reach of the Toce River in Italy (Fig.12). There are floodplains, reservoir, structures, and buildings etc. in this area. The scale between the experimental model and the real site is 1:100. The scale of the experimental model is 55mx13m [10]. Fig.12 shows the overview of the model geometry and topography. The experiment simulated a flood caused by a reservoir dam break in the upstream area of the modeled site (left hand side in Fig.12). The flood water flows into the modeled site through the AD boundary (Fig.15).

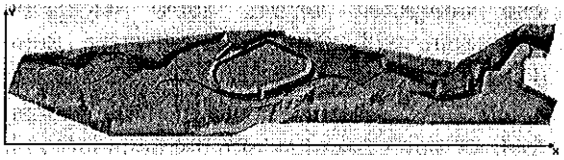

Fig. 12. An image of the experimental model taken from a DTM (Digital Terrain Model) (Figure from [10]).

In Fig.13, the gauge positions for measuring water depth in the experiment are shown. The Manning coefficient in the experiment is determined to be 0.0162. The experiment starts with the dry bed condition in the whole area. The discharge of the flood water flowing into the area is also computer controlled as the previous experiment in Japan. The discharge curve is presented in Fig.14.

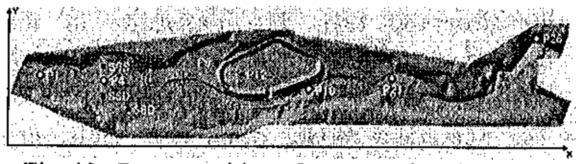

Fig. 13. Gauge positions for measuring the water depth (Figure from [10]).

Fig.14 shows the discharge of the flood water flowing into the experimental model site during the experiment. A total amount of about 18.4 m3 of water flows into the area during the experiment.

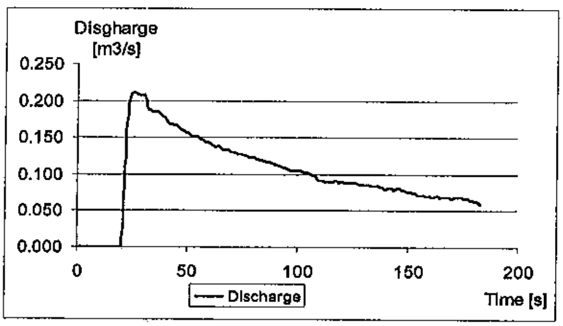

Fig. 14. The discharge of the flood water invading into the experimental model site.

Numerical model:

The experimental area is divided into 14651 quadrilateral elements (computational meshes) and 15000 nodes (the total number of all vertexes of the quadrilateral elements). The element size is 14cmx14cm. In this case, the topography is not too complicated so that, for convenience, we used quadrilateral elements. A structured-curvilinear mesh generator package CCHE Mesh Generator [12] is used to generate the computational meshes. The meshes can be generated as fine as we want. It can be seen in Fig.15 that the meshes generated are really fine so that they can reconstruct well the complicated topography of the experimental area.

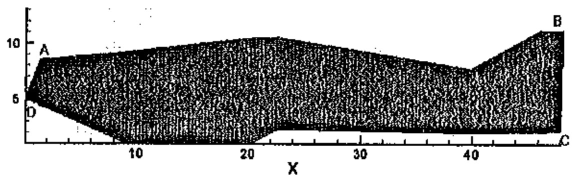

Fig. 15. The computational meshes generated using the CCHE mesh generator.

In Fig.15, AD is the inflow boundary; AB and CD are the rigid boundaries and BC is the free outflow boundary.

Computed results:

The computed results of the water depth are compared with the measured ones provided by CADAM project. The results of the comparisons are shown in from Fig.16 to Fig.19 below.

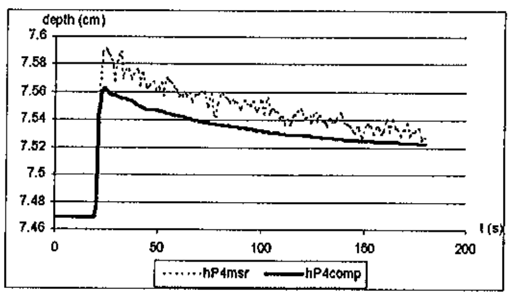

Fig. 16. Comparison of the water depth at point No.P4.

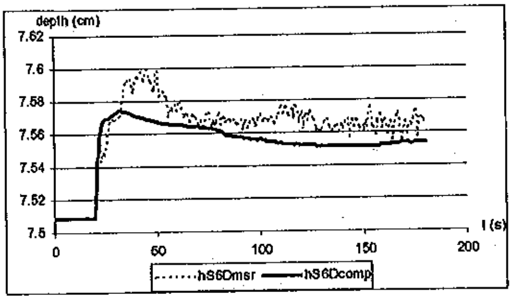

Fig. 17. Comparison of the water depth at point No.S6D.

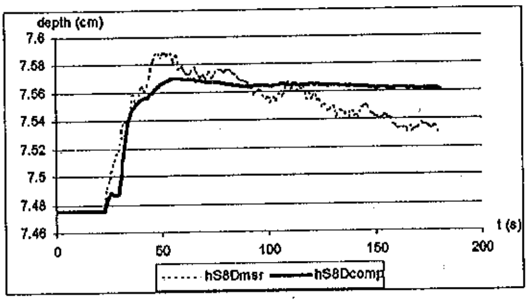

Fig. 18. Comparison of the water depth at point No.S8D.

Some remarks:

- The comparisons show that the computed water depths agree quite well with the measured ones. Moreover the arrival times of the advancing fronts (discontinuities) are modeled fairly exactly. This shows the advantageous feature of the Godunov-type scheme.
- The development of the flood over wet-dry bed with complicated topography has been reproduced.
- Using the model, overland floods caused by dam/dike break or overtopping into areas with different types of structures can be modeled properly.

4. Conclusions

A computer model for the simulation of overland floods in city/urban areas with complicated topography/geometry has been developed. A new discretization technique has been applied in the model. The model exploits advantageous features of a Godunov-type numerical scheme and the Roe's approximate Riemann solver which is originated in aerodynamics. This scheme deals well with hydraulic discontinuities in overland flood flows which are caused by dike or dam breaks. The model uses flexible computational meshes which are unstructured meshes. Therefore the model can be applied to problems with irregular geometries. The model has been applied to simulations of two experiments of overland floods in city areas in Japan and Italy. The computed results agree well with the measured ones. The treatment of wet-dry and moving boundaries implemented in the model does work properly. The model is highly prospective for studying overland floods in practical cases in real city areas.

Acknowledgements

The author is grateful to the Hydraulic Research Group in DPRI for providing their experimental results. The author also thanks Prof. Nguyen Van Diep at the Institute of Mechanics, VAST, who has been actively leading the research on dam/dike break and overland flood problems in Institute of Mechanics, for providing the experimental results from CADAM project.

References

[1] S. Haider, A. Paquier, R. Morel, J.-Y. Champagne, Urban flood modeling using computational fluid dynamics, *Proceedings of the Institution of Civil Engineers, Water & Maritime Engineering* 156 Issue WM2 (2003) 129.

[2] N.T. Thang, K. Inoue, K. Toda, K. Kawaike, Flood inundation analysis based on unstructured meshes for the Hanoi central area, *Annual Journal of Hydraulic Engineering, JSCE* 48 (2004), 601.

[3] Nguyen Tat Thang, Nguyen The Hung, Application of a Godunov type numerical scheme and a domain decomposition technique to the parallel computation of tidal propagation, *VNU Journal of Science, Earth Sciences* 25 No. 2 (2009) 104.

[4] Nguyen Tat Thang, Nguyen Van Hanh, Nguyen The Duc, Some initial results of the application of a Godunov type scheme to the numerical solution of the 2D shallow water equations, *Proceedings of the Annual National Conference on Fluid Mechanics* (2004) 565 (in Vietnamese).

[5] N.T. Thang, K. Inoue, K. Toda, K. Kawaike, A model for flood inundation analysis in urban area: Verification and Application, *The Annuals of DPRI, Kyoto University, Japan* (2004) 303.

[6] C.A.J. Fletcher, *Computational techniques for fluid dynamics I & II*, Springer (1991).

[7] E.F. Toro, *Riemann Solvers and Numerical Methods for Fluids Dynamics: A Practical Introduction* (2nd Edition), Springer Verlag, Berlin (1999).

[8] P. Brufau, M.E. Vazquez-Cendon, P. Garcia-Navarro, A numerical model for the flooding and drying of irregular domains, *International Journal for Numerical Methods in Fluids* 39 (2002) 247.

[9] H. Nakagawa, T. Ishigaki, Y. Muto, K. Inoue, K. Toda, H. Tagawa, Y. Yoshida, K. Tatsumi, H. Zhang, H. Yagi, Inundation by river water in urban cities: experiments using a large scale inundation model and their analyses, *The Annuals of DPRI, Kyoto University, Japan* (2003).

[10] S. Soares Frazao, G. Testa, The Toce River test case: Numerical results analysis, *Proceedings of the 3rd CADAM Meeting, CADAM Project* (1999).

[11] P.L. Roe, Approximate Riemann solvers: parameter vectors and difference schemes, *Journal of Computational Physics* 43 (1981) 357.

[12] CCHE Mesh Generator at http://www.ncche.olemiss.edu/software/downloads, National Center for Computational Hydroscience and Engineering, The University of Mississippi.