11-2001

Implementation of the Compressible Flow Solution Methodology for Solving 2D Shallow-Water Flow Problems

Joaquin Moran
Sheridan College, joaquin.moran@sheridancollege.ca

Jose A. RIncon
University of Zulia

Proceedings of
2001 ASME International Mechanical Engineering Congress and Exposition
November 11-16, 2001, New York, NY

IMECE2001/HTD-24300

IMPLEMENTATION OF THE COMPRESSIBLE FLOW SOLUTION METHODOLOGY FOR SOLVING 2D SHALLOW-WATER FLOW PROBLEMS

Joaquin E. Moran
Computational Simulation Laboratory
School of Mechanical Engineering
University of Zulia
Maracaibo, Venezuela
Email: jmoran@luz.ve

Jose A. Rincon
Computational Simulation Laboratory
School of Mechanical Engineering
University of Zulia
Maracaibo, Venezuela
Email: jrincon@luz.ve

ABSTRACT

This paper concerns with the implementation of the compressible flow solution methodology for solving 2D shallow water flow problems. It is well known that in both cases, the continuity and momentum conservation equations look quite similar, but depth replaces density of compressible flow, and the Froude number will replace the Mach number. Thus, any mass imbalance produces a change in depth equivalent to the density change for compressible flow. It is possible to combine momentum and continuity equations to obtain a predictor-corrector algorithm for establishing the depth field. However, as the Froude number increases, the governing equations change their character from elliptic to hyperbolic, with a parabolic transition at a Froude number of unity and this change is not reflected in the equivalent classical pressure-correction equation, which keeps its elliptic character. The extension of incompressible (SIMPLE-based methods) to compressible flows, incorporates a convection-like term (wave velocity related) to the pressure-correction equation. The drawback of the extension of the pressure-correction to compressible flows was the poor shock-capturing capability, which is due mainly to the treatment of the convective terms in the conservation equations. In this work, a high order bounded treatment of the convective terms along with the depth-correction for all Froude numbers is implemented. A numerical solution is presented for all Froude numbers, and it is compared with benchmark problems

INTRODUCTION

The 2D shallow water equations are similar to the compressible 2D equations with the depth term replacing the density, and the Froude number replacing the Mach number. Thus, any mass imbalance produces a change in depth equivalent to the density change for compressible flow. This similarity makes possible to use the techniques for compressible flow solution for solving the shallow water equations [1,2]. Technique for solving compressible flow can be classified in two categories, density-based and pressure-based procedures. Most of the successful numerical density-based schemes are time marching methods, and they utilize upwind approaches based on the particle and wave velocities. Explicit numerical schemes such Lax-Wendrof and Mac McCormack have been successfully used to solve shallow water equations, but they demand an appreciable informatics' resources due to the CFL restrictions [3,4]. Implicit schemes look like a more attractive alternative for long-term simulations. On the other hand, methods for incompressible flows using a pressure-based segregated procedure have been well established. The latter, combine momentum and continuity equations to obtain a predictor-corrector algorithm for establishing the pressure field. However, as the Mach number increases, the governing equations change their character from elliptic to hyperbolic, with a parabolic transition at a Mach number of unity and this change is not reflected in the conventional pressure-correction equation, which keeps its elliptic character. The SIMPLE (Semi Implicit Method for Pressure Linked Equations) algorithm of Patankar and Spalding [5] is widely used for solving incompressible flow problems and it has been utilized in its original form to solve 2D shallow waters equations by Zhou [6] and Bisson [7]. The extension of incompressible (SIMPLE-based methods) to compressible flows incorporates a convection-like term (wave velocity related) to the pressure-correction equation [8,9]. This extra term does not seem to have been incorporated in the depth-correction schemes according to the author's revision. The drawback of the extension of the pressure-correction to compressible flows has been the poor shock-capturing capability, which is due mainly to the treatment of the convective terms in the conservation equations. Convective terms are discretized using first order upwind (or its variants) to avoid numerical oscillations, which appear when central second order schemes are used. Thus, the use of first order upwind (this includes power law) for the convection treatment is equivalent to the addition of artificial viscosity in the density-based methods, but without the control of the user. Two strategies can be used in order to incorporate a high order treatment of the convective terms; one consists of using an implicit central high order and then adds sufficient artificial viscosity to damp the oscillations. The other one consists of using implicit first order upwind (very stable) and add the corresponding anti-diffusive contribution. The former has been implemented through a user-controlled blending factor that combines lower and higher-order schemes [12] and the latter is used along with an automatic switch to avoid overestimations [13, 9]. To sum up, most approaches use artificial viscosity explicitly added or implicitly added in form of first order upwind and the correct hyperbolic nature of the depth-correction is not incorporated in the pressure-based methodology. In this work the deferred correction of Khosla et al [10] and a high order bounded treatment of the convective terms [9] is implemented along with the depth-correction for all Froude numbers. The model is verified with other numerical results and experimental data.

NOMENCLATURE

A,B	coefficients in the discretization equations
C	coefficients in the depth correction equation
Fr	Froude number
Pe	Peclet number
f	Coriolis factor
g	gravitational acceleration
h	water depth
u,v	depth average velocity components
Z_b	bed elevation above horizontal datum
ρ	density
τ_b	shear stress (at bed)
ν	kinematic viscosity
Γ	diffusion coefficient
	Superscripts
*	quantity at the latest iteration
°	quantity at the previous time step
	Subscripts
E,W,N,S	refers to neighbors of the P grid point
P	main grid point
ue, uw, ...	refers to velocity components at the faces
t	refers to turbulent terms

GOVERNING EQUATIONS

The general governing equations for unsteady 2D shallow water flow in Cartesian co-ordinates may be written in conservative form as [6,7]:

$$\frac{\partial h}{\partial t}+\frac{\partial (hu)}{\partial x}+\frac{\partial (hv)}{\partial y}=0 \tag{1}$$

$$\begin{aligned}\frac{\partial (hu)}{\partial t}+\frac{\partial (huu)}{\partial x}+\frac{\partial (hvu)}{\partial y}&=-gh\frac{\partial h}{\partial x}-gh\frac{\partial Z_b}{\partial x}-\frac{\tau_{bx}}{\rho}\\&+fv+\frac{\partial}{\partial x}\left(h\nu_t\frac{\partial u}{\partial x}\right)+\frac{\partial}{\partial y}\left(h\nu_t\frac{\partial u}{\partial y}\right)\end{aligned} \tag{2}$$

$$\begin{aligned}\frac{\partial (hv)}{\partial t}+\frac{\partial (huv)}{\partial x}+\frac{\partial (hvv)}{\partial y}&=-gh\frac{\partial h}{\partial y}-gh\frac{\partial Z_b}{\partial y}\\&-\frac{\tau_{by}}{\rho}-fu+\frac{\partial}{\partial x}\left(h\nu_t\frac{\partial v}{\partial x}\right)+\frac{\partial}{\partial y}\left(h\nu_t\frac{\partial v}{\partial y}\right)\end{aligned} \tag{3}$$

Where u and v represent the depth-averaged velocity components in the x and y directions; h is the water depth; g is the gravitational acceleration; f is a Coriolis factor; Z_b is bed elevation above horizontal datum; ν_t is the depth-averaged turbulence viscosity and ρ is the fluid density. τ_{bx} and τ_{by} represent the bed shear stresses in the x and y directions respectively, usually described in terms of the depth-averaged velocities as:

$$\tau_{bx}=\rho\left(\frac{gn^2}{h^{1/3}}\right)u\sqrt{u^2+v^2} \tag{4}$$

$$\tau_{by}=\rho\left(\frac{gn^2}{h^{1/3}}\right)v\sqrt{u^2+v^2} \tag{5}$$

Where n represents the Manning coefficient. Additionally, the depth-averaged turbulence viscosity is calculated as

$$\upsilon_t=0.067h\left[\left(\frac{gn^2}{h^{1/3}}\right)\left(u^2+v^2\right)\right]^{1/2} \tag{6}$$

SOLUTION METHOD

Following the procedure described by Versteeg and Malalasekera [11], a staggered grid arrangement is used but the convective terms are treated using a bounded second order upwind scheme [9]. As all scheme of order higher than one produce oscillatory solutions, a combination of an implicit first order upwind and an explicit second order contribution is used for the convection treatment *i.e.* a deferred correction [10] as it was previously described. Equation (2) and (3) are discretised using a finite volume approach as:

$$A_P u_P = A_E u_E + A_W u_W + A_N u_N + A_S u_S + A_O \qquad (7)$$

$$B_P v_P = B_E v_E + B_W v_W + B_N v_N + B_S v_S + B_O \qquad (8)$$

Where

$$A_E = \frac{(h\nu_t)_e \Delta y}{\Delta x} + max\{0, -(hu)_e \Delta y\} \qquad (9)$$

$$A_W = \frac{(h\nu_t)_w \Delta y}{\Delta x} + max\{0, (hu)_w \Delta y\} \qquad (10)$$

$$A_N = \frac{(h\nu_t)_n \Delta x}{\Delta y} + max\{0, -(hv)_n \Delta x\} \qquad (11)$$

$$A_S = \frac{(h\nu_t)_s \Delta x}{\Delta y} + max\{0, (hv)_s \Delta x\} \qquad (12)$$

$$A_P = A_E + A_W + A_N + A_S + h^{t-1}\Delta x \Delta y/\Delta t + \Gamma_b \Delta x \Delta y \sqrt{u_P^2 + v_P^2} \qquad (13)$$

$$A_O = -\frac{g}{2}\Delta y(h_e + h_w)\{(h_e + Z_e) - (h_w + Z_w)\} + \frac{h^{t-1}\Delta x \Delta y}{\Delta t} u^{t-1} + fv \qquad (14)$$

$$B_E = \frac{(h\nu_t)_e \Delta y}{\Delta x} + max\{0, -(hu)_e \Delta y\} \qquad (15)$$

$$B_W = \frac{(h\nu_t)_w \Delta y}{\Delta x} + max\{0, (hu)_w \Delta y\} \qquad (16)$$

$$B_N = \frac{(h\nu_t)_n \Delta x}{\Delta y} + max\{0, -(hv)_n \Delta x\} \qquad (17)$$

$$B_S = \frac{(h\nu_t)_s \Delta x}{\Delta y} + max\{0, (hv)_s \Delta x\} \qquad (18)$$

$$B_P = B_E + B_W + B_N + B_S + h^{t-1}\Delta x \Delta y/\Delta t + \Gamma_b \Delta x \Delta y \sqrt{u_P^2 + v_P^2} \qquad (19)$$

$$B_O = -\frac{g}{2}\Delta y(h_n + h_s)\{(h_n + Z_n) - (h_s + Z_s)\} + \frac{h^{t-1}\Delta x \Delta y}{\Delta t} v^{t-1} - fu \qquad (20)$$

Anti-diffusive contributions are added as source terms using the previous iteration values. The depth-velocity coupling is achieved by the extended SIMPLE algorithm, which combines the continuity equation and a depth correction h' instead of the pressure correction. Thus, the h' values can be obtained by solving the following equation:

$$C_P h'_P = C_E h'_E + C_W h'_W + C_N h'_N + C_S h'_S + C_O \qquad (21)$$

Where

$$C_E = D_{ue}\Delta y(h_P^* + h_E^*) + max\{0, -2u_e \Delta y\} \qquad (22)$$

$$C_W = D_{uw}\Delta y(h_P^* + h_W^*) + max\{0, 2u_w \Delta y\} \qquad (23)$$

$$C_N = D_{vn}\Delta x(h_P^* + h_N^*) + max\{0, -2v_n \Delta x\} \qquad (24)$$

$$C_S = D_{vs}\Delta x(h_P^* + h_S^*) + max\{0, 2v_s \Delta x\} \qquad (25)$$

$$C_P = C_E + C_W + C_N + C_S \qquad (26)$$

$$C_O = [(h_P^* + h_E^*)u_e^* - (h_P^* + h_W^*)u_w^*]\Delta y + [(h_P^* + h_N^*)v_n^* - (h_P^* + h_S^*)v_s^*]\Delta x + (h_P^o - h_P)\Delta x \Delta y/\Delta t \qquad (27)$$

$$D_{ue} = gh_e \Delta y / a_P \qquad (28)$$

$$D_{vn} = gh_n \Delta x / b_P \qquad (29)$$

In which the superscript ^{o} denotes the value of a quantity at the previous time step and the superscript * denotes the values at the latest iteration.

The equivalent Peclet number for equations (22, 23, 24 and 25), which represents the ratio between convective and diffusive contribution, is given by

$$Pe = \frac{2u_e^* a_P}{g h_e \Delta y} \qquad (30)$$

and since $\frac{a_P}{\Delta y} \propto u$ then $Pe \propto \frac{1}{Fr^2}$

Thus, the relative importance of the two parts of the mass flux correction depends of the kind of flow. For subcritical flow (Froude<1) this factor is several orders of magnitude larger than 1, so the diffusion-like term dominates and the other one is negligible. For supercritical flows with high Froude numbers, the contrary is true. In this case the convection-like term dominates in agreement with the right hyperbolic behavior.

The coefficients of the depth correction equation for subcritical flows represent an approximation to the Laplacian operator and the pressure information travels in all directions. For critical and supercritical the presence of convective terms in the depth correction equation makes that pressure waves travel along the characteristic lines from inlet to outlet boundaries and the solution of the h′ algebraic equations system is unique. Thus, the same method can be used for the entire domain since it automatically adjusts to the local nature of the flow.

BOUNDARY CONDITIONS

Inflow boundaries: The number of variables that can be specified at an inflow boundary depends on the number of incoming characteristics at that surface. Thus, for example for subcritical inflow two variables must be specified. Whereas, if the inflow is supercritical, all variables must be specified.

Outflow boundaries: Similar to the inflow boundary condition, the number of variables to be specified at an outflow boundary is equal to the number of incoming characteristic. For a subcritical outflow this requires specification of one variable at that boundary. The most common practice is to define the depth. All other variables are extrapolated from the interior of the calculation domain. If supercritical conditions prevail at that exit boundary, nothing needs to be specified, including depth equation boundary condition, this, according to the hyperbolic nature of the h′ equation.

TEST OF THE MODEL

In order to demonstrate the validity of the model, three numerical experiments are carried out: The hydraulic jump in a straight channel which allows to show the shock-capturing features of the method and for which experimental data is available; and two cases of hydraulic jump in channels with different flow regimes along them.

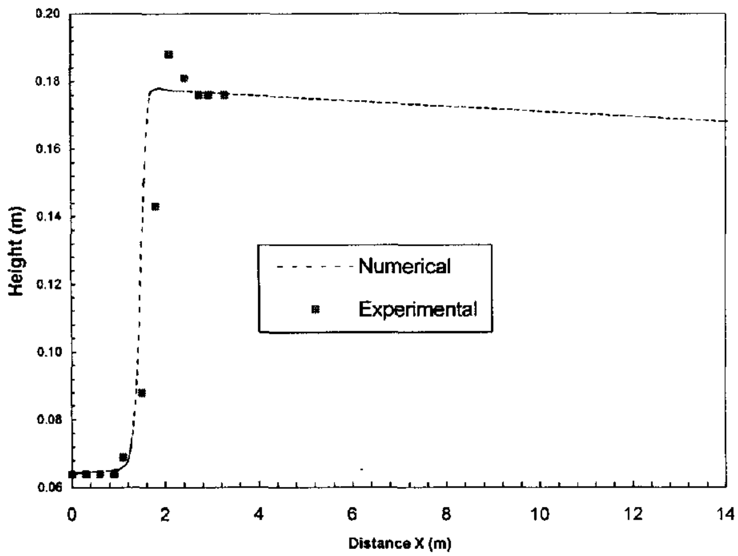

Figure 1. Comparison between numerical and experimental results.

1. Hydraulic jump in a straight channel

The hydraulic jump is a basic physical phenomenon in open channel and river flows characterized by a strong turbulent mixing and energy dissipation. It occurs when flow changes from a supercritical to a subcritical state, thus, it constitute a good test problem for the method. The values used in this problem correspond to those utilized by Gharangik and Chaudhry [14]. The dimensions of the straight rectangular channel for which the experimental data was obtained are 0.46 m wide and 14.1 m long. A uniform grid of 49x7 nodal points was used to approximate the domain. The velocities at the inflow boundary were fixed as u=1.82 m/s and v=0 m/s and the inlet depth was fixed in 0.064 m which corresponds to a Froude number of 2.3, for this case the hydraulic jump is enforced and subcritical condition is likely than occurs at the outlet, then the depth has been fixed in 0.168 m/s.

Figure 1 shows the comparison of the numerical results (at the central line of the channel) obtained using the proposed method and the experimental data of Gharangik and Chaudhry. The results agree very well with the experimentaldata except for the points at the top of the hydraulic jump where the numerical solution exhibits a sort of clipping, typical of bounded schemes and that deserves special attention. The important issue is that no oscillations are observed before or after the jump.

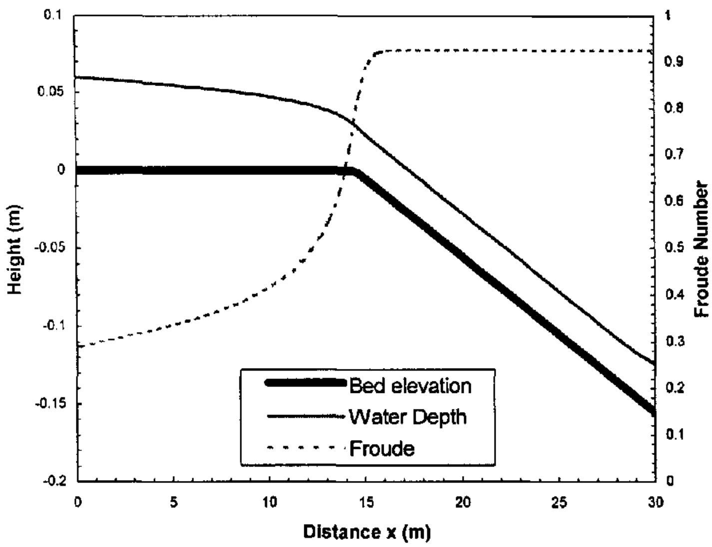

Figure 2. Numerical results for subcritical flow in terms of depth and local Froude number.

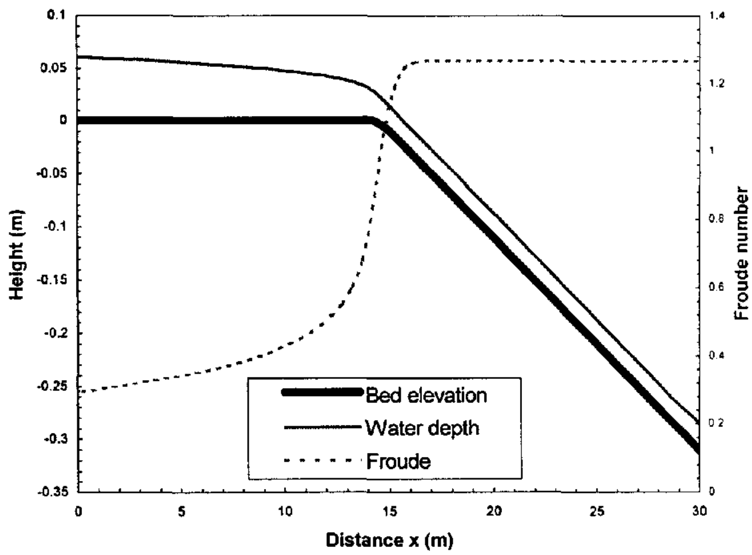

Figure 3. Numerical results for subcritical to supercritical flow.

2. Subcritical flow in a straight channel with slope change

For this problem a straight channel with two reaches with different slopes: an upstream horizontal reach, followed by a reach with a mild slope of 0.01 was selected. The first reach is 14.5 m long, and the second is 15.5 m long, both are 1.4 m wide and a value of 0.019 for the Manning's coefficient was fixed. A uniform grid of 60x10 nodal points was used to approximate the domain. The velocities at the inflow boundary were fixed as u=0.22 m/s and v=0 m/s and the inlet depth was fixed in 0.06 m which corresponds to a Froude number of 0.29, for this case subcritical conditions are enforced along the whole channel and the elliptic nature of the h' equation demands the specification of depth or mass flow at the outlet, then the depth has been fixed in 0.3 m/s.

Figure 2 shows the numerical results (at the central line of the channel) in terms of depth and local Froude number. Even though the velocity increase due to the presence of the slope change, the Froude number is always subcritical and it stays constant after the slope change as depth does. The inlet and outlet depth boundary conditions are both satisfied according to the elliptic behavior of the h′ equation.

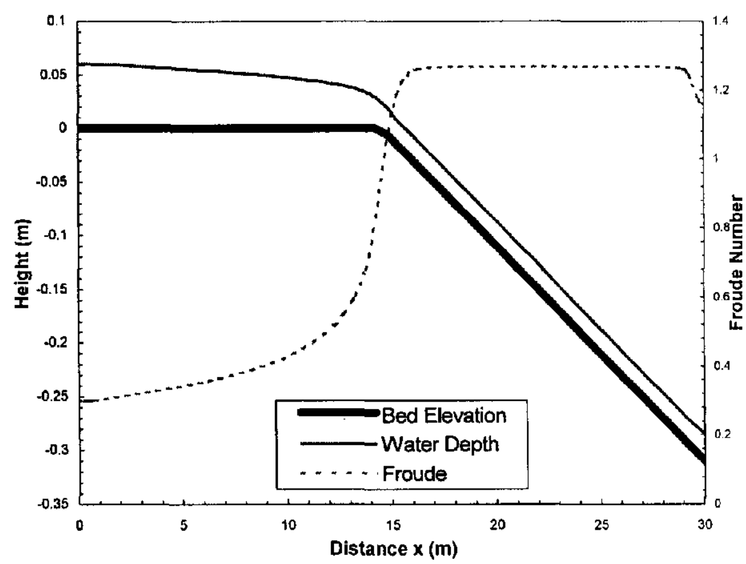

Figure 4. Numerical results for subcritical to supercritical flow fixing h at the outlet boundary

3. Changing Froude regimes in a straight channel with slope change

As in the previous case the physical situation is a straight channel with two reaches with different slopes: an upstream horizontal reach, followed this time by a reach with a steeper slope of 0.02. The first reach is 14.5 m long, and the second is 15.5 m long, both are 1.4 m wide and a value of 0.019 for the Manning's coefficient was established. Again a uniform grid of 60x10 nodal points was used to approximate the domain. The velocities at the inflow boundary were u=0.23 m/s and v=0 m/s and the inlet depth was fixed in 0.06 m which corresponds to a Froude number of 0.30, for this case subcritical conditions prevail at the horizontal portion of the channel pass through critical condition near the point of slope change and remain at supercritical condition (Fr=1.27) at the second part of the channel. Thus, the depth correction equation goes from elliptic to hyperbolic with a parabolic transition at Froude =1, with none outlet information of depth traveling to the interior of the domain as it is shown in figure 3. If the depth is enforced at the outlet boundary, that information does not affect the h' equation but it can affect slightly the momentum equations near that border as can be observed in figure 4, where a depth of 0.02 m was imposed. Again, no oscillations are observed before or after the transition.

CONCLUSIONS

An efficient extension of the pressure-based method originally aimed to solve compressible flow problems has been used for solving shallow water problems. The computational results suggest that the algorithm is very successful for this kind of problem. Comparison with experimental results shows that the method provides results with good accuracy. The use of a bounded second order scheme for the convective terms along with the deferred correction proved to be stable and efficient to improve the shock-capturing capabilities of the method, although some clipping problems were detected. The incorporated convective-like term in the depth correction equation allows capturing automatically the right mathematical behavior of the problem without the user intervention. All this results have shown that the present algorithm can solve a wide range of flow problems in river and open channels. The method may be extended to simulate physical situations with arbitrary boundary shapes using body-fitted co-ordinates or unstructured meshes.

REFERENCES

[1] Toro, E. F., "Riemann problems and the WAF method for solving the 2-dimensional shallow-water equations". Philosophical Transactions of the Royal Society of London, Series A. Physical Sciences and Engineering 1649, **338**, pp. 43-68, 1992.

[2] Sleigh, P. A., Gaskell, P. H., Berzins, M. and Wright, N. G., "An Unstructured finite-volume algorithm for predicting flow in rivers and estuaries". Computers and Fluids, 27, **4**, pp. 479-508, 1998.

[3] Abbot, M. B., *Computational Hydraulics*. Pitman publications, 1979.

[4] Cunge, J. M., Holly, F. M. and Verwey, *A. Practical aspects of computational river Hydraulics*. Pitman Publications, 1998.

[5] Patankar, S. V. and Spalding, D. B., "A calculation procedure for heat, mass and momentum transfer in three-dimensional parabolic flows". International Journal of Heat and Mass Transfer, **5** pp.17-87, 1972.

[6] Zhou, J. G., "Velocity-depth coupling in shallow water flows". Journal of Hydraulic Engineering Division, ASCE, **121**, pp. 717-724, 1995.

[7] Bisson, J., *Modelisation d'ecoulement a surface libre en deux dimensions utilisant un schema implicite*. MSc. Thesis. Universite de Montreal. Canada, 1987.

[8] Lilek, Z., *Ein Finite-Volumen verfahren zur berechnung von inkompressiblen und kompressiblen Strömungen in komplexen geometrien mit beweglichen rändern und freien oberflächen*. Dissertation, University of Hamburg, Germany, 1995.

[9] Rincon, J. and Elder, R., "A high resolution pressure-based method for compressible flow". Computers and fluids, **26**, 3, 1998.

[10] Khosla, P. K. and Rubin, S. G., "A diagonally dominant second-order accurate implicit scheme". Computers and Fluids, **2**, pp. 207-209, 1974.

[11] Versteeg, H. K. and Malalasekera, W., *An introduction to Computational Fluid Dynamics*. The Finite Volume Method. Longman, England, 1995.

[12] Ferziger, J. H. and Peric, M., *Computational Methods for Fluid Dynamics*. Springer-Verlag, Berlin Heidelberg 1996.

[13] Leonard, B. P., "Bounded higher-order upwind multidimensional finite-volume convection-diffusion algorithms". In W. J. Minkowycz, E M. Sparrow (eds.), Advances in Numerical Heat Transfer, chap. 1, pp. 1-57, Taylor and Francis, New York, 1997.

[14] Gharangik, A. M. and Chaudhry, M. H., "Numerical simulation of hydraulic jump". Journal of Hydraulic Engineering Division, ASCE, **117**, pp. 1195-1211, 1991.