SIMULATION CODE OF TRANSIENT FLOW IN SHALLOW WATER

Sara Coelho da Silva

sarasilva@utfpr.edu.br

Eloy Kaviski

eloy.dhs@ufpr.br

Liliana Madalena Gramani

l.gramani@gmail.com

PPGMNE - Postgraduate Program in Numerical Methods in Engineering, Federal University of Paraná - UFPR.

Campus Centro Politécnico, Avenida Cel. Francisco H dos Santos, s/n, Jardim das Américas, Zip code 81530-900, Curitiba, Paraná, Brazil.

Adilandri Mércio Lobeiro

alobeiro@utfpr.edu.br

Mathematics Department, Federal Technological University of Paraná - UTFPR

Campus Campo Mourão, Via Rosalina Maria dos Santos, 1233, Zip code 87301-899, Campo Mourão, Paraná, Brazil.

Abstract. *This scientific paper demonstrates a study on hydrodynamic models described by the shallow water equations with the assistance of computer simulation. The finite volume method (FVM) was adopted for the discretization of the mathematical model, and initial and boundary conditions were imposed. The solution of the system of linearized equations was obtained by a computational code, using the Gauss-Seidel iterative method and the Fortran programming language. Case studies were used to validate the computational model. The implemented code allows the temporal analysis of the level, volume, and flow in reservoirs that have discharge devices in its exits. A suitable graphical interpretation of the studied physical phenomenon was obtained with a good precision guarantee, and a stable solution with low computational cost.*

Keywords: *Computational code, Fortran, Implicit scheme, Finite volume method (FVM), Shallow water equations.*

1 INTRODUCTION

The use of computer techniques to find the solution to complex engineering and physics problems is nowadays a reality, due mostly to the vertiginous development of high-speed computers with great storage capacity. (Maliska, 2013)

Due to this computational availability, there was an increase in programming techniques for the development of algorithms to find solutions to many problems in different areas. According to Souza et al. (2006), the advances in computer capacity in recent years and the study of numerical techniques have boosted a substantial growth of numerical modeling as a tool for studying flows.

As stated by Tannehill et al. (1997), computational fluid dynamics (CFD) is the area of knowledge that deals with numerical simulation of fluid flows, using programming.

In this present paper, it is presented a computational model that was developed to simulate transient two-dimensional water flow in a reservoir, using a simplified mathematical formulation of shallow water equations. For the discretization of the model, the finite volume method is used with upwind spatial discretization and a totally implicit temporal discretization scheme.

According to Cruz and Tucci (2007), the urbanization of cities and the sequential increase in the construction of buildings and execution of paving make the soil impermeable, with great impact on the water flow, such as the occurrence or aggravation of urban flooding.

A possible solution to the problem of flooding is in the control of the flow at its source, through holding reservoirs. Cruz et al. (1998) and Drumond et al. (2014) define such reservoirs as structures of temporary accumulation of rainwater that assist in the reduction of urban floods. Rainwater can be conducted and stored in the reservoirs so that after the drying of the soil occurs, these waters can be discharged by drainage structures into the drainage network.

Nonetheless, Oliveira et al. (2016) point out the difficulty in measuring discharges in watercourses, especially in situations with high speeds, rapid changes in its level, and with difficult access.

In this context, Souza et al. (2006) suggest numerical modeling as a studying tool for hydrodynamic variables, using partial differential equations systems that describe mathematically fluid drainage according to the fundamental laws of physics.

The purpose of the present study is to make use of a computational code to evaluate the variation of level and volume when discharge devices are triggered. There was also an analysis of the flow velocity field in the reservoir and in the internal and external borders, in a transient form and in two directions.

The research is justified by the need of computational simulation for predicting and monitoring the flows in large reservoirs.

The programming of the simulation code was done in FORTRAN programming language due to its high capacity of calculation and data storage.

2 MATHEMATICAL MODEL

According to Vreugdenhil (1994), the hyperbolic partial differential equations system that commands transient incompressible two-dimensional flows in shallow waters is obtained by integrating along the depth of the three-dimensional equations that shape the amount of movement and conservation of mass in a flow. In this research, simplifying hypotheses presented in Mahmood (1975) were adopted: (i) the vertical velocity is small and its corresponding acceleration can be neglected; (ii) the pressure is hydrostatic; (iii) the topography of the base of the reservoir is invariant; (iv) the cross section is always rectangular; (v) there is no friction at the base; (vi) there are no lateral contributions to the flow in the reservoir; (vii) the specific mass and viscosity of the fluid in the reservoir does not change significantly over time and space; (viii) the shear stresses are negligible. Thus, the determining equations of the simplified hydrodynamic model are given by:

$$\frac{\partial h}{\partial t} + \frac{\partial}{\partial x}(uh) + \frac{\partial}{\partial y}(vh) = 0, \tag{1}$$

$$\frac{\partial}{\partial t}(uh) + \frac{\partial}{\partial x}(u^2 h + \frac{1}{2}gh^2) + \frac{\partial}{\partial y}(uvh) = 0, \tag{2}$$

$$\frac{\partial}{\partial t}(vh) + \frac{\partial}{\partial x}(uvh) + \frac{\partial}{\partial y}(v^2 h + \frac{1}{2}gh^2) = 0, \tag{3}$$

where $h(m)$ is the water depth in the reservoir, $u(m/s)$ and $v(m/s)$ are components of the flow velocity, $g(m/s^2)$ is the gravitational acceleration and $t(s)$ is time.

For solving the system of equations it is necessary to establish the initial and boundary conditions.

At the first instant, the h level of the water surface in the reservoir and the flow velocity must be known at each point in the two-dimensional domain.

In the external and internal contours of the domain, the flow and the level of the water blade must be determined.

At the boundaries, the h level is considered to be equal to the height of the water at a point of the nearest discretization domain.

Waterproof walls are allowed in the flow analysis. Therefore, around the internal borders and at the external borders where there are no water discharges, the flow is null. The flow at the external borders, where there are dump devices, is described in accordance with the level h and the discharge coefficient, as stated by Silva et al. (2016).

3 NUMERICAL MODEL

The discretization of the system of equations (1) - (3) was obtained using the finite volume method presented by Malalasekera (2007) and Blazek (2001).

The two-dimensional domain is transformed into a discrete rectangular mesh having spacings $\Delta x, \Delta y$ in the directions x, y, respectively. The discrete points of the W, P, E (from west to east) are separated by the distance Δx, and points S, P, N (from south to north) by Δy, as shown in Figure 1. It is noticed that the general control volume centered in P has an area of $\Delta x \Delta y$, and borders located half a distance of the neighboring centroids, in the west (w), east (e), south (s) and north (n) sides.

As attested by Hirsch (2007), the finite volume method consists of the spatial and temporal integration of the model equations over an arbitrary control volume. The values of the variables

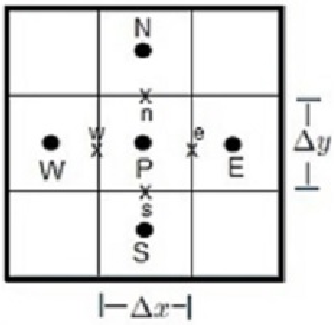


Figure 1: Generic control volumes of the discretized domain.

are obtained in the centroids of the control volumes of the mesh, using a computational code that scans the mesh, indexing the variables according to the centroid in which they are calculated. In this indexation, the point P is identified as (i, j) and the neighboring centroids W, E by $(i-1, j)$, $(i+1, j)$, respectively. Likewise, the centroids S, N, are described by $(i, j-1)$ and $(i, j+1)$.

In spatial integration of the system of equations (1) - (3), the variables on the w, e, s, n sides of the control volume are obtained by interpolation. For the convective terms, a combination of the central interpolation with the upwind is used, identifying the signal of the velocity components on the sides with the coefficients $\alpha_i^{'}$s. The description of these interpolation schemes can be found in Silva et al. (2016).

By adopting a totally implicit formulation in time integration, the discretization obtained for the system of equations (1) - (3) using the finite volume method is given by:

$$a_{hP}h_P + a_{hE}h_E + a_{hW}h_W + a_{hN}h_N + a_{hS}h_S = b_{hP} \tag{4}$$

$$a_{uP}u_P + a_{uE}u_E + a_{uW}u_W + a_{uN}u_N + a_{uS}u_S = b_{uP}, \tag{5}$$

$$a_{vP}v_P + a_{vE}v_E + a_{vW}v_W + a_{vN}v_N + a_{vS}v_S = b_{vP}, \tag{6}$$

whose coefficients of the variables in centroid P are given by:

$$a_{hP} = 1 + a_{PE} + a_{PW} + a_{PN} + a_{PS}, \tag{7}$$

$$a_{uP} = h_P + a_{PE}h_e + a_{PW}h_w + a_{PN}h_n + a_{PS}h_s = a_{vP}, \tag{8}$$

$$b_{hP} = h_P^0 = h(P, t), \tag{9}$$

$$b_{uP} = h_P^0 u_P^0 + \frac{\Delta t}{\Delta x}\frac{1}{2}g[h_w^2 - h_e^2], \tag{10}$$

$$b_{vP} = h_P^0 v_P^0 + \frac{\Delta t}{\Delta y}\frac{1}{2}g[h_s^2 - h_n^2] \tag{11}$$

$$a_{PE} = \frac{\Delta t}{2\Delta x}u_e(1+\alpha_e), \quad a_{PW} = -\frac{\Delta t}{2\Delta x}u_w(1-\alpha_w), \tag{12}$$

$$a_{PN} = \frac{\Delta t}{2\Delta y}v_n(1+\alpha_n), \quad a_{PS} = -\frac{\Delta t}{2\Delta y}v_s(1-\alpha_s), \tag{13}$$

and the coefficients of the variables in the neighboring centroids to P are obtained from equations (12) and (13)) by changing the coefficients signal $\alpha_i^{'}$s.

The solution to the system of linear equations (11) – (13) is computationally obtained using the Gauss-Seidel iterative method. In order to establish a stopping criterion in the iterative calculation, the global residue was computed according to the residues rh, ru, rv rv (difference between the right and left terms of equations (11), (12) and (13), respectively) and the total number of nodes in the mesh, with a tolerance of 10^{-5} and maximum number of iterations $itmax = 1000$.

4 CASES STUDY

The simplifying hypotheses (that were previously described in the study of transient two-dimensional flow in two types of storage reservoir) were adopted for the validation of the numerical solution obtained in Section 3 and for the computational simulation of the model.

At a first case, a rectangular reservoir $(7m \times 7m)$) is taken into consideration, with rigid and impermeable walls and also with two discharge devices, as it shows in Figure 2.

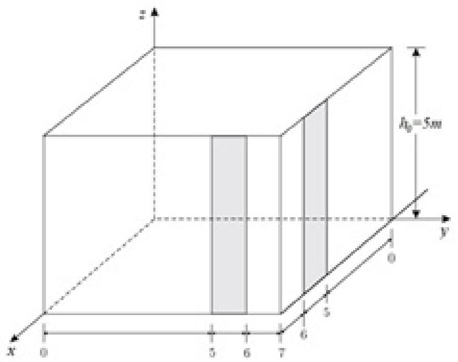


Figure 2: Rectangular-based reservoir with lateral discharge devices.

In a second case, a model closer to reality was adopted, considering a rectangular underground reservoir $(10m \times 7m)$, with nine supporting pillars inside it and with two discharge devices, as shown in Figure 3.

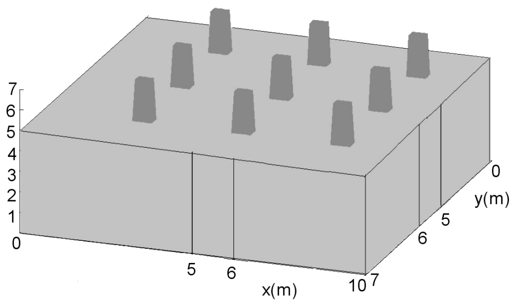


Figure 3: Reservoir with internal pillars and discharge devices.

In both cases, it is initially assumed that the water in the reservoir is at rest and the level of the water surface throughout the reservoir is $5m$ in height. As its walls and pillars are impermeable, the flow of stored water occurs only at the outlets, by the opening of discharge controller devices.

In Case 1, the discharge flow (described by the components of the flow velocity) in the east and north outlets, has the following mathematical formulation:

$$u(7, y, t) = C\sqrt{2gh(7, y, t)}\,, \quad 5 \leq y \leq 6, \tag{14}$$

$$v(x, 7, t) = C\sqrt{2gh(x, 7, t)}, \quad 5 \leq x \leq 6. \tag{15}$$

Due to the length of the reservoir walls in Case 2, the velocity components are given by:

$$u(10, y, t) = C\sqrt{2gh(10, y, t)}\,, \quad 5 \leq y \leq 6, \tag{16}$$

$$v(x, 7, t) = C\sqrt{2gh(x, 7, t)}, \quad 5 \leq x \leq 6, \tag{17}$$

where $C = 0.1$, being the discharge coefficient, $h(m)$ is the water level in the reservoir and $g(m/s^2)$ is the gravitational acceleration.

5 THE COMPUTATIONAL CODE

The computational simulation code was programmed in Fortran, compiled in the Fortran PowerStation 4.0 version, licensed by Microsoft Corporation. This programming language was chosen due to its simplicity, execution speed and high capacity of data storage.

In the initial stage of the code, the main variables are declared as matrices (h, u, v) as well as the secondary variables that serve in the calculation of the main ones. The model data is then read, such as the length in the x, y, and the subintervals numbers of space and time. Then, the initial conditions of the model, h_0, u_0 and v_0, are read.

In Case 1, there was a selection of a structured computational mesh with a spatial spacing of $\Delta x = \Delta y = 0.1m$ and a time step of $\Delta t = 0.001s$. In Case 2, utilized the same time step of Case 1 and a more refined mesh were chosen, with $\Delta x = \Delta y = 0.05m$, to better treat the domain around the pillars. In this case, mathematical logic was used to declare the points of the domain, circumventing the internal pillars.

In both cases, the same initial conditions were adopted, where $h = 5m$ and $u = v = 0$ in all points in the domain.

The program was designed with a data backup. If the code is being compiled for the first time ($backup = 0$), the initial data discussed above is maintained. But if the code has already been compiled ($backup = 1$), the values of the variables obtained in the last calculation are used as initial data. In this way, flow analysis is possible at any time instance.

Subsequently, the variables are calculated in the internal points of the mesh and in the internal and external borders, using loopings that guarantee the sweep of the whole mesh, in each time step. The reassignment of the variables is performed at each step of the temporal looping until the last time step (n_t). In other words, the values obtained for h, u, v at time t assume h_0, u_0, v_0, for $t = 1, \cdots, n_{t-1}$.

As the flow in the outlets is dependent on the h level, the smaller the water surface in the reservoir, the slower is the stream velocity and consequently, smaller the variation of h. In this context, the stopping criterion of the code is to obtain a minimum value for the water level, $h = 10^{-5}m$. If the stopping criterion is not satisfied, new initial values h_0, u_0, v_0 will be used for the calculation of the current variables h, u, v until the stopping criterion is satisfied or the temporary looping is completed.

Figure 4 presents a flowchart that synthesizes the elaborated code, emphasizing the steps sequence and calculations performed.

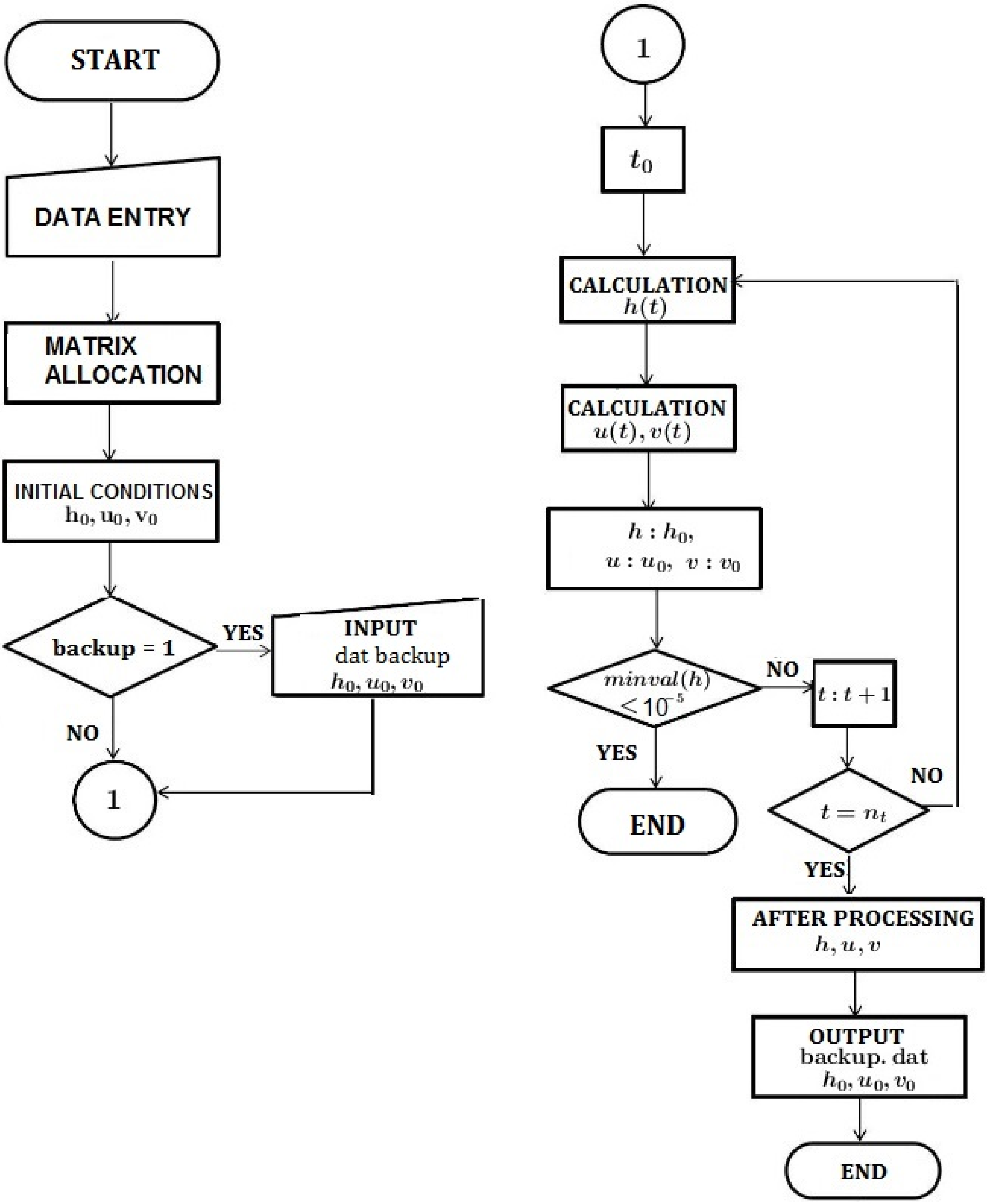


Figure 4: Flowchart of the computational code for the studied model.

6 NUMERICAL VALIDATION

For the validation of the numerical solution obtained, the consistency analysis of the discretized equations was performed, as well as the stability and convergence of the method applied in the presented case study were verified.

In the consistency analysis, there is an expansion in Taylor series of the variables described in centroids E, W, N, S around P, in the linearized equations to retrieve the shallow water equations from the discrete model, calculating the limit with the spatial and temporal spacing tending to zero.

As for the stability, initial and boundary conditions were established in accordance with the investigated physical phenomenon, and a totally implicit temporal discretization scheme was adopted, which is in general unconditionally stable. In both cases, it was observed that there was no numerical instability or error accumulation in the simulation.

For the convergence of the method analysis, there was a use of the refinement of the space mesh and an estimate for analytical solution obtained by Richardson extrapolation. It is verified that in both analyzed cases, the shorter the length and width of the finite volumes of the mesh, the smaller is the relative error in the calculation of the volume V of the reservoir, as shown in Figure 5.

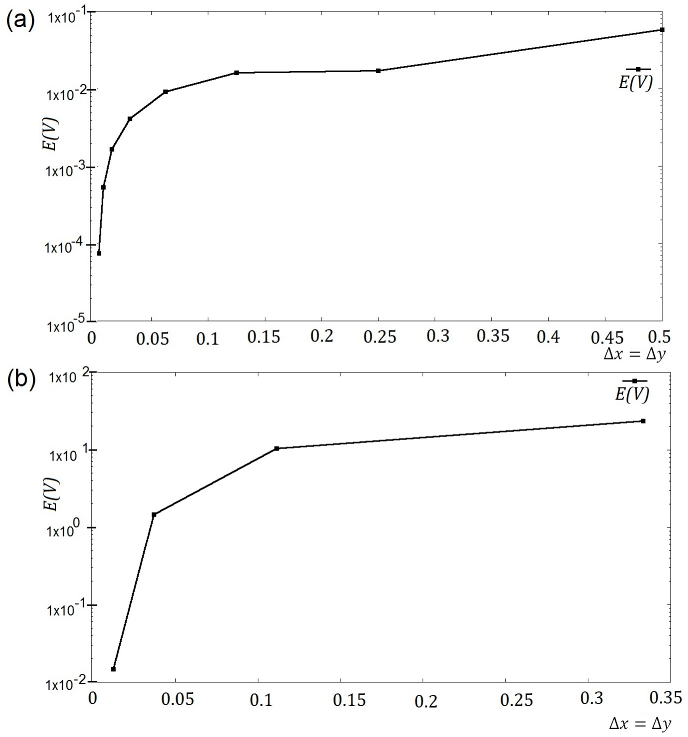


Figure 5: Convergence of the method analysis for Case 1 (a) and Case 2 (b).

7 RESULTS

For a graphical visualization of the results obtained by the code implemented in Fortran, the free software Gnuplot was used.

The elaborated code allowed not only the validation of the numerical solution but also the verification of the fidelity of the hydrodynamic model with the phenomenon of total emptying of the reservoir and water flowing around the pillars, according to Figures 6 - 9.

Due to the discharge devices presented in Case 1, the volume in the reservoir decreases exponentially, as shown in Figure 6.

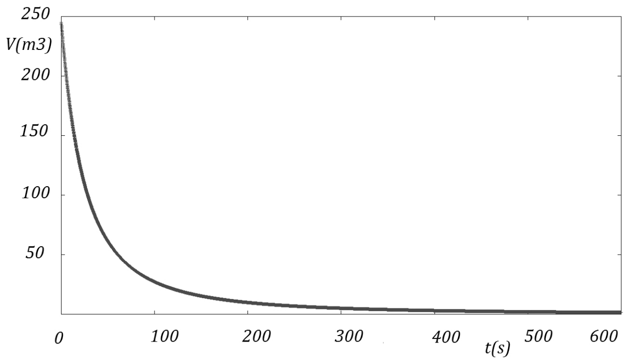
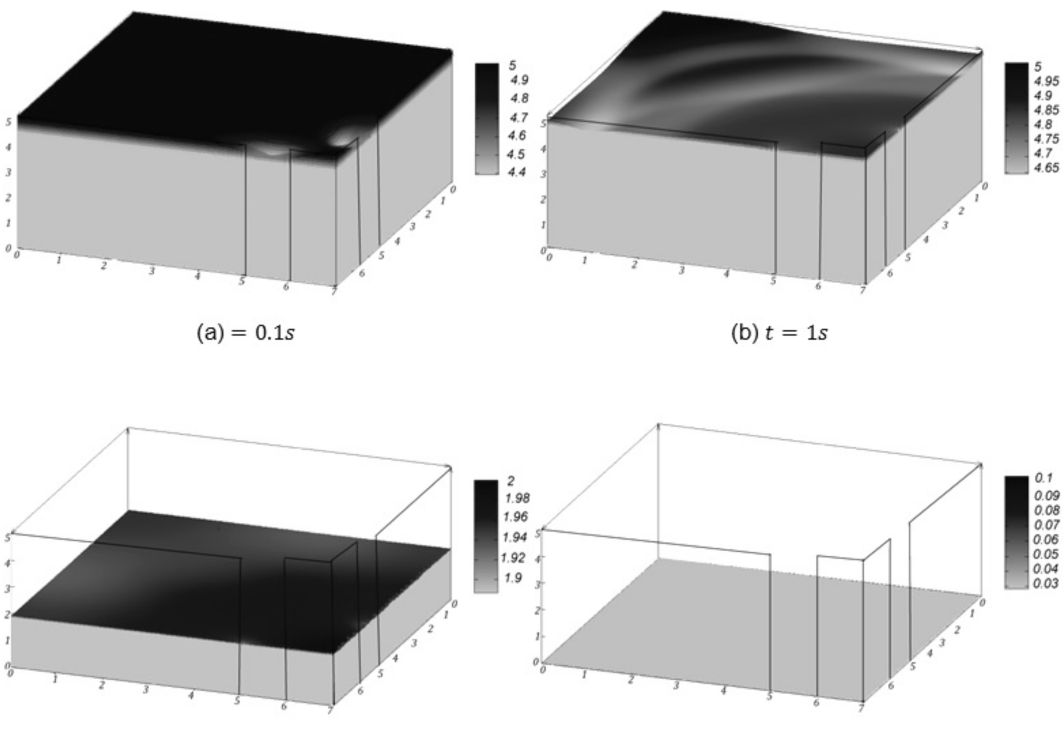


Figure 6: Time variation of the volume in the reservoir in Case 1.

At the instant $t = 0.1s$, the water level in almost the entire reservoir is $h = 5m$, with a small slope at the bottom of the reservoir only in the vicinities of the outlet openings. As for $t = 1s$, small elevation waves are observed in the reservoir and a small slope on the water level. For $t = 300s$ the water is practically leveled, and within the first $600s$ the level is practically null throughout the reservoir, according to Figure 7.



(a) = $0.1s$ (b) $t = 1s$

(c) = $300s$ (d) $t = 600s$

Figure 7: Temporal variation of the reservoir level in Case 1.

In Case 2, small waves were initially observed in the reservoir and a significant slope in the water level in the first $220s$ of simulation, according to Figure 8.

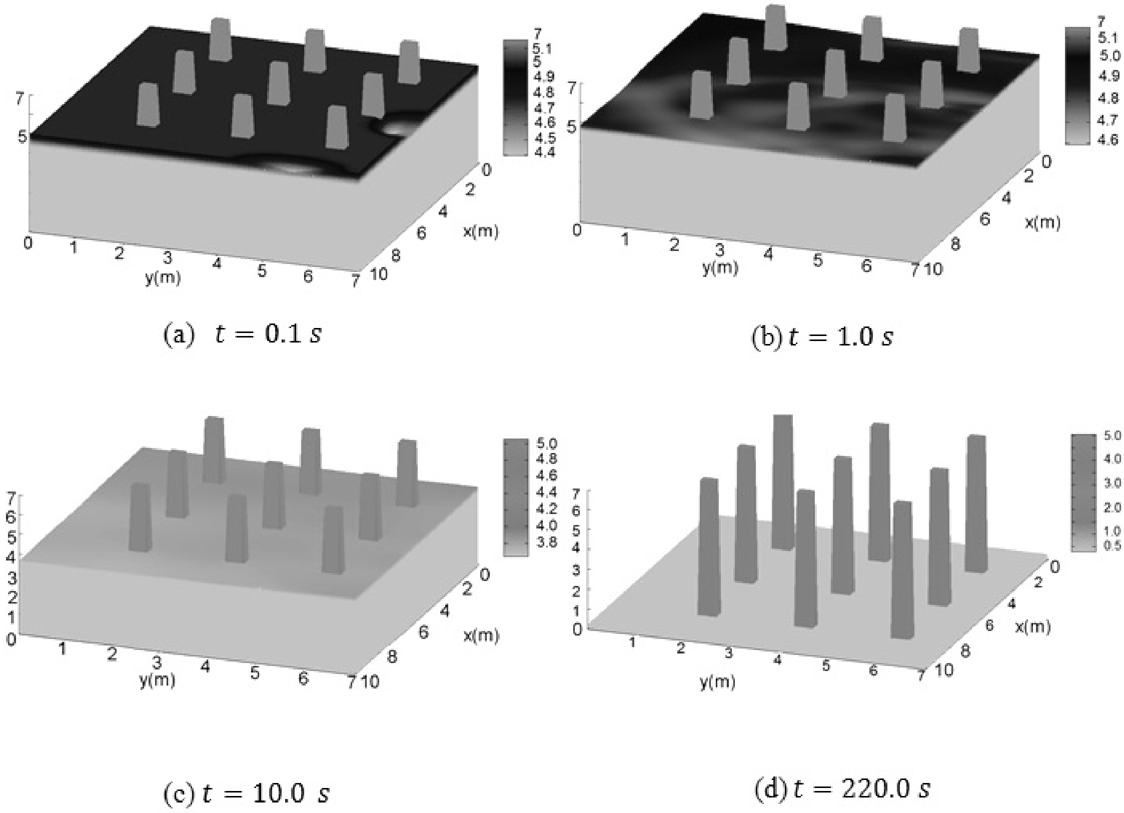


Figure 8: Water level in the reservoir at different time moments.

Figure 9 shows the intensity and direction of the flow at all points of the reservoir at $t = 2.0s$, highlighting the stream around the pillars, in Case 2. It is noticed a higher intensity of the flow in the exits spreading along the entire reservoir over time and the expected flow boundary around internal borders.

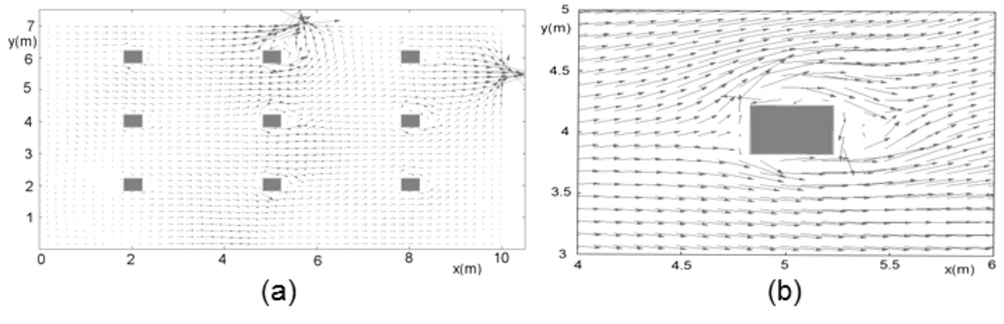


Figure 9: Stream velocity field (a), detailed around the central pillar (b).

8 CONCLUSIONS

The hydrodynamic model presented in this paper efficiently simulates the two-dimensional flow in a storage reservoir, allowing forecasting the emptying time and controlling the discharge flows.

As in what concerns to the time of simulation of the flow and stability, the implicit discretization scheme revealed to be superior to the explicit scheme presented in Silva et al. (2016). The incorporation of appropriate boundary conditions was necessary for accuracy to the physical model analyzed.

The velocity field information is crucial for the control of the discharge flow in storage reservoirs. Other case studies can be investigated by changing the initial and boundary conditions.

This model can be attached to a pollutant transportation model to analyze the variation of the specific mass of the compound in the reservoir.

ACKNOWLEDGEMENTS

The first author is grateful to the financial support given from the Coordination for the Improvement of Higher Education Personnel (CAPES, in Portuguese) and the Federal Technological University of Parana (UTFPR), for granting leave to attend a PhD in the Postgraduate Program in Numerical Methods in Engineering (PPGMNE) of the Federal University of Parana (UFPR).

REFERENCES

Blazek, J., 2001. *Computational fluid dynamics: principles and applications.* Oxford, UK: Elsevier.

Cruz, M. and Tucci, C. E. M., 2007. Otimização das obras de controle de cheias em uma bacia urbana. *Revista Brasileira de Recursos Hídricos*, vol. 12, n. 2, pp. 63-80.

Cruz, M., Tucci, C. E. M., and Silveira, A., 1998. Controle do escoamento em lotes urbanos. *Revista Brasileira de Recursos Hídricos*, vol. 3, n.4, pp. 19-31.

Drumond P. P.; Coelho, M. M. L. P.; Moura, P., 2014. Investigação Experimental dos Valores de Coeficiente de Descarga em Tubos de Saída de Microrreservatórios. *Revista Brasileira de Recursos Hídricos*, vol. 19, n.2, pp. 267-279.

Hirsch, C., 2007. *Numerical Computation of Internal and External Flows: The Fundamentals of Computational Fluid Dynamics.*(2nd ed.). Oxford, UK: Elsevier.

Mahmood, K. and Yevjevich, V., 1975. *Unsteady flow in open channels.* Fort Collins, Colo: Water Resources Publications.

Maliska, C. R., 2013. *Transferência de Calor e Mecânica dos Fluidos Computacional.*(2nd ed.).Rio de Janeiro, BR: LTC.

Oliveira, F.A.; Pereira, T. S. R.; Soares, A. K.; Formiga, K. T. M., 2016. Uso de Modelo Hidrodinâmico para determinação de vazão a partir de medições de nível. *Revista Brasileira de Recursos Hídricos*, vol. 21, n.4, pp. 707-718.

Silva, S. C.; Lobeiro, A. M.; Kaviski, E.; Gramani, L. M., 2016. Modelagem do fluxo hidrodinâmico bidimensional num reservatório usando o método dos volumes finitos. *I Simpósio de Métodos Numéricos em Engenharia.*

Souza, L.; Villela, S. M.; Schulz, H. E., 2006. Simulação computacional do assoreamento em uma armadilha de sedimento. *Revista Brasileira de Recursos Hídricos*, vol. 11, n.4: pp. 163-174.

Tannehill, J. C.; Anderson; D. A.; Pletcher, R. H., 1997. *Computational Fluid Mechanics and Heat Transfer.* PA: Taylor & Francis.

Versteeg, H. K.; Malalasekera, W., 2007. *An Introduction to Computational Fluid Dynamics.* (2nd ed.) Pearson Education Limited.

Vreugdenhil, C. B., 1994. *Numerical Methods for Shallow Water flow,* vol. 13. Springer Science +Business Media Dordrecht.