Archives of Hydro-Engineering and Environmental Mechanics
Vol. 52 (2005), No. 4, pp. 321–333

Numerical Simulation of Extreme Flooding in a Built-up Area

Michał Szydłowski

Gdańsk University of Technology, Faculty of Civil and Environmental Engineering
ul. Narutowicza 11/12, 80-952 Gdańsk, Poland, e-mail: mszyd@pg.gda.pl

(Received May 20, 2005; revised July 18, 2005)

Abstract

Two numerical simulations of extreme (flash) flood propagation in an urban area are presented. The simulations are performed to recognize some specific features of flow in a built-up area. As the mathematical model of free surface unsteady water flow the shallow water equations are assumed. In order to solve the equations, a numerical scheme based on finite volume method is applied. For approximation of mass and momentum fluxes the Roe method is used. The calculations are examined against the experimental data. The measured variations of water depth at some control points of flooded area are available due to physical modelling. The experiments of model city flooding events were carried out at the hydraulic laboratory of ENEL-CESI in Milan (Italy) in a framework of EC IMPACT project. The aim of these experiments was to simulate a flood in the area where a building group representing a simplified city configuration was introduced.

Key words: flash flooding, urban area, mathematical modelling, shallow water equations, finite volume method, numerical simulation

1. Introduction

As the name implies, flash floods are sudden, extreme, rapid and often unpredictable flooding events which can occur with little or no warning and reach full peak in a short time. Extreme flooding in an urban area can be the result of natural events, like heavy rains for instance, or can be caused by failures of flood defence structures, such as dams, weirs, sluices or flood embankments. Although floods in built-up areas are relatively rare, they are much more devastating than in any other areas. Moreover, they can pose a significant threat to human life. Considering this, it seems that numerical simulations of flash flood events can play an essential role in urban hydrology. Mathematical modelling of flow in built-up areas seems to be the main tool for assessment and reduction of risks from extreme flooding. Simulations of flood events are necessary to better protect cities and increase public safety. They can be used to develop emergency plans, control development within potential flood zones or even to determine insurance premiums for some properties. Information about potential flood must contain such data as:

- time of arrival of the flood wave at some characteristic points of the city,
- extreme water levels in flooded area,
- duration and range of the flooding,
- water depth and velocities in the flood zone.

The flash flow in an urban area is usually rapidly varied and often supercritical. In addition, flow in a city has some specific features like interaction with buildings and other structures and significant variation of flow profile. Moreover, flow discontinuities like hydraulic jumps can occur due to wave reflections against walls or abrupt changes of city topography. Considering all these features it is clear that transcritical flow with discontinuities must be modelled to simulate the extreme flooding in a built-up area. Regarding complexity of city structure, simulations of flow in two dimensions (at least) in a horizontal plane, are indispensable.

Methods for modelling flash floods in urban areas are rather limited. However, satisfactory results of modelling of dam-break flows in natural valleys based on shallow water equations (SWE) (Morris 2000, Szydłowski 2003) make this mathematical model a promising candidate to describe the urban flow also. In order to asses if the SWE model is an adequate representation of urban flood wave propagation process or not, it must be validated. Extreme flooding events are usually of a catastrophic nature and are insufficiently documented, thus the results of numerical simulations are difficult to verify using field data. However, computations can be examined against results of laboratory experiments. The simulations presented here concern flash flow through a regular built-up model city. This problem was investigated at the hydraulic laboratory of ENEL–CESI in Milan (Italy) and used as a test case in the framework of EC IMPACT project (No. EVG1-CT2001-00037).

2. Governing Equations and Solution Method

Shallow water equations – known as 2D dynamic wave model – can be obtained from the Navier Stokes equations using a depth averaging procedure (Szymkiewicz 2000). Therefore, SWE are not a true mathematical representation of the free surface water flow. For equations derivation it is assumed that vertical velocities can be neglected, which means that vertical accelerations are equal to zero, pressure field is hydrostatic, bottom slope is small and bottom friction can be approximated as in steady flow conditions. Considering the urban flow characteristic features it is obvious that this kind of flow implies the movement of water in a vertical direction, thus, in general, SWE assumptions are not satisfied for this 3D rapidly varied flow and the model seems to be a poor representation of a real phenomena. However, most mathematical models of flood propagation are based on SWE.

The system of SWE in conservative form can be written as (Abbott 1979)

$$\frac{\partial \mathbf{U}}{\partial t} + \frac{\partial \mathbf{E}}{\partial x} + \frac{\partial \mathbf{G}}{\partial y} + \mathbf{S} = 0, \tag{1}$$

where

$$\mathbf{U} = \begin{pmatrix} h \\ uh \\ vh \end{pmatrix}, \tag{2a}$$

$$\mathbf{S} = \begin{pmatrix} 0 \\ -g\,h\left(S_{0x} - S_{fx}\right) \\ -g\,h\left(S_{0y} - S_{fy}\right) \end{pmatrix}, \tag{2b}$$

$$\mathbf{E} = \begin{pmatrix} uh \\ u^2h + 0.5g\,h^2 \\ uvh \end{pmatrix}, \tag{2c}$$

$$\mathbf{G} = \begin{pmatrix} vh \\ uvh \\ v^2h + 0.5g\,h^2 \end{pmatrix}. \tag{2d}$$

In this system of equations h represents water depth, u and v are the depth-averaged components of velocity in x and y directions, respectively, S_{0x} and S_{0y} denote the bed slope terms, S_{fx} and S_{fy} are the bottom friction terms defined by the Manning formula and g is the acceleration due to gravity. Equation (1) can be written in another form

$$\frac{\partial \mathbf{U}}{\partial t} + \operatorname{div}\,\mathbf{F} + \mathbf{S} = 0, \tag{3}$$

where, assuming unit vector $\mathbf{n}=(n_x, n_y)^T$, vector $\mathbf{F}$ is defined as $\mathbf{Fn}=\mathbf{E}n_x+\mathbf{G}n_y$. In order to integrate the SWE in space using the finite volume method, the calculation domain is discretized into a set of triangular cells (Fig. 1).

After integration and substitution of integrals by corresponding sums, equation (3) can be rewritten as

$$\frac{\partial \mathbf{U}_i}{\partial t}\Delta A_i + \sum_{r=1}^{3}\left(\mathbf{F}_r\mathbf{n}_r\right)\,\Delta L_r + \sum_{r=1}^{3}\mathbf{S}_r\,\Delta A_r = 0, \tag{4}$$

where $\mathbf{F}_r$ is the numerical (computed at r^{th} cell-interface) flux and ΔL_r represents the cell-interface length. $\mathbf{S}_r$ and ΔA_r are the components of source terms and area of cell i assigned to r^{th} cell-interface.

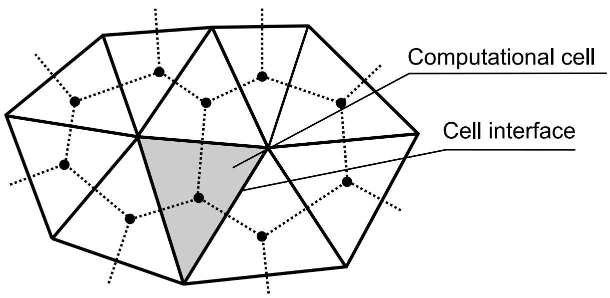


Fig. 1. FVM discretization of calculation domain

In order to calculate the fluxes $\mathbf{F}_r$, the Roe (1981), scheme is used. Detailed description of the method is available in the literature (Glaister 1993, Toro 1997, Szydłowski 2001a, 2003) therefore it is omitted here. The source term vector **S** contains two sorts of elements, which depend on bottom and friction slopes, respectively. Both of them pose numerical integration difficulties. In order to ensure the proper bottom slope approximation this term is up-winded in the same way as fluxes **F** (Bermudez and Vazquez 1994). The second source term, friction, is inconvenient when flow of great velocity and small depth is present. In order to avoid the numerical integration problem the splitting technique with respect to physical processes is applied. The numerical algorithm is completed using a two-step explicit scheme of finite difference method for SWE integration in time. This scheme is of second-order accuracy in time and its stability is restricted by Courant number. For 2D shallow water equations the stability condition can be written as (Potter 1977)

$$Cr = \frac{\max\left(\sqrt{u_i^2 + v_i^2} + \sqrt{g\,h_i}\right)}{\min\left(d_r\right)/\Delta\,t} \le \frac{1}{\sqrt{2}}, \tag{5}$$

where subscript i denotes given cell and d_r represents the distances between centre points of the cell i and its neighbouring volumes.

3. Experimental Investigation of City Flooding

In order to carry out the experiments on city flooding, the physical model of Toce valley was adapted. Originally, the model – built at the hydraulic laboratory of ENEL-CESI in Milan, Italy – was used to investigate dam-break flows in natural valleys (Testa 1999). A 5 km reach of the Toce river was reproduced as a 1 : 100

scale model. The model – 50 m long and 11 m wide – was built in concrete and reproduced the riverbed, floodplain and some details of the valley geometry like bridges and buildings. The geometry of the model was defined as a digital terrain model. The Manning friction coefficient was estimated for steady flow conditions as $n = 0.0162\ \text{m}^{-1/3}\text{s}$. The water depth laboratory measurements were used as data sets for validation of some mathematical models in a framework of EC CADAM project (Morris 2000). The results of physical modelling of dam-break flows in Toce river valley were also presented and discussed by Szydłowski (2001b, 2003).

In order to investigate the extreme flooding in a built-up area, the model of the city has been located at the upstream part of the original valley model about 4 m downstream the inflow section. In general, only a 6 m long area (containing building models) close to the upstream end of the Toce model was considered during physical and mathematical modelling. In order to simplify the flow structure the 'urban area' was separated from the valley margins by two masonry walls placed parallel to the model main axis. Other features of the physical model have remained the same as originally.

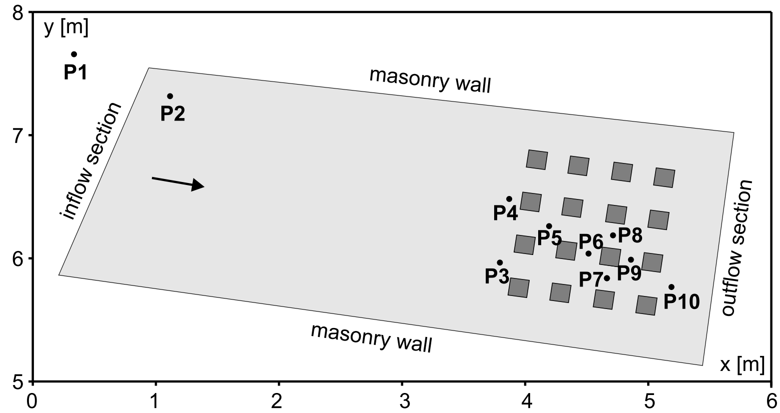
The models of houses were built as concrete cubes with 0.15 m side. Two configurations of building layout were investigated during physical modelling – aligned and staggered. In the former (case a) buildings are placed in rows approximately parallel to the main axis of the valley forming a grid configuration (Fig. 2a). In the latter (case b), cubes representing houses are arranged in a checker board configuration (Fig. 2b). In order to measure the depth variation during the experiment the control points (P3–P10) have been placed upstream, among and downstream of the buildings (Fig. 2a, b).

Initially there is no water in the whole model. The flash flooding of the simplified urban area is achieved using a pump system by suddenly rising water level in a feed tank connected to the upstream part of the model. The extreme flood hydrograph, recorded as inflow discharge – common for two test cases presented in this paper – is shown in Figure 3. In order to define the boundary condition at the inflow section, the measurements of depth variation at points P1 and P2 were also used. During the experiments the flow through the inlet section is subcritical. At the downstream end of the model city the outflow is sub- or supercritical, depending on actual flow conditions.

4. Numerical Simulations and Discussion of the Results

In order to simulate the city flooding experiments for aligned and staggered building configuration, the calculation domain is covered by unstructured triangular mesh composed of 5559 and 5185 computational cells, respectively in case (a) and (b), Fig. 4a, b.

The mesh used in each simulation is locally refined in an urban area between buildings to better represent the complex structure of the city. Therefore, the a)



b)

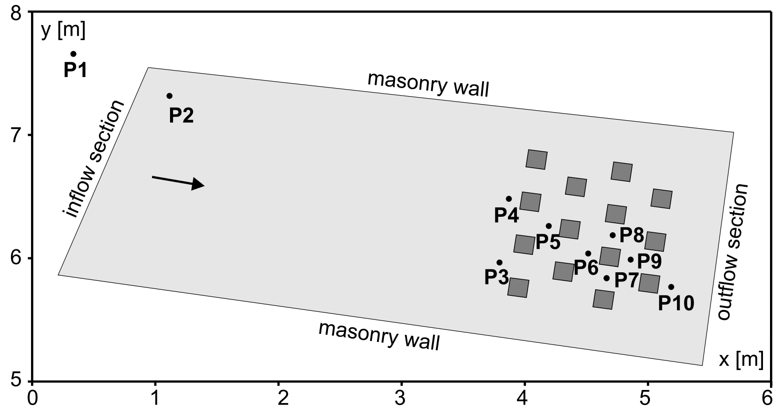


Fig. 2. Geometry of flow area for a) aligned and b) staggered building configuration

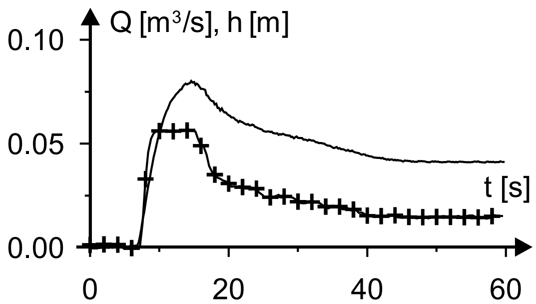


Fig. 3. Flooding hydrograph recorded at inflow section of the model: discharge (+) and depth (–)

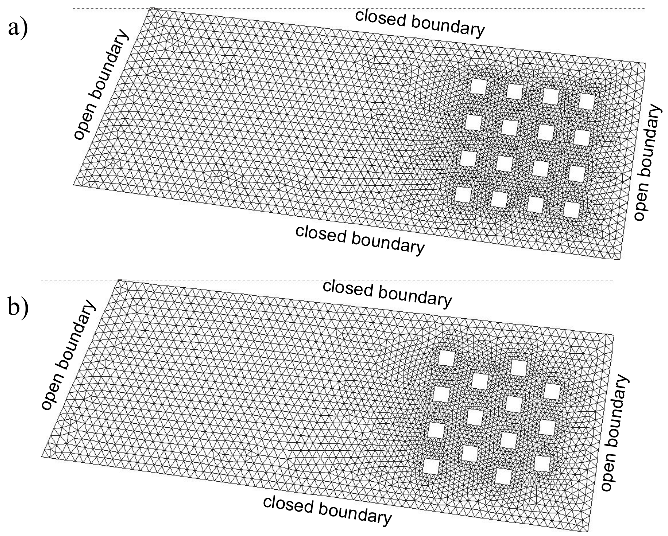


Fig. 4. Numerical mesh for a) aligned and b) staggered building configuration

length of sides of computational cells (elements) around each building is equal to 0.04 m, increasing to 0.08 m near the boundaries of flow area.

Initially the whole domain is covered by a thin water film (0.0001 m) simulating the dry bottom of the flow area. The boundary conditions are imposed in accordance with the experiment. At the inflow section the velocities in the normal direction to the boundary are imposed. At the outflow boundary the free outflow condition is imposed. The same boundary conditions are assumed for both simulations. Calculations are carried out with time step $\Delta t = 0.01$ s, ensuring the calculation stability. The simulation time is equal to 60 s.

The flash flow in an urban area can be analysed using results of numerical simulations presented in Figure 5. As can be seen, the front of the flood wave reaches and crosses the first row of buildings after about 11 seconds simulation (Fig. 5a). Until this moment the flow is the same both in case (a) of aligned buildings and (b) – for staggered configuration. After reaching the urban area, the water level swelling and decrease of velocity can be observed upstream the first row of buildings. There the flash wave reflects against the first row of buildings (area of sudden reduction of flow section) resulting in the formation of a hydraulic jump moving upstream which is similar in cases (a) and (b).

The significant difference in structure of flow for two analysed building configurations can be observed in the centre of the city. If the buildings are arranged in a grid (case a), forming a system of perpendicular channels (streets) the flow suddenly accelerates along longitudinal streets (Fig. 5b). At the downstream side of each building a re-circulation zone with a great gradient of water surface can

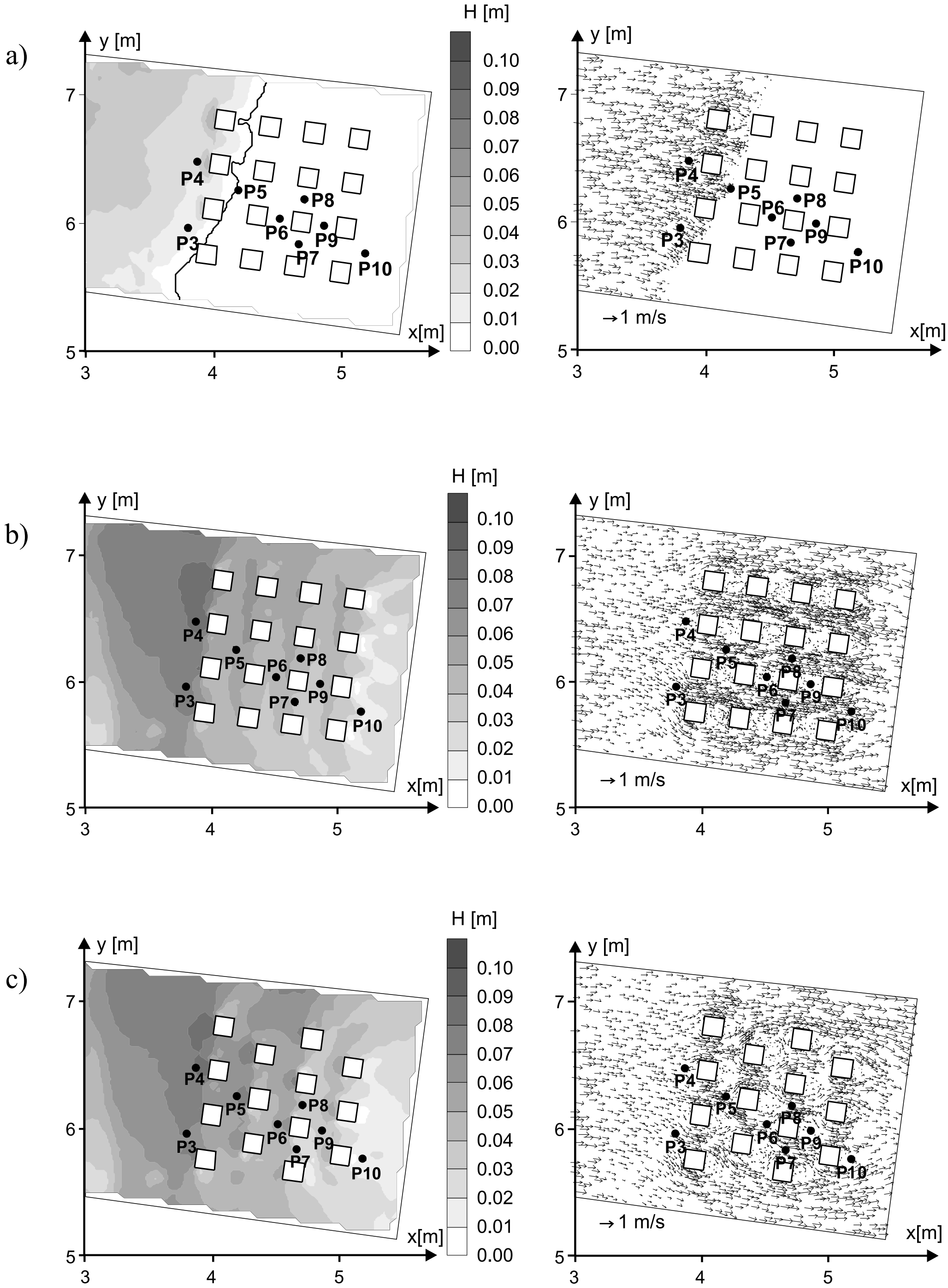


Fig. 5. Computed contours of water depth and velocity fields for aligned building configuration after a) 11 s and b) 15 s and for staggered building configuration after c) 15 s from the beginning of the simulation

be observed. Additionally, upstream of each row of buildings (located perpendicular to the main flow direction) the linear water surface swelling together with large velocity gradients can be seen (Fig. 5b). In case (b), when the buildings are arranged in a checker board configuration, the flow structure between buildings is more complex (Fig. 5c). Analysing the shape of water surface and velocity field it can be observed that flow is split upstream each building forming a swelling in front of an obstacle and a wake zone downstream. The numerous and separate swellings and depressions result in significant gradients of water level and velocity, in both the main flow direction as in cross sections.

Comparisons between computed and measured depth variations at some control points are shown in Figures 6 and 7. For both simulations (cases a and b), the resulting agreement at each point is quite good, except for point P5 in case (a) of aligned buildings (Fig. 6). The calculated water depth is greatly overestimated at this location. It seems that the reason for this disagreement can be measurement error – which is impossible to verify at this moment – or maybe it is the result of the specific location of point P5 near which the water swelling is observed. This means that even a small error in simulated swelling location can result in significant disagreement between computed and measured water depth.

It can be seen that in both simulations observed and calculated depth variation have a similar shape. Only at points P3 and P4 (Figs. 6 and 7) – placed upstream of the first row of buildings (Figs. 2a, b) – the first peak in the water level has not occurred in computed results. This rapid water level increase occurs just after the flood wave reflection against buildings and it probably represents a water splash effect which cannot be modelled using the SWE model. Some smaller water level oscillations can also be observed in measurements at other points. They occur in the flow due to reflections against walls and sudden reductions and expansions of flow section. Generally, they are the result of complex structure of built-up flow area. At the same time the graphs of computed water depth history are fairly smooth. This is observed because 3D hydraulic effects cannot be represented in 2D numerical simulations at all. Moreover, the numerical scheme used to solve SWE – able to handle flow discontinuities – is locally dispersive and adds some numerical diffusion to the solution ensuring the stability of computations and resulting in more regular shape of solution.

Analysing results of simulations it can be seen that in case (b) of staggered buildings measured depth is reproduced fairly accurately regardless of control point position (Fig. 7, Fig. 2b). Depth variation as well as wave arrival time are simulated properly here. Points P5, P6, P7, P8 and P9 are located between buildings. Here the numerical model gives a precise representation of water level evolution, irrespective of point location. The highest maximum water depth can be observed in the upstream part of a built-up area (P5, P6, P8, Fig. 7) and the lowest in the downstream part (P7, P9, P10). Point P10 (Fig. 2b) lies in the downstream part of the model city in a wake zone. At this point a slight difference between

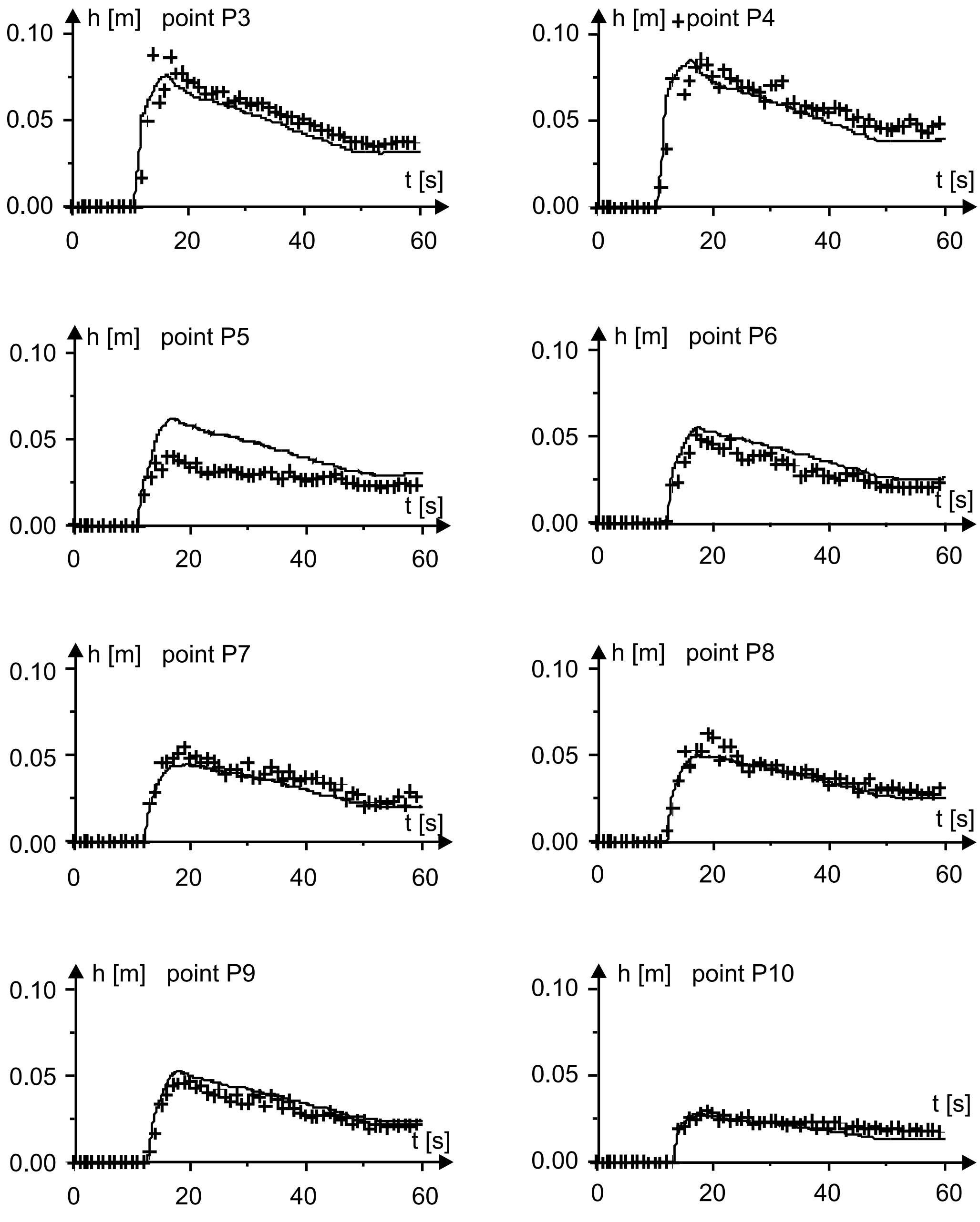


Fig. 6. Calculated (–) and measured (+) depth variation for aligned building configuration

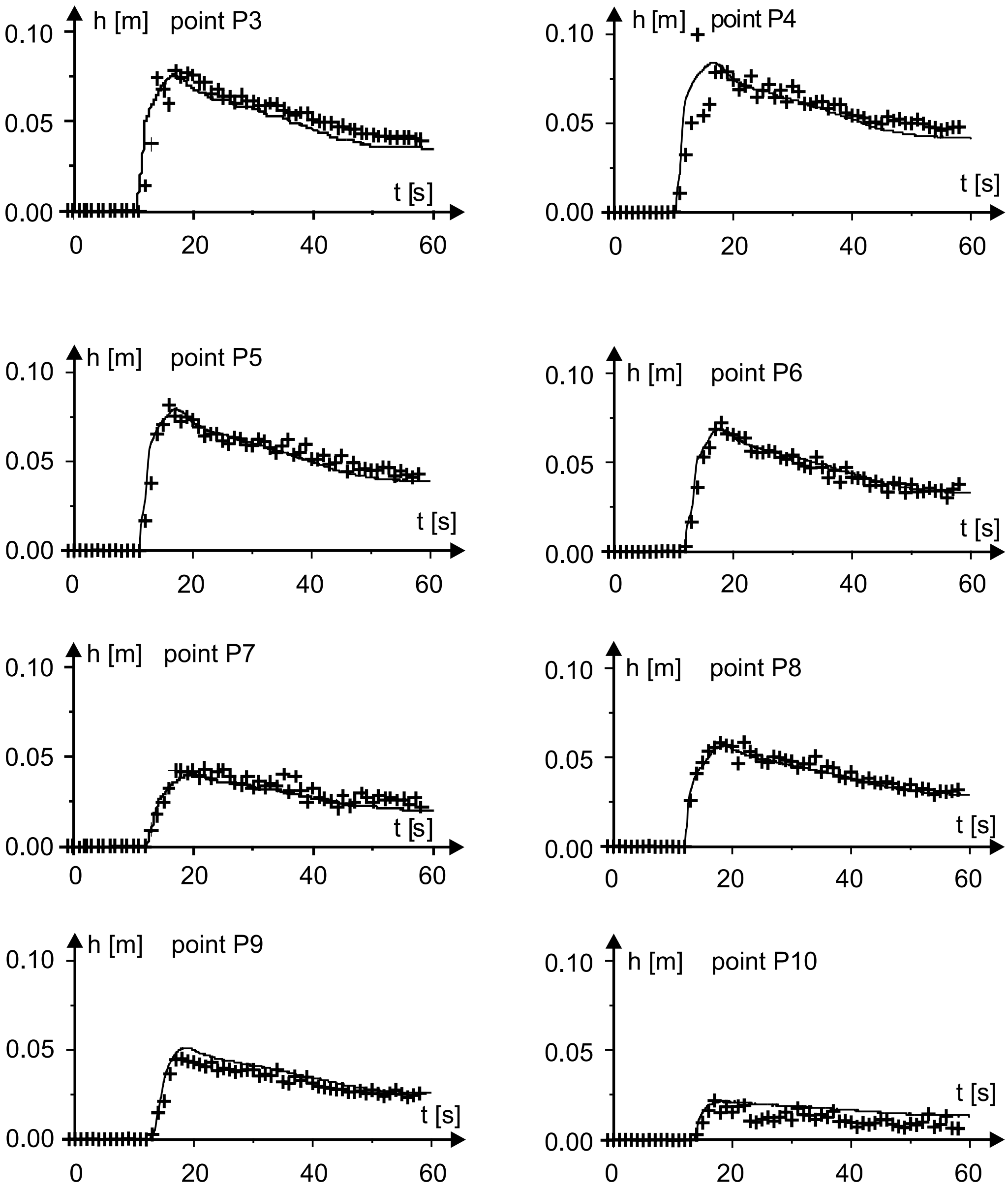


Fig. 7. Calculated (–) and measured (+) depth variation for staggered building configuration

calculated and measured results can be observed (Fig. 7). This point is located between the city model outflow and downstream boundary, where the flow varies from sub- to super-critical depending on actual flow conditions. It makes the calculation uncertain in this region as the boundary condition is unknown there.

In the case (a) of aligned buildings (Fig. 2a) the highest maximum water depth in an urban area is observed at points P6, P7, P8 and P9 (Fig. 6). The lowest is at point P10 located downstream from the last buildings row. It seems that calculated maximum water depth is slightly underestimated at points located in main longitudinal streets (P7, P8). It may be due to some additional energy losses (i.e. local, building walls friction) not represented in mathematical model in which only the bottom friction is approximated using a simple Manning formula.

5. Conclusions and Remarks

The city flash flooding in two configurations of built-up area were simulated using SWE. The results of numerical simulations were used to recognize and investigate the characteristic features of rapidly varied flow in a simplified urban area. The computations were verified basing on experimental data from physical modelling. Quite good agreement between results was observed.

In general, it can be concluded that although SWE are not a true mathematical representation of the free surface water flow, numerical model based on these equations is able to simulate the rapidly varied flow in built-up areas. Of course, such a model cannot reproduce details of local flow or short wave effects (i.e. inner structure of hydraulic jump or water surface oscillations, respectively), but the main features of extreme flow, like reflections against buildings, transitions from supercritical to subcritical flow with hydraulic jumps, steep wave fronts with significant gradients of depth and velocity and circulation areas can be successfully simulated.

Additionally, it can be presumed that simplified versions of dynamic wave (i.e. diffusive wave) cannot be used to model the extreme urban flow at all. They are not able to represent the flood flow also in the complex natural river valley (Szydłowski 2001b) as well as in the built-up structure of city area.

Acknowledgment

The author would like to express his sincere thanks to the research group of PIS (Polo Idraulico et Strutturale), ENEL-CESI, Milan (Italy), for the experimental data necessary to make this work possible. Dr. Guido Testa of CESI SPA is gratefully acknowledged for all information and permission to present the CESI results in this paper.

References

Abbott M. B. (1979), *Computational Hydraulics: Elements of the Theory of Free-Surface Flows*, Pitman, London.

Bermudez A. and Vazquez M. E. (1994), Upwind methods for hyperbolic conservation laws with source terms, *Computers and Fluids*, 23, 1049–1071.

Glaister P. (1993), Flux difference splitting for open-channel flows, *International Journal for Numerical Methods in Fluids*, Vol. 16, 629–654.

Morris M. W. (editor) (2000), *Final Report – Concerted Action on Dambreak Modelling*, HR Wallingford Ltd., Wallingford.

Potter D. (1977), *Computational Physics*, PWN, Warsaw (in Polish).

Roe P. L. (1981), Approximate Riemann solvers, parameters vectors and difference schemes, *Journal of Computational Physics*, 43, 357–372.

Szydłowski M. (2001a), Two-dimensional shallow water model for rapidly and gradually varied flow, *Archives of Hydro-Engineering and Environmental Mechanics*, Vol. 48, No. 1, 35–61.

Szydłowski M. (2001b), Simulation of dam-break event in the Toce River Valley, *Proceedings of the VII International Symposium on Water Management and Hydraulic Engineering*, Międzybrodzie Żywieckie, Poland, 103–110.

Szydłowski M. (editor) (2003), *Mathematical Modelling of Dam-Break Hydraulic Effects*, Monographs of Water Management Committee of Polish Academy of Science, Vol. 22, Warsaw.

Szymkiewicz R. (2000), *Mathematical Modelling of Open Channel Flows*, PWN, Warsaw (in Polish).

Testa G. (editor) (1999), *Proceedings of CADAM Project Meeting*, Milan, Italy.

Toro E. F. (1997), *Riemann Solvers and Numerical Methods for Fluid Dynamics*, Springer-Verlag, Berlin.