Archives of Hydro-Engineering and Environmental Mechanics
Vol. 50 (2003), No. 1, pp. 37–57

Numerical Simulation of Rapidly Varied Water Flow in the 'Wild River' Type Water Slide

Kazimierz Burzyński, Michał Szydłowski

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

(Received February 12, 2003; revised April 17, 2003)

Abstract

The numerical analysis of the water flow along the 'Wild River' type water slide is presented. As the mathematical model of the free surface flow – shallow water equations are assumed. In order to solve the equations, when transient, rapidly varied flow is present, the numerical scheme based on finite volume method is applied. The numerical simulation of water slide flow is computed on unstructured, triangular mesh. The results of calculation are examined against flow parameters observed on the real object installed in water park in Sopot. Generally good agreement between measured and calculated results was observed. Moreover, the calculations are compared to experimental data available due to physical modelling. As the similarity between physical phenomena of flow within water slide and in the river valley after dam-break event is observed, the investigation was realized within the framework of the State Committee for Scientific Research 6 P06S 041 21 project.

Key words: rapidly varied flow, water slide, mathematical modelling, shallow water equations, finite volume method

1. Introduction

Numerical simulations have become an essential component of most projects dealing with hydrodynamics developments. The increasing role of mathematical modelling can be observed due to continuously improving numerical methods and computer power. In recent years considerable effort was devoted to one- and two dimensional flows in open channels and reservoirs, but numerical simulations can be also very useful for water slide designing. The water flow through that kind of hydraulic structure is usually rapidly varied, therefore transitions between subcritical and supercritical flows are often observed. The rapidly varied flow presents an important problem for the design engineers due to the presence of moving, standing and oblique water surface shocks, depressions and their interactions and reflections. The proper prediction of possible location of these hydraulic phenomena and accurate estimation of water depth and velocity can be necessary to design the water slide configuration and channel walls heights to avoid accidents during slide operation. Until quite recently, the investigation of the water flow in such relatively complex structures, as the water slides are, was possible only thanks to physical modelling (Burzyński and Szydłowski 2002). In order to simulate the water flow along the slide, the mathematical model of the phenomenon must be assumed. Basing on the results of physical modelling, the shallow water equations (SWE) are assumed as the model of the unsteady flow in water slide. This choice was made because the water motion along the slide, where some reservoirs are present, must be analysed in at least two dimensions in the plan. The third dimension was neglected in the mathematical model despite the vertical velocity component observed during experiments. However, if SWE in conservative forms are used and the internal structure of hydraulic discontinuities is omitted, the model can be adopted to simulate the transient, rapidly varied flow problem (Cunge et al. 1980, Glaister 1993, Szydłowski 1998, 2001). The simplification mentioned makes the solution easier ensuring sufficient quality of the results. The most common methods of solving SWE are the finite difference method (FDM) and finite element method (FEM). FDM algorithms do not conserve mass precisely and require special schemes to resolve flow discontinuities (Cunge et al. 1980). FEM conserves mass over the entire domain, but not within each element or at each node. In addition, the basic schemes of the method are useless for rapidly varied flow modelling, when hydraulic jumps are present. The oscillations near the discontinuities usually destroy the calculations. In order to avoid this the scheme based on finite volume method (FVM) is applied. In this method the integral form of the shallow water equations in computational cells is solved. Therefore, mass and momentum are conserved in each cell, even for the presence of flow discontinuities. The method was successfully applied to simulate the rapidly varied flow due to dam-break (Bermudez and Vazquez 1994, Glaister 1993, Szydłowski 1998, 2001). Except for handling discontinuities the numerical solution of SWE for rapidly varied flow poses other numerical problems, such as proper source terms approximation. In the method applied to simulate the water slide flow, some special techniques for abrupt bathymetry and friction term integration are applied. The numerical algorithm ensures also the flexibility of calculation domain approximation due to unstructured triangular mesh used in the simulation. The model was verified and examined against experimental data (Szydłowski 2002). Some tests showed the efficiency of the method, particularly for flooding dry areas. It seems that the model can also be satisfactorily used to simulate the flow in the 'Wild River' water slide, being a part of the 'Świat Wodny Nemo' water park in Sopot (Burzyński and Szydłowski 2002). Moreover, comparison between data computed and that observed on the real object has completed verification of the rapidly varied two-dimensional flow model prepared in the framework of the State Committee for Scientific Research 6 P06S 041 21 project. The verification was possible due to hydraulic similarity between dam-break flow and unsteady flow in the system of channels and reservoirs as is the 'Wild River' object.

2. Governing Equations and Numerical Solution Method

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

$$\mathbf{U}=\begin{pmatrix} h \\ uh \\ vh \end{pmatrix}, \quad \mathbf{S}=\begin{pmatrix} 0 \\ -g\,h\,(S_{ox}-S_{fx}) \\ -g\,h\,(S_{oy}-S_{fy}) \end{pmatrix}, \tag{2a, b}$$

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

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

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

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).

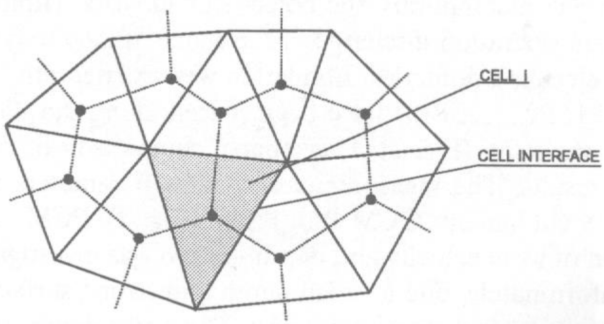


Fig. 1. Discretization of calculation domain

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

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

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. In order to calculate the fluxes $\mathbf{F}_r$ the solution of the approximated Riemann problem proposed by Roe (1981) is used. Description of the method is available in the literature (Glaister 1993, Toro 1997) therefore it is omitted here. The source term vector $\mathbf{S}$ contains two sorts of elements dependent on bottom and friction slopes, respectively. Both of them pose numerical integration problems. In order to ensure the proper bottom slope approximation it is up-winded in the same way as fluxes $\mathbf{F}$ (Bermudez and Vazquez 1994). The second source term (friction term) 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 by a two-step explicit scheme of time integration. Details of the numerical scheme were presented by Szydłowski (1998, 2001).

3. Numerical Simulation of Water Slide Flow

A 72 m long 'Wild River' water slide consists of four channels conveying water between five basins (Fig. 2). The first and last reservoirs constitute the water inflow-outflow system. The others are pools where taking a rest during sliding is possible. The total bottom level difference at start and finish sections is equal to 4.3 m ensuring 6% average bottom slope. The 'Wild River' is built in concrete but the bottom and walls of channels and reservoirs are covered by coats of epoxy resins, thus the Manning friction coefficient can be assumed equal to 0.004 $m^{1/3}$s. According to project assumptions, the object should work within the semi-closed cycle with constant operation discharge in the range of 0.60 to 0.80 m^3/s. The test run of the real object and numerical simulation were carried out for 0.68 m^3/s. The control points (P1, P2, …, P13) have been placed along the slide to control the flow parameters variation (Fig. 2). They enable comparison of the measurements and simulation results. The measurements of flow parameters were carried out using the NIVUS equipment: OCM Pro, Probe (Fig. 3), PCM and PVM meters.

The variation of local velocity and depth in time was investigated at points P5, P8 and P12. Unfortunately, due to small depth and strong surface oscillations the water level measurements were unsuccessful. The water depth was also measured inside reservoirs (points P3, P7, P10). Apart from the measurements mentioned the arrival time of the propagating wave front was controlled at points P2, P4, P6, P9, P11 and P13.

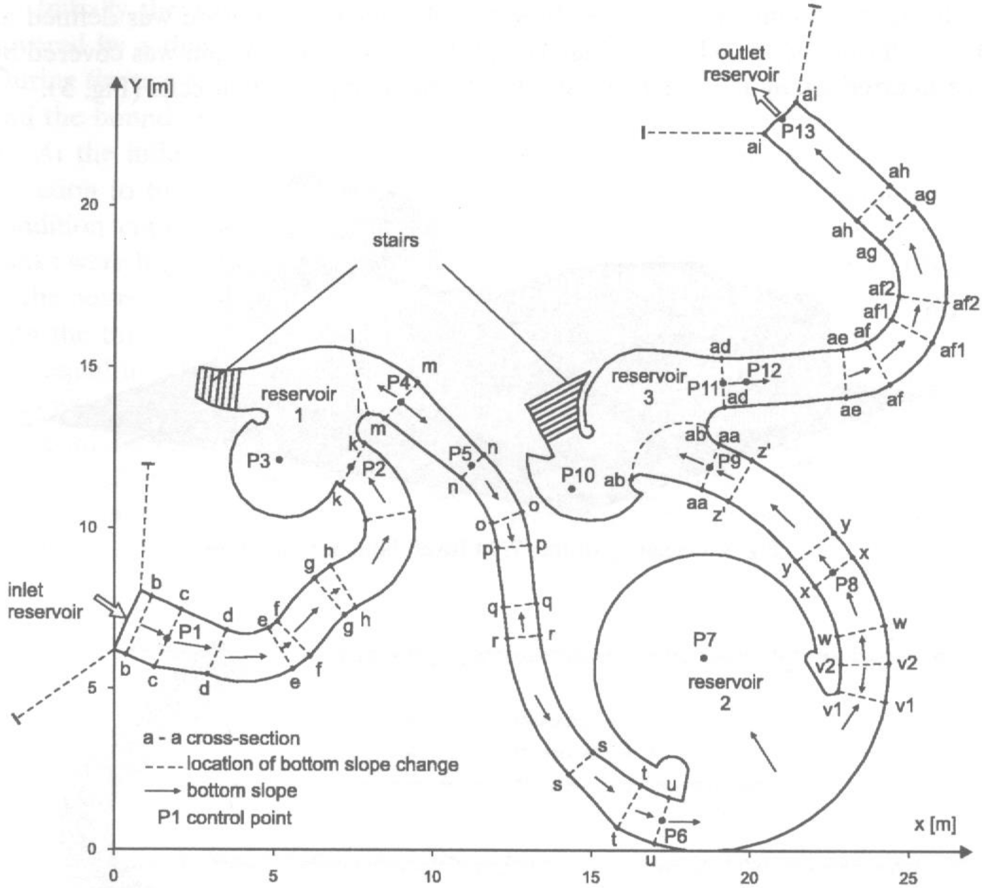


Fig. 2. Scheme of the water slide path

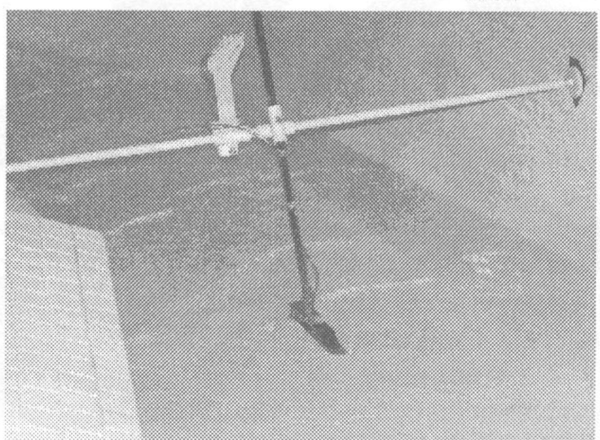


Fig. 3. OCM Pro flow-meter and Probe water level meter (NIVUS)

In order to simulate the water flow, the geometry of the slide was defined as Digital Terrain Model (DTM) (Fig. 4) and the calculation domain was covered by unstructured triangular mesh composed of 8946 computational cells (Fig. 5).

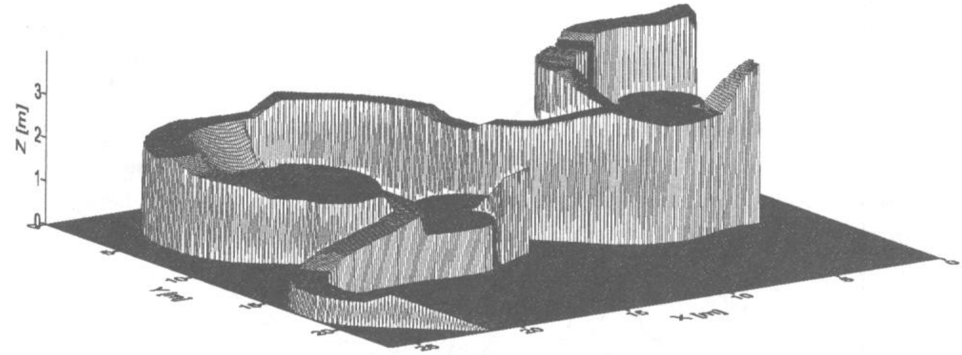


Fig. 4. Geometry of the 'Wild River' bottom from DTM

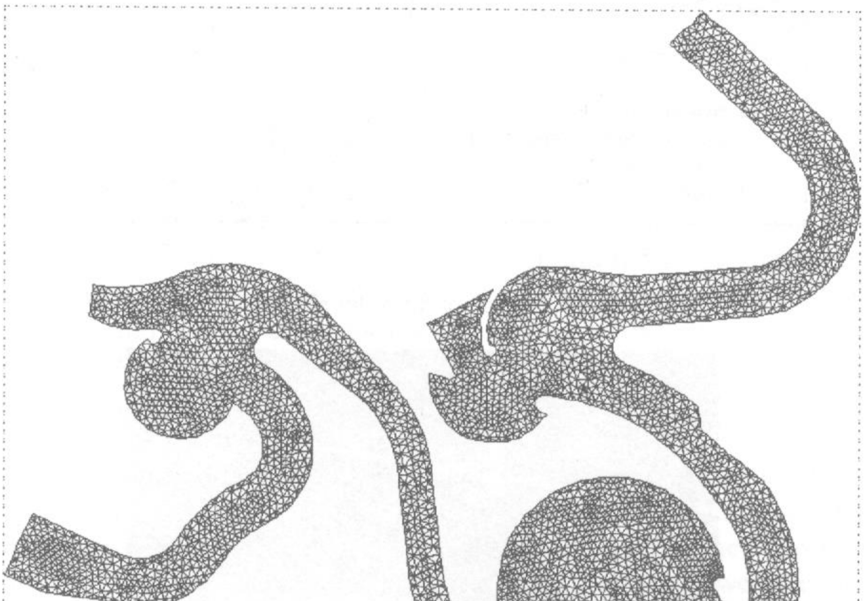


Fig. 5. Numerical mesh of 8946 computational cells

Initially the water slide was empty so the whole calculation domain could be covered by a thin water film (0.0001 m) simulating the dry bottom of the slide. During the test run the flow through the inlet and outlet sections was supercritical and the boundary conditions were imposed in accordance with observed data.

At the inflow section (point P1) the water depth and velocity in the normal direction to the boundary were measured (Fig. 6). At the outflow boundary the condition could be neglected due to supercritical flow. The channel and pool banks were high enough for there to be no overtopped during the flow so the rest of the boundaries were treated as closed and slip. The calculation was carried out with the time step $\Delta t = 0.01$ s ensuring numerical stability. The simulation time was equal to 600 s. After that time each pool was filled with water and the flow became steady.

In addition, before the test on the real object, the 'Wild River' water slide was reproduced in 1:10 scale physical model (Burzyński and Szydłowski 2002). The model reproduced all details of the real slide geometry and enabled comparison of the numerical results of water flow with observations.

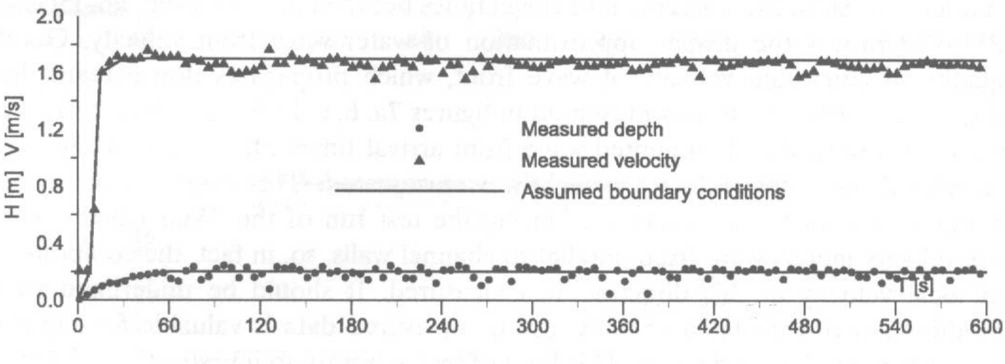


Fig. 6. Inflow boundary conditions

The analysis of numerical simulation results can be divided into two parts. In the first one, the hydraulic parameters of unsteady flow are examined. Such flow could be observed at the beginning of the water slide start-up, when the system was being filled with water. First the water appeared in the inflow reservoir due to pumping. However, such a flow in the outflow reservoir, was not analysed in the simulation. After filling the reservoir, the flow over the crest has caused the water wave propagation downstream the open channel system. The wave front, moving on the dry bottom, can be compared to flood wave propagation in the river valley after catastrophic dam collapse. From this point of view the arrival time of propagating wave front is essential information. Measured using timers and computed wave front arrival times to some control points along the slide are presented in Table 1. Generally good conformity between measured and calculated time values can be observed.

Table. 1. Wave front arrival times

CONTROL POINT	ARRIVAL TIME [s]		TIME INCREMENT [s]	
	measured	computed	measured	computed
P2	8.16	9.84		
P4	20.21	21.85	12.05	12.01
P6	31.57	33.61	11.36	11.75
P9	43.20	45.69	11.63	12.08
P11	58.17	57.44	14.97	11.75
P13	70.14	67.77	11.97	10.33

For validation purposes it is more important to consider the travel time of the wave between two points rather than the exact arrival time at each one of them. Therefore, the time increment between each two points was defined. The biggest difference is seen for points P9 and P11. This time represent duration of filling Reservoir No. 3 with water. As the reservoirs generally split the slide path into three separate segments, the disagreement mentioned is not essential for model verification. Sufficient agreement of travel times between points P4–P6, P6–P9 and P11–P13 proves the proper approximation of water wave front velocity. Good quality of computed velocity of wave front, which propagates downstream the slide, is confirmed by results presented in figures 7a, b, c. In this graph consistence between observed and computed wave front arrival times can be seen. However, calculated local flow velocity is slightly overestimated. This may be due to the manner of velocity measurement. During the test run of the 'Wild River' slide, the velocity meters were fixed parallel to channel walls, so, in fact, the component of local velocity in this direction was measured. It should be underlined that examination of calculated velocity against measured data is valuable for rapidly varied flow model verification. This kind of test, contrary to investigation of water level evolution, is difficult to realize outside the laboratory. The reasonably good conformity of observed and calculated velocities completes the verification of the model which was originally developed for dam-break flow simulation (Szydłowski 2001, 2002).

The progress of water wave front during period of unsteady water flow is also presented in Figures 8 and 9. In these pictures the contour plots of water depth and instantaneous velocity field after 10 and 50 seconds of simulation, respectively, are shown.

The computed and measured depth variation in reservoirs is shown in Figure 10a, b, c. The conformity of results is quite good for reservoirs Nos. 1 and 3 (P3, P10). In the first reservoir computed water depth is underestimated by about 0.07 m due to difference of bottom level assumed in the model (taken from project) and built in the real object. Substantial disagreement between computation

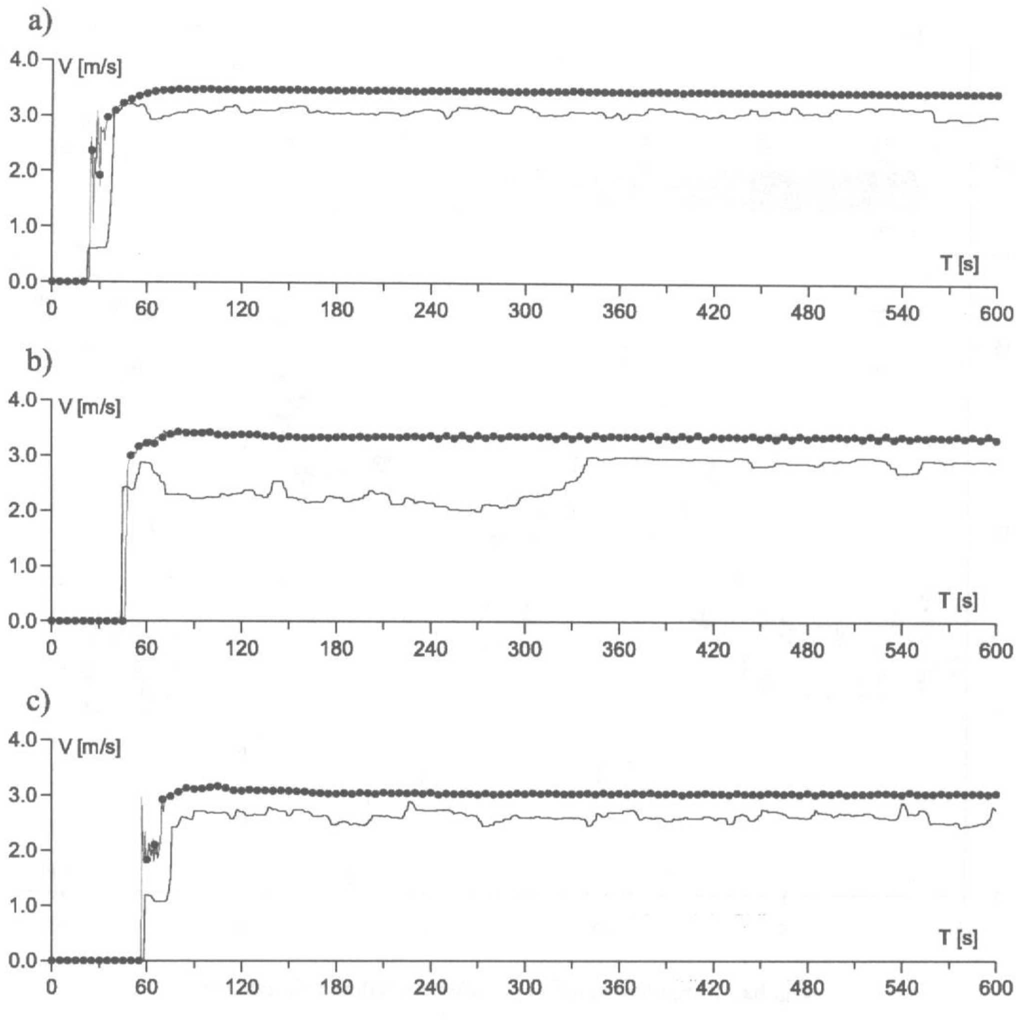


Fig. 7. Measured (solid line) and computed (dots) local flow velocity at control points: a) P5, b) P8, c) P12

and measurements can be observed in Reservoir No. 2 (P7). This side-basin is filling with water slower than predicted in numerical simulation. It seems to be a very sensitive area where abrupt bottom slope change between channel and reservoir plays a meaningful role. The sudden increase of slope causes the three dimensional flow effects which are not represented in the assumed mathematical model at all.

After about ten minutes from the water slide system start-up the flow can be assumed to be steady. However, due to changes of slide geometry and strong oscillation of water depth and local velocity it is not a stationary phenomenon. The results of numerical simulation of steady flow were examined against obser-

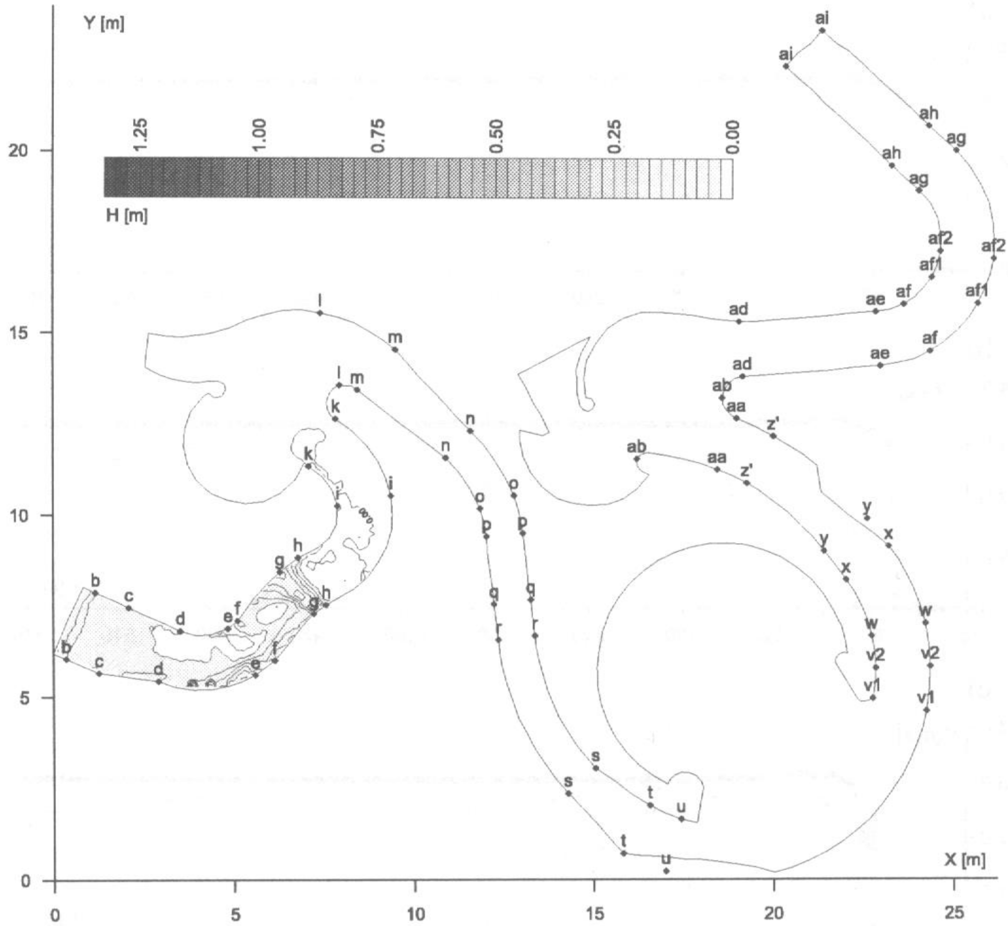


Fig. 8a. Computed water depth after $t = 10$ s of simulation

vation on the real object in the water park in Sopot and on the physical model installed in the Laboratory of Hydro- and Environmental Engineering of Gdańsk University of Technology (Burzyński and Szydłowski 2002). Comparison between results of numerical simulation and observed data can be limited to the assessment of location of some hydraulic local effects. The proper prediction of possible location of complex hydraulic phenomena such as hydraulic jumps, surface swellings and depression seems to be indispensable to design the water slide geometry. Investigation of local values of flow parameters in the whole flow area, although interesting, is not essential from this point of view. The contour map of computed water depth and velocity field along the water slide are presented in Figure 11.

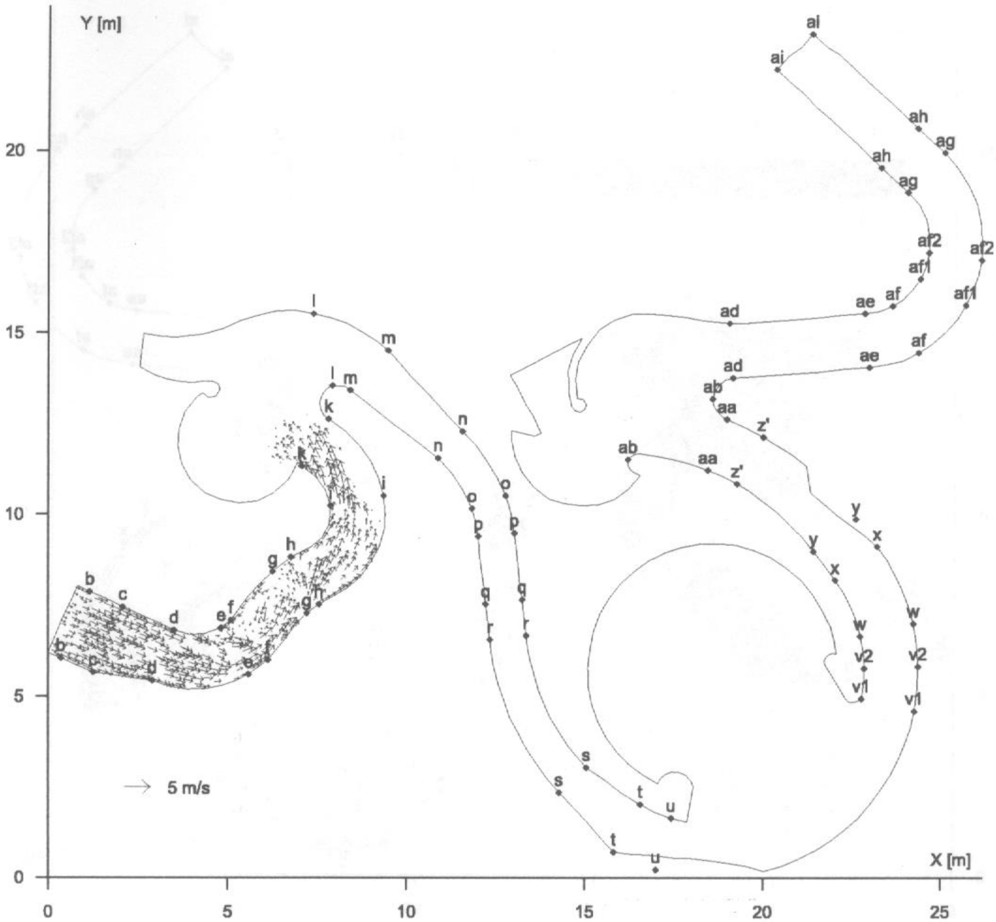


Fig. 8b. Computed velocity field after $t = 10$ s of simulation

Four slide sections were chosen to compare calculated and observed configuration of water surface and velocity. In Figures 12a, b the water flow before Reservoir 1 observed during physical modelling and then during the 'Wild River' object start-up is presented. The main stream is present near the right bank of the channel. Therefore, on that bank water swellings are present. On the left one, near cross-section k–k, a zone of reverse flows can be observed. In the region of abutment, strong whirls are visible in the pictures. The same hydraulic effects can be recognized analysing the results of numerical simulation. The water swellings are represented by locally increasing water depth on the channel bend (Fig. 11a) and reverse flow and whirls are seen on the velocity field map (Fig. 11b).

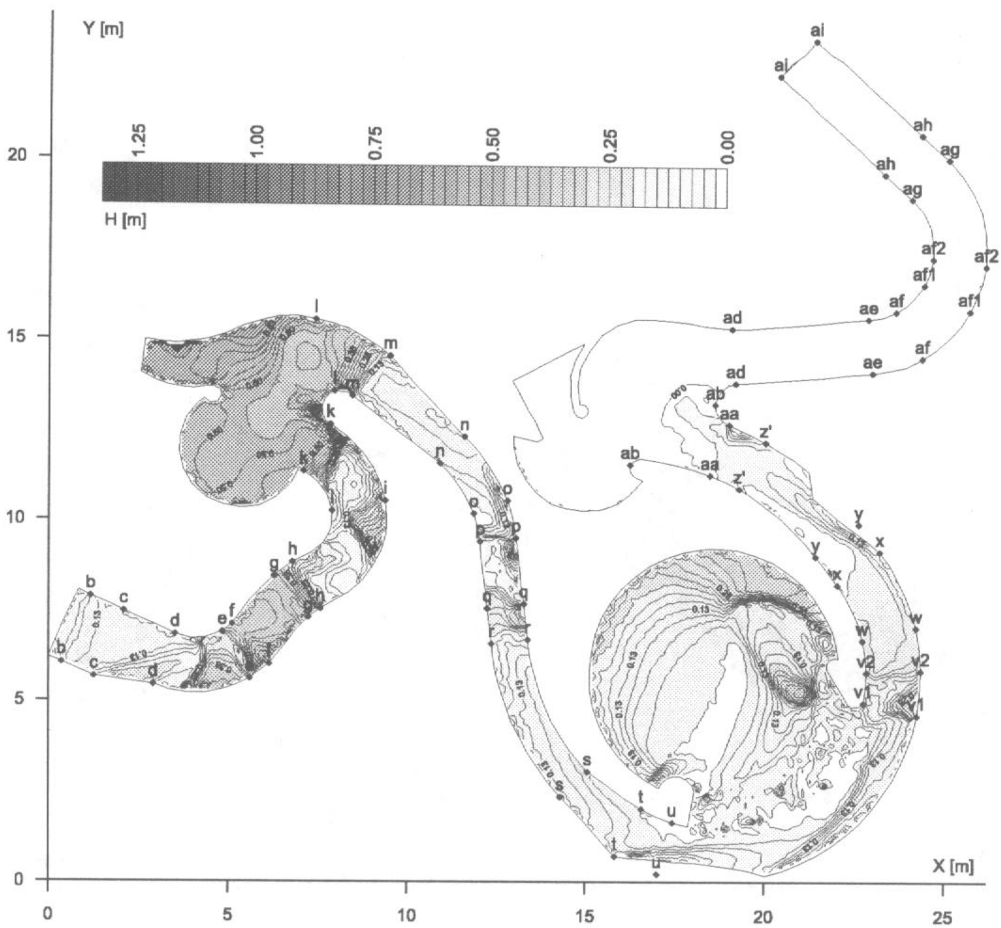


Fig. 9a. Computed water depth after $t = 50$ s of simulation

The complex structure of shocks and depressions of water surface can be observed along the right channel bank near Reservoir 2 (Fig. 13a, b). Those wave-type effects penetrating each other in space are present in this place due to the sudden change of bottom slope and channel geometry. In this area, between cross-sections t–t and v1–v1 (Fig. 2), first channel contraction as in a flume and then enlargement due to side-basin, are present. The following one by one water swellings and depressions are also visible in the contour map of computed water depth (Fig. 11a).

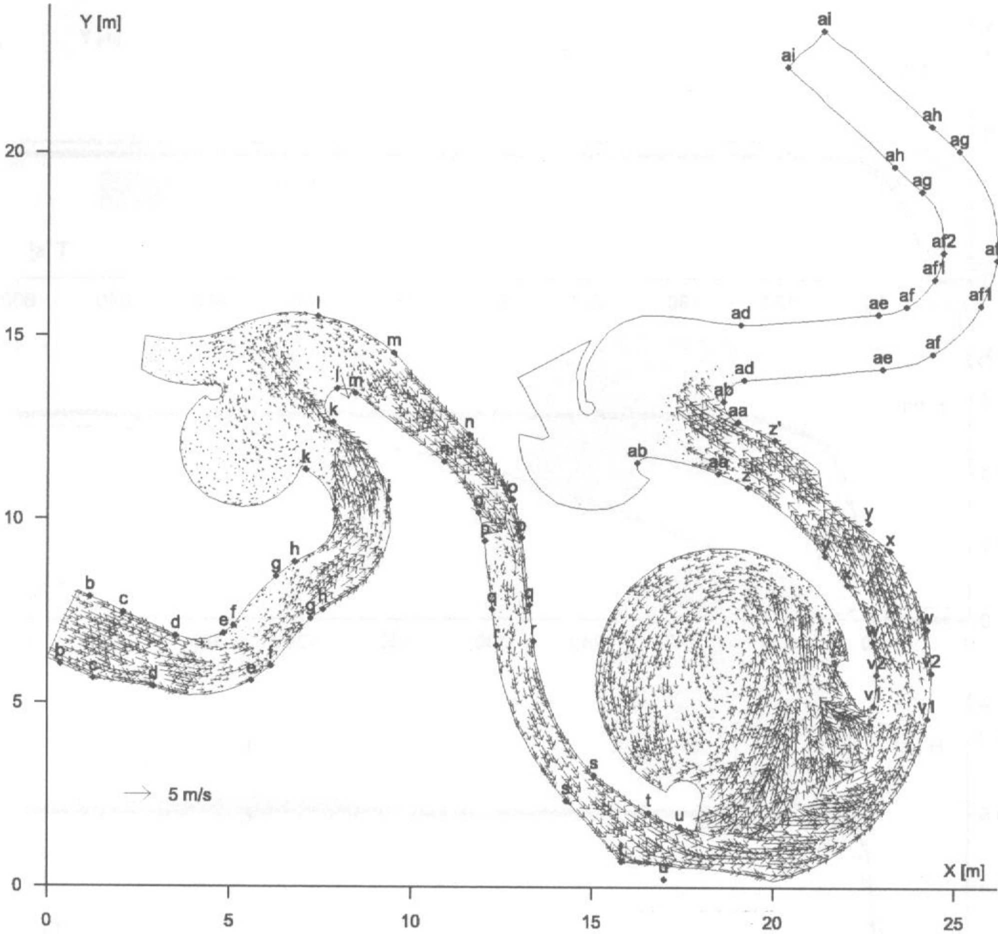


Fig. 9b. Computed velocity field after $t = 50$ s of simulation

In pictures 14a, b configuration of water circulation in Reservoir 3 can be observed. This reservoir is composed of two parts. In the first, where inlet and outlet are located, oscillations of water surface and strong currents are visible. In the second one, located behind the embankment and near the stairs, the flow is quiet and water circulates in the opposite direction. The field of calculated velocity is shown in Figure 11b and is consistent with the one observed.

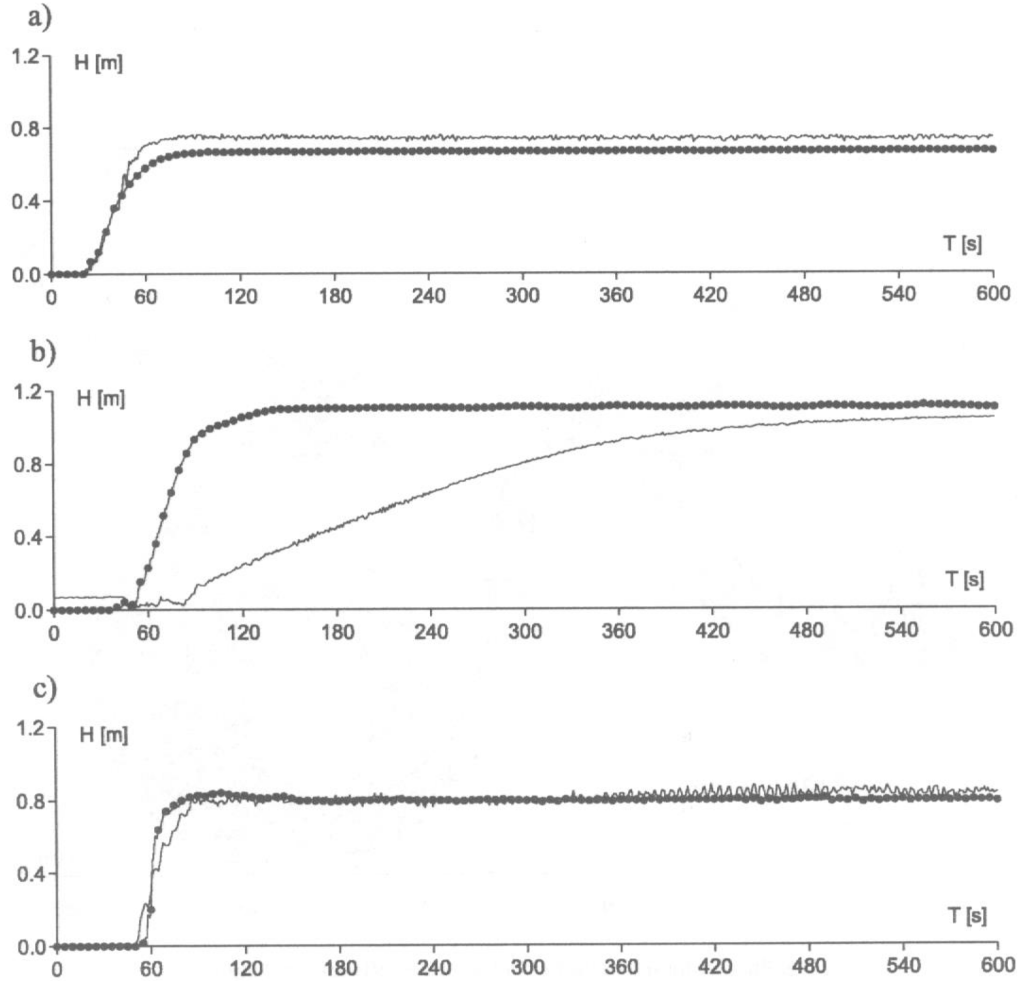


Fig. 10. Measured (solid line) and computed (dots) water depth at control points: a) P3, b) P7, c) P10

Finally in Figures 15a, b water flow at the last bend of the slide channel, observed on the model and real object, respectively, is presented. Along the right bank the significant water swelling with culmination between cross-sections af1–af1 and af2–af2, appearing due to the steep bottom slope and sudden change of channel direction, is visible. The maximum water depth, measured in this region during physical modelling and real object test run, was equal to 0.60 m and 0.62 m. The local velocity was equal to 2.98 m/s and 2.95 m/s, respectively. The calculated flow parameters at culmination point, shown in Figures 11a, b, are equal to 0.605 m and 3.044 m/s. At the left bank spreads the zone of reverse flow and slight depth, with partially exposed bottom of the channel. It is located near the left side of

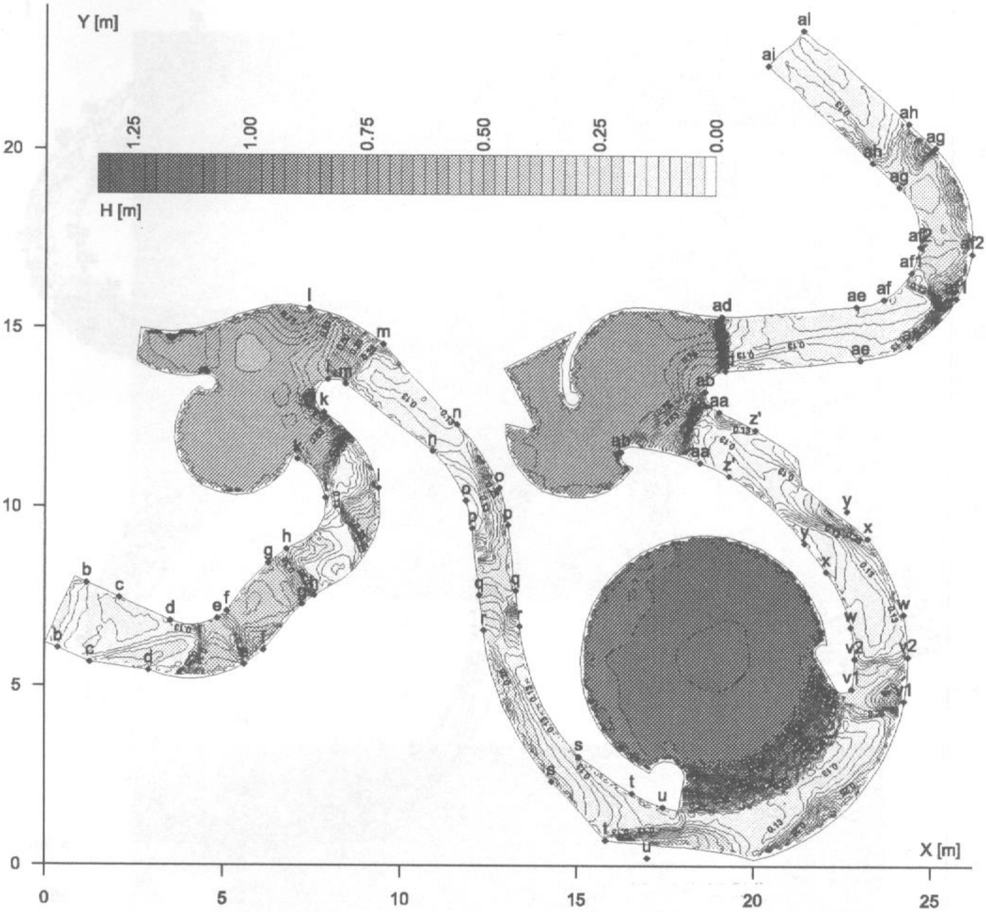


Fig. 11a. Computed water depth after $t = 600$ s of simulation (steady state

cross-section af1–af1. The same hydraulic effects can be recognized analysing the results of numerical simulation. Reverse flow is seen clearly on the velocity field map (Fig. 11b).

4. Conclusions

In the paper the results of numerical simulation of the flow within the 'Wild River' type water slide were presented. The mathematical model was based on equations of unsteady, gradually varied free surface water flow. In the model, the internal structure of hydraulic discontinuities is omitted and hydraulic jumps are defined as the single discontinuities. If the shallow water flow equations in conservative form are used, the model can be adapted to simulate the transient, rapidly varied flow problem observed in the slide during physical modelling and real object test run.

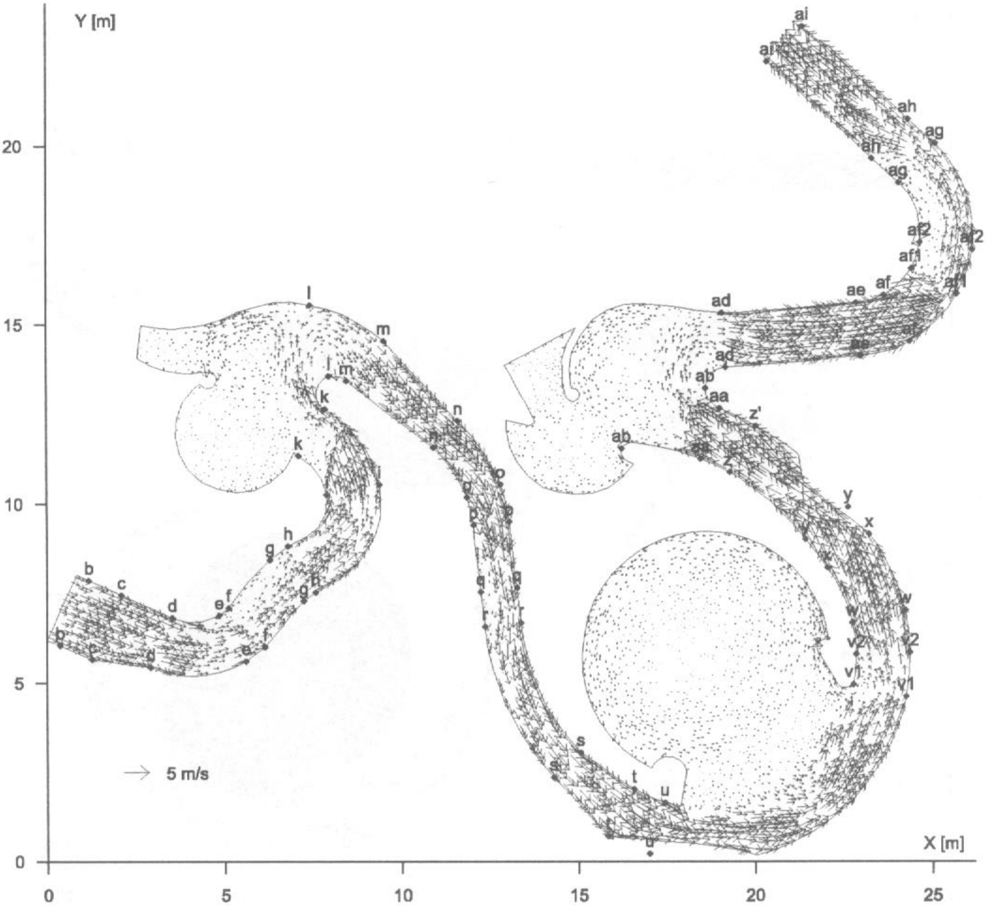
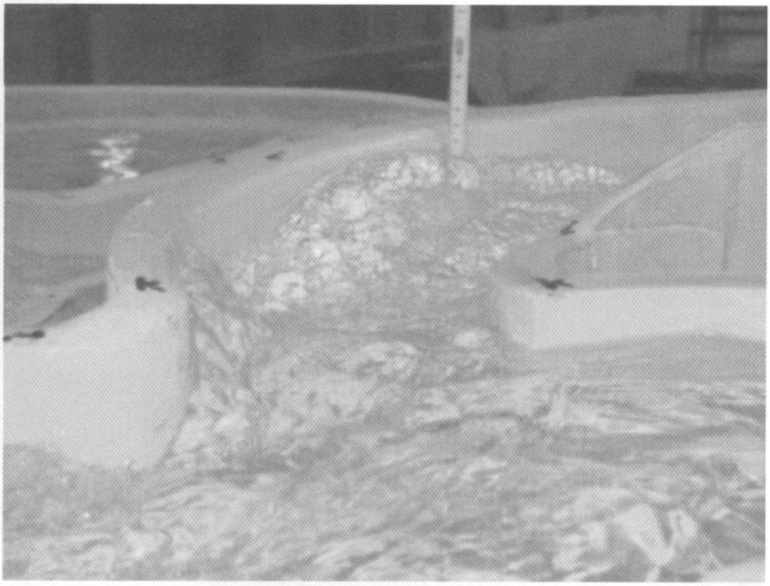


Fig. 11b. Computed velocity field after $t = 600$ s of simulation (steady state)

The calculated hydraulic parameters of flow along slide path were compared to experimental data. Generally sufficiently good qualitative and quantitative agreement of results was observed. The worst results were obtained at regions where sudden change of bottom level plays a meaningful role. Three dimensional effects accompanying the abrupt bottom slope variation are not represented in the assumed mathematical model of the considered phenomenon. In addition a quite good representation of flood wave arrival time to control points was observed which proves the proper reproducing of wave front velocity. It seems that numerical simulation properly represents the majority of hydraulic effects present in the steady and unsteady rapidly varied flow observed in the 'Wild River' object.

a)



b)

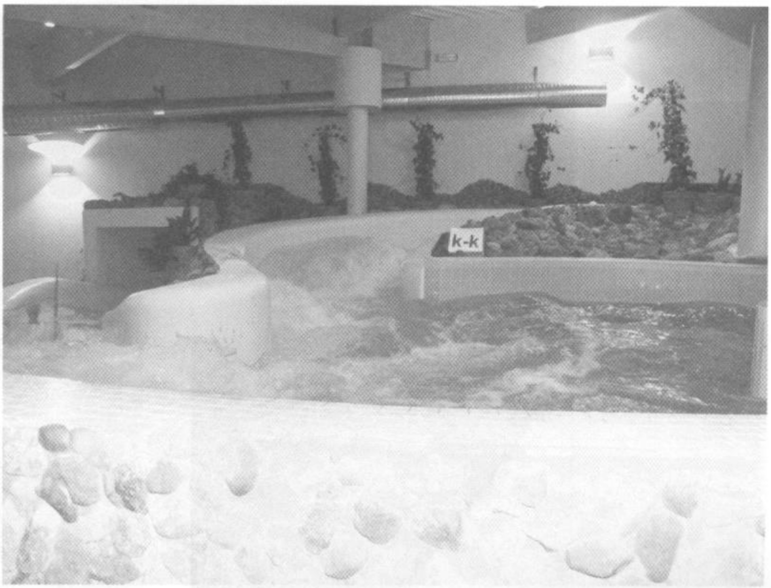
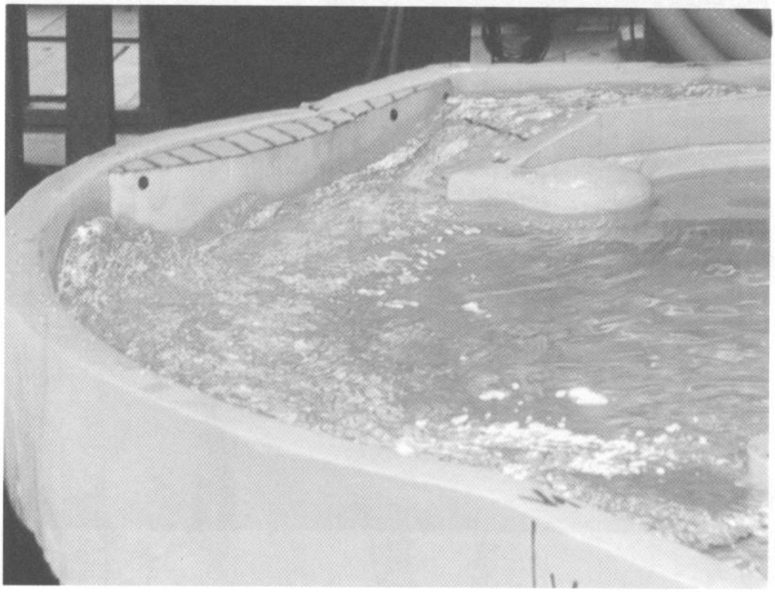


Fig. 12. Water flow between cross-sections i–i and k–k (reservoir 1) physical model (a) and real object (b)

a)



b)

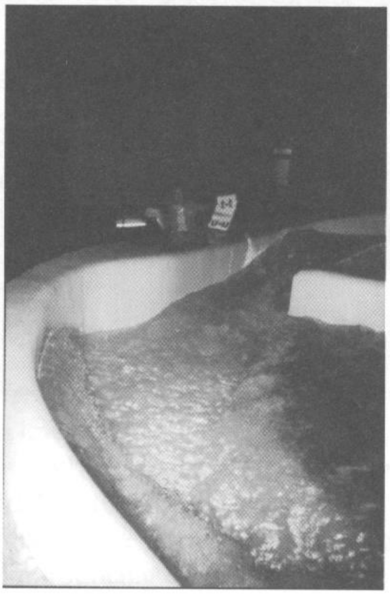


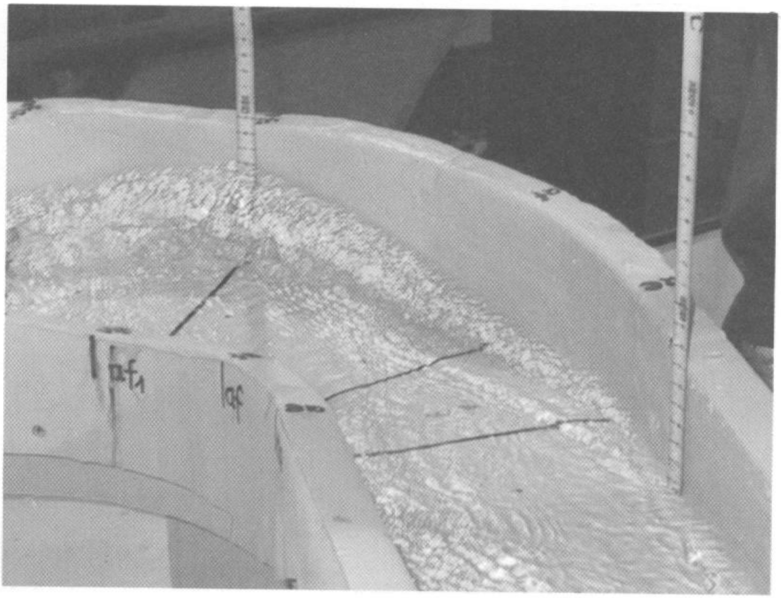
Fig. 13. Water flow between cross-sections u–u and v1–v1 physical model (a) and real object (b)

a)

b)

Fig. 14. Water flow between cross-sections aa–aa and ad–ad (reservoir 3) physical model (a) and real object (b)

a)



b)

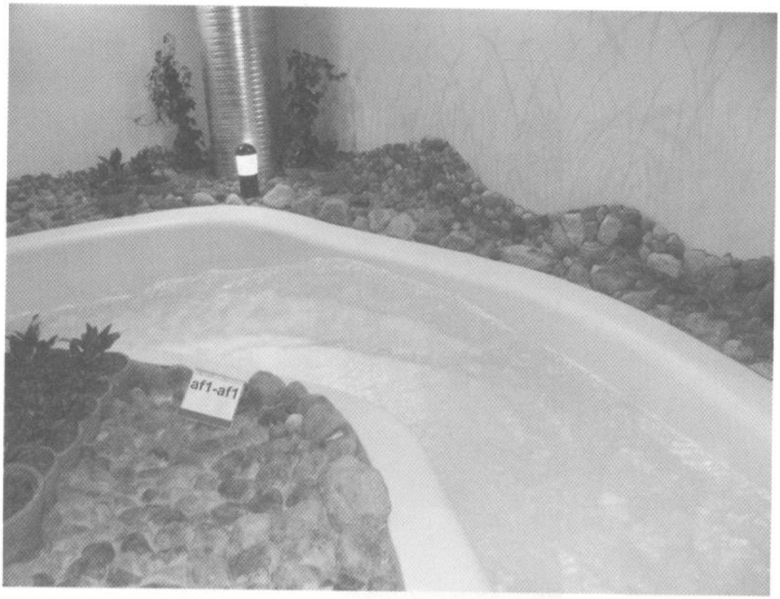


Fig. 15. Water flow between cross-sections ae–ae and af1–af1 physical model (a) and real object (b)

The examination of calculated flow velocity variation against experimental data is valuable for rapidly varied flow model verification. This kind of test is difficult to realize outside the laboratory. The reasonably good conformity of observed and calculated velocities completes the verification of model which was originally developed for dam-break flow simulation in the framework of the State Committee for Scientific Research 6 P06S 041 21 project.

The satisfactory results of flow simulation has proved that the model presented here can be useful not only to answer the question as to how far, how long, how deep and how intensively the river valley can be covered by the water after a dam-break event, but also for water slide designing as well. The proper prediction of hydraulic phenomena and accurate estimation of water depth and velocity can be necessary for preparing emergency and flood protection plans, analysing of flood risk and designing of warning systems in river valleys downstream the dams and planning of water slide configuration, respectively.

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.

Burzyński K. and Szydłowski M. (2002), Physical Modelling of Water Flow in 'Wild River' Type Water Slide, *Archives of Hydro-Engineering and Environmental Mechanics*, Gdańsk, XLIX, No. 4, 37–54.

Cunge J. A., Holly Jr F. M., Verwey A. (1980), *Practical Aspects of Computational River Hydraulics*, Pitman, London.

Glaister P. (1993), Flux Difference Splitting for Open-Channel Flows, *Int. Journal for Numerical Methods in Fluids*, 16, 629–654.

Roe P. L. (1981), Approximate Riemann Solvers, Parameters Vectors and Difference Schemes, *Journal of Computational Physics*, 43, 357–372.

Szydłowski M. (1998), *Numerical Simulation of Rapidly Varied Flow with Discontinuities*, PhD thesis, Gdańsk University of Technology (in Polish).

Szydłowski M. (2001), Two-dimensional Shallow Water Model for Rapidly and Gradually Varied Flow, *Archives of Hydro-Engineering and Environmental Mechanics*, Gdańsk, 48, No. 1, 35–61.

Szydłowski M. (2002), *Modeling of Dam-Break Water Flow*, Agricultural University of Wroclaw, Wroclaw, No. 437, 321–331 (in Polish)

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