Archives of Hydro-Engineering and Environmental Mechanics
Vol. 53 (2006), No. 4, pp. 295–309

Two-Dimensional Vertical Reynolds-Averaged Navier-Stokes Equations Versus One-Dimensional Saint-Venant Model for Rapidly Varied Open Channel Water Flow Modelling

Michał Szydłowski, Piotr Zima

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

(Received March 27, 2006; revised July 18, 2006)

Abstract

The paper concerns mathematical modelling of free surface open channel water flow. In order to simulate the flow two models are used – two-dimensional vertical Reynolds-Averaged Navier-Stokes equations and one-dimensional Saint-Venant equations. The former is solved with SIPMLE algorithm of finite difference method using Marker and Cell technique to trace a free surface movement. The latter is solved using the finite volume method. The dam-break (water column collapse) problem on horizontal bottom is investigated as a test case. The calculated results are compared with each other. The numerical simulations are examined against laboratory experiment presented by Koshizuka et al (1995). The possibility of using the described models to simulate rapidly varied flow is discussed.

Key words: mathematical modelling, Reynolds-Averaged Navier-Stokes equations, Saint-Venant equations, free surface flow, rapidly varied flow, dam-break problem

1. Introduction

During the last almost half century, a lot of work on mathematical modelling in fluid dynamics was carried out. The considerable progress in hardware and software development has made possible the use of mathematical models in engineering and industry. The numerical solutions of equations describing hydrodynamics processes in open channels – being a part of computational fluid dynamics (CFD) – are also intensively investigated. This development already started during the eighties (Abbott 1979, Cunge et al 1980, Granatowicz and Szymkiewicz 1989) and continues (Fletcher 1991, Anderson 1995, Szymkiewicz 2000).

The mathematical equations (representing fundamental laws of physics) that govern the phenomenon of free surface water flow, the Navier-Stokes (NS) equations, are well known. However, their solution is practically impossible (despite current computer powers) for the scale of any real case. Hence the simplified models are usually solved for numerical simulation of open channel flow. The most widely used models of one-dimensional flow are the Saint-Venant (SV) equations. They can be derived from NS equations using the spatial averaging procedure. In this paper we compare numerical solutions for rapidly varied free surface flow in open channel, computed with two-dimensional vertical Reynolds-Averaged NS (RANS) equations and one-dimensional SV model. The analysis will be useful to assess whether the SV equations constitute an adequate model of free surface water flow near some local effects (i.e. steep wave fronts, hydraulic jumps, etc.) or not and answer the question as to whether they can be used for simulation of typical water engineering problems.

2. Mathematical Models of Free Surface Water Flow

2.1. Navier-Stokes Equations

In general, the open channel water flow is a three dimensional time dependent, incompressible, fluid dynamics problem with a free surface. The well known NS equations (together with continuity equation) describe the dynamics of a portion of any fluid (Sawicki 1998). However, the water flow in natural and artificial channels and reservoirs is usually turbulent. In order to eliminate the problem of turbulence the NS equations can be averaged in time to obtain RANS equations that describe the mean flow (Sawicki 1998). Then the effects of the turbulent fluctuations on the mean flow can be imposed applying some turbulence models. The three dimensional RANS model is often used in technical fluid mechanics, but is still too complex to be applied to describe open channel flow for practical cases. Many numerical techniques for solving RANS equations have been successfully applied so far (Fletcher 1991, Anderson 1995). They are usually based on the finite difference method (FDM), finite element method (FEM) or finite volume method (FVM). One of the most popular and often used technique is a SIMPLE scheme (Semi Implicit Method for Pressure Linked Equations) of FDM (Potter 1982). In this algorithm a divergence of RANS equations is considered. It leads to the Poisson equation which describes pressure field evolution (pressure-correction equation). In the study presented in this paper the SIMPLE algorithm is used to solve two-dimensional vertical RANS equations on a rectangular, staggered grid.

If RANS model is used to describe the open channel flow the free surface movement problem must be solved. The free surface moves with the velocity of the fluid particles located at the boundary, and therefore its position must be found during computation. One of the techniques to solve this problem is the Marker and Cell (MAC) method (Welch et al 1966). The application of this method to solve open channel flow is possible, but unique, due to huge computational power needed to simulate some real cases. Therefore, the solutions are often limited to two dimensional (in the vertical plane) test cases (Maronnier et al 1999, Mohapatra et al 1999, Zima 2005, Zwart et al 1999).

2.2. Saint-Venant Equations

Two-dimensional shallow water (SW) equations can be obtained from the RANS model using a depth averaging procedure (Szymkiewicz 2000). This process eliminates the free surface location problem from the solution. For the equations derivation, it is assumed that the vertical component of velocity can be neglected, pressure field is hydrostatic, bottom slope is small and bottom friction can be approximated as for steady flow conditions. Therefore, SW equations are not a true mathematical representation of the free surface water flow, but they reduce one of the spatial dimensions from the problem. Finally, two-dimensional SW equations can be reduced to the SV model considering only one-dimensional water flow in the channel. In general, Saint-Venant assumptions are not satisfied for rapidly varied flow in an open channel and this model seems to be a poor representation of real phenomena of the flow near local hydraulic effects. In this paper we present the comparison between numerical solution to RANS and SV models for rapidly varied flow in an open channel during the dam-break problem (collapse of water column) on a horizontal bottom. Unfortunately, the FDM and FEM numerical methods for SV model solution are often inefficient for modelling rapidly varied flow, when discontinuities like hydraulic jumps or steep fronts exist. In order to ensure the proper solution to SV equations the FVM is applied (Szydłowski 2004). The calculated results analysis illustrates the influence of RANS equations spatial averaging procedure leading to the SV model on the quality of numerical simulation of rapidly varied free surface water flow.

3. RANS Equations and Solution Method

The governing equations for incompressible viscous flow are the continuity equation (1) and RANS equations (2), which can be written in the following form (Sawicki 1998):

$$\nabla \mathbf{u} = 0, \tag{1}$$

$$\frac{D\mathbf{u}}{Dt} = \mathbf{f} - \frac{1}{\rho}\nabla p + \nu\Delta\mathbf{u}, \tag{2}$$

where:

- $\mathbf{u}$ – velocity vector,
- $\mathbf{f}$ – vector of external forces,
- p – pressure,
- ρ – density,
- ν – kinetic turbulent viscosity factor.

If the divergence operator is applied, equation (2) can be rewritten as pressure correction equation (Potter 1982):

$$\Delta \bar{p} = -\nabla \cdot (\mathbf{u} \cdot \nabla \mathbf{u}), \tag{3}$$

where $\bar{p}$ denotes normalized pressure expressed as $\bar{p} = p/\rho$.

In the vertical Cartesian plane x-y (neglecting one of the horizontal dimensions) RANS equations are solved only for two velocity components (horizontal u_x in x-direction and vertical u_y in y-direction) and for the pressure p. The external body forces vector includes the acceleration due to gravity g with components ρg. The equations (1, 2, 3) can be rewritten in differential form for both velocity components as follows:

$$\frac{\partial u_x}{\partial x} + \frac{\partial u_y}{\partial y} = 0, \tag{4}$$

$$\begin{cases} \dfrac{\partial u_x}{\partial t} + u_x \dfrac{\partial u_x}{\partial x} + u_y \dfrac{\partial u_x}{\partial y} = g_x - \dfrac{1}{\rho}\dfrac{\partial p}{\partial x} + \nu \left(\dfrac{\partial^2 u_x}{\partial x^2} + \dfrac{\partial^2 u_x}{\partial y^2} \right), \\ \dfrac{\partial u_y}{\partial t} + u_x \dfrac{\partial u_y}{\partial x} + u_y \dfrac{\partial u_y}{\partial y} = g_y - \dfrac{1}{\rho}\dfrac{\partial p}{\partial y} + \nu \left(\dfrac{\partial^2 u_y}{\partial x^2} + \dfrac{\partial^2 u_y}{\partial y^2} \right), \end{cases} \tag{5}$$

$$\frac{\partial^2 \bar{p}}{\partial x^2} + \frac{\partial^2 \bar{p}}{\partial y^2} = -\left\{ \left(\frac{\partial u_x}{\partial x} \right)^2 + 2 \left(\frac{\partial u_x}{\partial y} \right) \left(\frac{\partial u_y}{\partial x} \right) + \left(\frac{\partial u_y}{\partial y} \right)^2 \right\}. \tag{6}$$

Equations (5, 6) are solved using a SIMPLE algorithm of FDM. The main idea of this method is applying the splitting technique to the solution. The first step is a prediction of the velocity field integrating equations (5). The explicit scheme is used to obtain values of velocity components with values of pressure from previous time step. In the second step, the correction of pressure field is computed using equation (6). Then, it is used to correct the velocity field to satisfy the zero divergence condition (4).

In order to determine the free surface location the flow domain is defined using the MAC method (Welch et al 1966).

To integrate the RANS equations system (5) and Poisson equation (6) in space by FDM the two-dimensional domain x-y should be discretized into a set of computational cells. In order to make this partition the Euler staggered grid was used. In each cell variables u_x, u_y and $\bar{p}$ are located in different places (Fig. 1). There are four types of cells (Fig. 2): full (F), boundary (B), surface (S) and empty (E) depending on location of the cells and fluid inside the computational domain.

The velocity components are specified at the cell-interfaces while the pressure is specified at the cell center-point. For RANS equations (5) approximation on a staggered grid, the Lax-Wendroff scheme was applied (Potter 1982). This approach

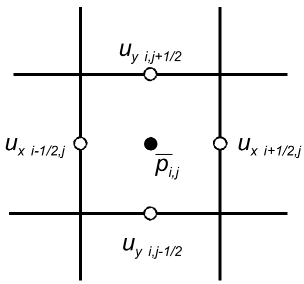


Fig. 1. Cell specification for Euler staggered grid

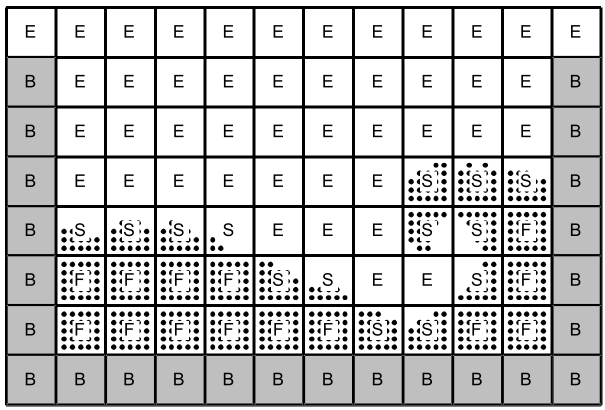


Fig. 2. Discretization of two-dimensional calculation domain (notations for cells in the MAC method)

ensures second order accuracy of the numerical scheme. The pressure-correction equation (6) is solved using the Successive Over-Relaxation (SOR) method (Remson et al 1971). The boundary conditions for these equations and their solution stability conditions must be satisfied. The initial velocity field **u**, pressure $\bar{p}$ and the initial domain fill (initial position of the free surface) are specified as the initial conditions. The velocity terms are explicit, computed using known values, but the pressure term is implicit, based on the unknown pressure values at the next time step. The position of the markers in cells for the new time level is computed using the corrected velocity **u** and Newton's second law (Potter 1982).

4. SV Equations and Solution Method

The unsteady one-dimensional open channel flow is usually described using Saint-Venant equations. In original (non-conservative) form the model can be written as follows (Szymkiewicz 2000):

$$\frac{\partial h}{\partial t} + \frac{\partial}{\partial x}(uh) = 0, \tag{7a}$$

$$\frac{\partial u}{\partial t} + u\frac{\partial u}{\partial x} + g\frac{\partial h}{\partial x} = g\left(S_0 - S_f\right), \tag{7b}$$

where u is the flow mean velocity (depth averaged velocity component in x-direction – u_x), h water depth, g acceleration due to gravity, x and t represent distance and time and S_0 and S_f denote bed and friction slopes, respectively. The friction slope can be defined by Manning's formula, which for the rectangular channel of unit width has the form:

$$S_f = \frac{n^2 u\,|u|}{h^{4/3}}. \tag{8}$$

The model (7a, b) describes gradually varied, one-dimensional free surface flow. Unfortunately, this non-conservative form of equations is inadequate when hydraulic jumps or steep water wave fronts can appear. It was proved (Abbott 1979) that the water flow with discontinuities can be properly described using a conservative form of SV model only. This can be written in vector form as follows (Cunge et al 1980)

$$\frac{\partial \mathbf{U}}{\partial t} + \frac{\partial \mathbf{F}}{\partial x} + \mathbf{S} = 0, \tag{9}$$

where:

$$\mathbf{U} = \begin{pmatrix} h \\ uh \end{pmatrix}, \quad \mathbf{F} = \begin{pmatrix} uh \\ u^2h + 0.5gh^2 \end{pmatrix}, \quad \mathbf{S} = \begin{pmatrix} 0 \\ -gh\left(S_0 - S_f\right) \end{pmatrix}. \tag{10a, b, c}$$

Equation (10) can be rewritten in equivalent form as

$$\frac{\partial \mathbf{U}}{\partial t} + \operatorname{div}\mathbf{F} + \mathbf{S} = 0. \tag{11}$$

In order to integrate the equations system (11) in space the FVM was chosen (LeVeque 2002). Applying this method one-dimensional domain x must be discretized into the set of line segment cells (Fig. 3). Each cell is defined by its centre-point and each flow parameter is averaged inside the cell.

$$\frac{\partial \mathbf{U}_i}{\partial t}\Delta x_i + \left(\mathbf{F}_{i+1/2} - \mathbf{F}_{i-1/2}\right) + \left(\mathbf{S}_{i+1/2} + \mathbf{S}_{i-1/2}\right)\Delta x_i = 0. \tag{12}$$

In order to calculate the fluxes $\mathbf{F}$ at cell interfaces the Roe (1981) scheme is used. Detailed description of the method is available in the literature (Toro 1997, Szydłowski 2004, Zoppou and Roberts 2003) therefore it is omitted here. The source terms $\mathbf{S}$ are approximated using the method proposed by Bermudez and Vazquez

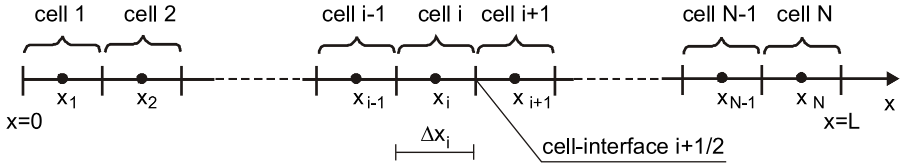


Fig. 3. Discretization of one-dimensional calculation domain

(1994). The numerical algorithm is completed with a two-step explicit scheme of FDM for integration in time. This scheme is of second-order accuracy in time and its stability is restricted by Courant number (Potter 1982).

5. Numerical Calculations and Results Discussion

The mathematical models described in points 2 and 3 of this paper have been used to simulate a CFD standard problem – dam-break flow test on the horizontal bottom. The problem is equivalent to the water column collapse effect. The flow on dry bottom downstream of the 'dam' was considered. Numerical results obtained with both models were compared. Moreover, the results of a laboratory experiment carried out by Koshizuka et al (1995) were used to examine the calculations. It is well known that the SV model is not a true mathematical representation of the free surface water flow. However, shallow water models are commonly used for simulating the dam-break flows, assuming that the vertical velocities and non-hydrostatic pressure distribution do not affect the long term results (Morris 2000, Szydłowski 2003). In this analysis, we compare the numerical results for the free surface profiles and the front velocity obtained using SV and RANS models at the beginning of the water flow process – immediately after water column collapse.

Geometry of domain and initial position of water column is presented in Figure 4. In the laboratory experiment, a glass box with the scale $L = 14.6$ cm was used. The water column was supported by the vertical wall, which was drawn up rapidly (about 0.05 s) for the beginning of collapse. The experiment was recorded using a video camera. Some pictures of water body during the experiment are presented in Figure 5. Before the 'dam-break' the water body was at rest. After sudden water release, two effects were observed – depression of water table travelling upstream and steep water front going downstream of the initial water surface discontinuity. After reaching the box wall, the wave moving downstream reflects there, forming a sudden surface swelling going upstream. Moreover, a water splash effect can be observed near the wall.

The numerical simulation with RANS model with the following parameters was carried out: number of cells 1891 ($\Delta x = \Delta y = 0.973$ cm), number of particles $n = 800$. Gravitation unit $g_y = -9.806$ ms^{-2} and turbulent kinetic viscosity factor $\nu = 10^{-3}$ m^{2}s^{-1} were imposed. However, viscous effect on the bottom and walls of the channel was neglected – the slip boundary condition was imposed there.

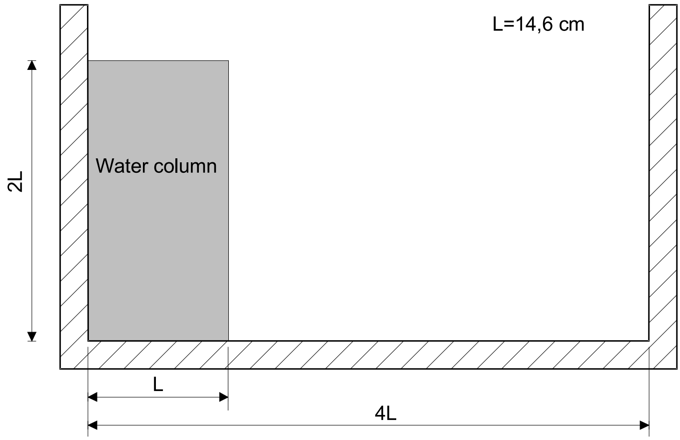


Fig. 4. Geometry of domain and initial shape of water column

Numerical simulation with SV model was carried out using a mesh composed of 1169 computational cells (Δx = 0.0005 m). Time integration was done using a two-step explicit scheme with time step Δt = 0.0001 s ensuring solution stability. In the first simulation a bottom friction was neglected. The slip boundary condition on the channel bottom for RANS equations and Manning coefficient equal to zero in SV model are equivalent to the frictionless open channel flow case. This assumption makes computed results possible to compare with others. Moreover, this simplification is not far from the test case, where the flow over a glass surface of small friction is considered.

The free surface profiles obtained using an SV model for different times is shown in Fig. 5 as solid lines. The results of computing with RANS model (particles locations only) for the example calculation are presented in the same Figure as the circle marks.

In order to analyse the wave front propagation problem the flow was considered until the front have touched the vertical box wall. The position of the water front is overestimated by the SV model, as compared with the numerical solution to RANS equations. It was observed that the wave front approximated with SV model is about two times faster than that computed with RANS model or an observed one. This effect can also be seen watching the shapes of water table computed with SV model for times equal to half the simulation time (dotted line in Fig. 5). The water level for these moments fits the RANS computations and observations quite well.

The front location discrepancy is a result of difference of wave front propagation velocity predicted using SV and RANS equations. The distance travelled by the front as a function of time for numerical simulations and experiments is shown in Fig. 6.

The measurements and computed results in non-dimensional relation x/L are shown on the normalized time ($t_n = t\sqrt{2g/L}$) background. The slope of the curve defines the velocity of the wave front. The significant disagreement between SV

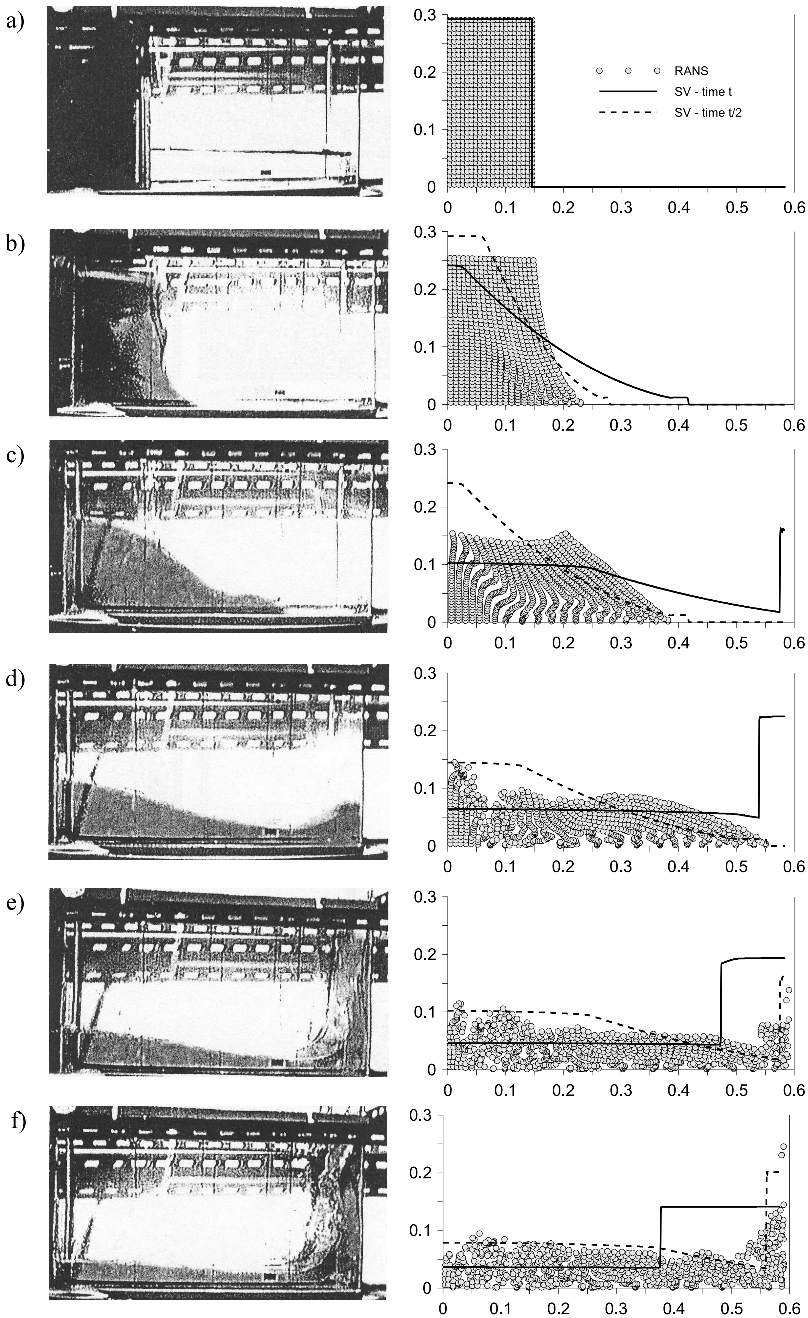


Fig. 5. Observed by Koshizuka et al (1995) and calculated with RANS and SV models shapes of water body after a) $t = 0.0$ s, b) $t = 0.1$ s, c) $t = 0.2$ s, d) $t = 0.3$ s, e) $t = 0.4$ s, f) $t = 0.5$ s of simulation

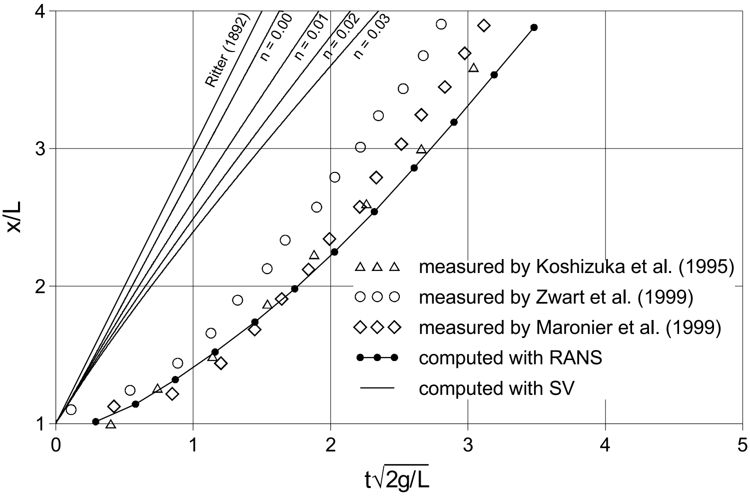


Fig. 6. Wave front position – experimental and calculated results

model results and RANS equations is observed. Moreover, it can be observed that the numerical solution to RANS model matches quite well with the laboratory measurements. It can be seen that velocity of the wave front predicted with numerical solution to SV equations is overestimated in comparison with experiment and solution to two-dimensional RANS equations. It is found here that wave front simulated with SV model moves on a horizontal and frictionless bottom with the averaged non-dimensional velocity of about 1.8. This value is close to analytical solution to SV equations for dry bottom conditions proposed by Ritter (1892), where the wave front non-dimensional speed is equal to 2. This difference has occurred because the dry bottom in analysed numerical simulation is substituted with the zone covered by the thin water film (10^{-6} m) while the Ritter solution is obtained for a depth equal to zero. However, it can be seen (Fig. 6) that measured and simulated with RANS model the wave front just after a water column collapse propagates radically slower with averaged non-dimensional velocity of about 0.96. This is consistent with experimental results presented by Maronnier et al (1999) and Zwart et al (1999) who have reported the measurements where the averaged non-dimensional wave front speed (for similar dam break problems) was about 1.1.

Analyzing the wave front velocity obtained with SV and RANS models (Fig. 6) it can be seen that the front velocity approximated using the former is constant (as long as the depth upstream the dam remains equal to the initial value) while the latter gives significant front speed variation. Just after the water release, the front is slower than that computed with SV equations and then an acceleration can be observed (in accordance with experiments). Finally – after a short time from the beginning of water flow – the front velocity is fixed (the slope of distance-time curve becomes constant).

On the other hand it can be seen (Fig. 6) that if the bottom friction (defined as the Manning coefficient) is added to SV model solution (the only one possible way to consider the water viscosity and flow turbulence in this model) it will produce the front velocity close to the speed computed with RANS model for the period after a short-term effect of front acceleration. In Figure 6 it can be observed as the comparable slopes of distance-time curves for both models.

The discrepancy of computations with RANS and SV models is a result of different representation of pressure and velocity distribution along depth in both models. In one-dimensional averaged SV equations the pressure field is always hydrostatic and velocity is uniform along depth. However, it can be observed that during the first short-term period after water release, the pressure is non-linear and the horizontal velocity component is not constant along depth at the dam (discontinuity) cross-section ($x = L$ in Fig. 4). It is well presented in Figure 7 where the horizontal velocity and pressure distributions for the first moments after water release, computed with RANS model are presented. After this short-term effect the pressure and velocity become almost hydrostatic and uniform, respectively. It can be observed that at other cross-sections ($x = 2L$ and $x = 3L$) the pressure and velocity variation along depth is rather slight and decays with simulation time.

The evolution of pressure on the bottom at the dam cross-section is presented in Figure 8. The pressure computed with RANS model is equal to zero immediately after the dam-break because of free surface conditions at this location, while the water depth is defined there as an initial upstream surface level. Then the pressure increases as the water begins flowing. It can be seen that the computed pressure is not equal to the hydrostatic pressure (equivalent to water depth) at the beginning of the flow. However, the computed and hydrostatic pressure approach each other a short time after water column collapse. Moreover, it can be pointed out that for other cross-sections the evolution of pressure and depth fit well, which means that non-linear pressure distribution is not significant at these locations.

The results presented in the paper confirm that the non-linear pressure distributions and non-uniform horizontal velocity variation near the flow discontinuity are the short-term effects and they are decaying with simulation time increase. It suggests that if these short-term effects are not essential for a simulated flow problem they can be omitted causing no damage to long-term simulation. It proves that SV equations can be used to describe the rapidly varied flow for the most practical water engineering problems where the local effects occur, but their inner structure can be omitted.

If the short-time flow simulation is indispensable or some special local hydraulic effects like water splashing must be considered, the RANS type models should be applied.

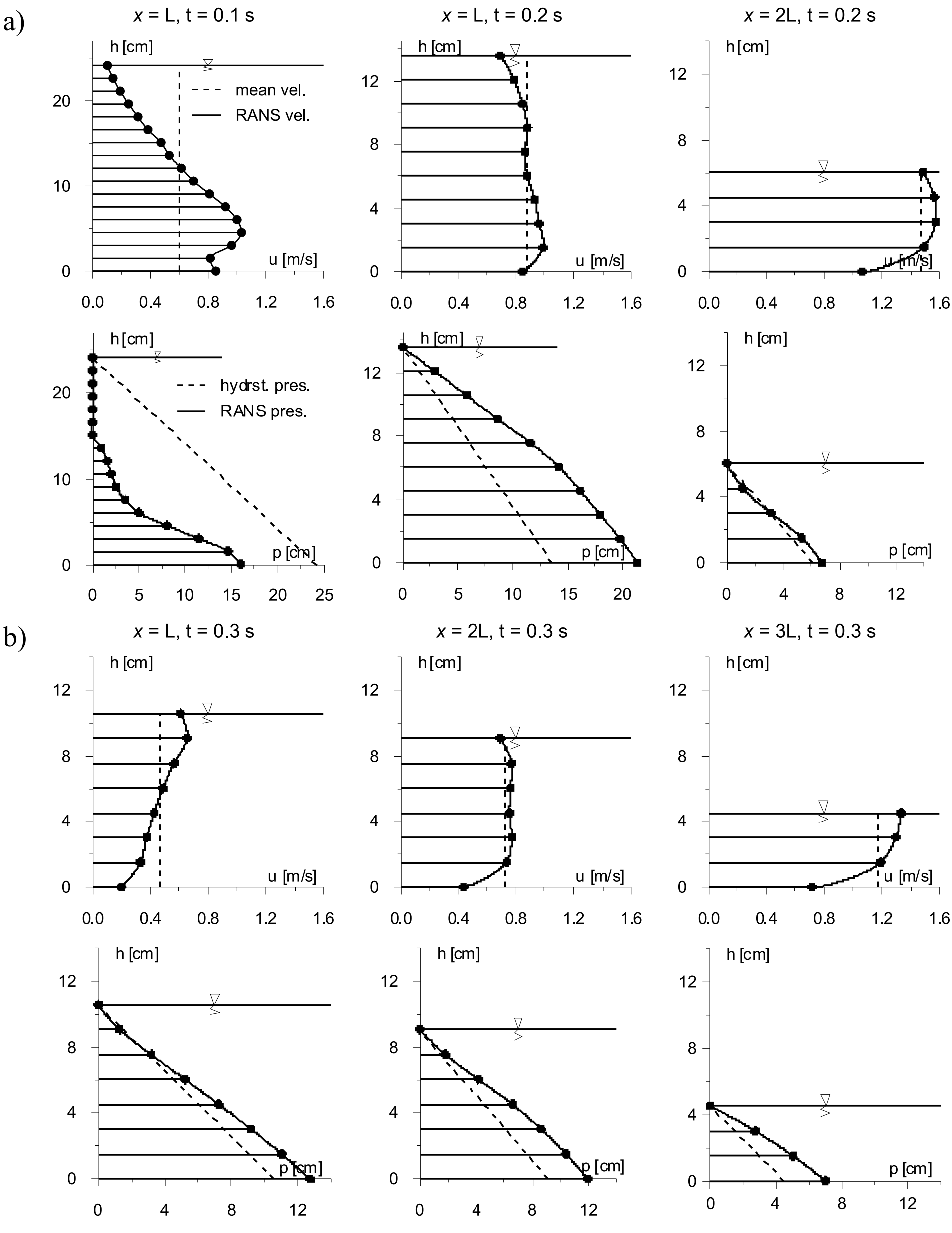
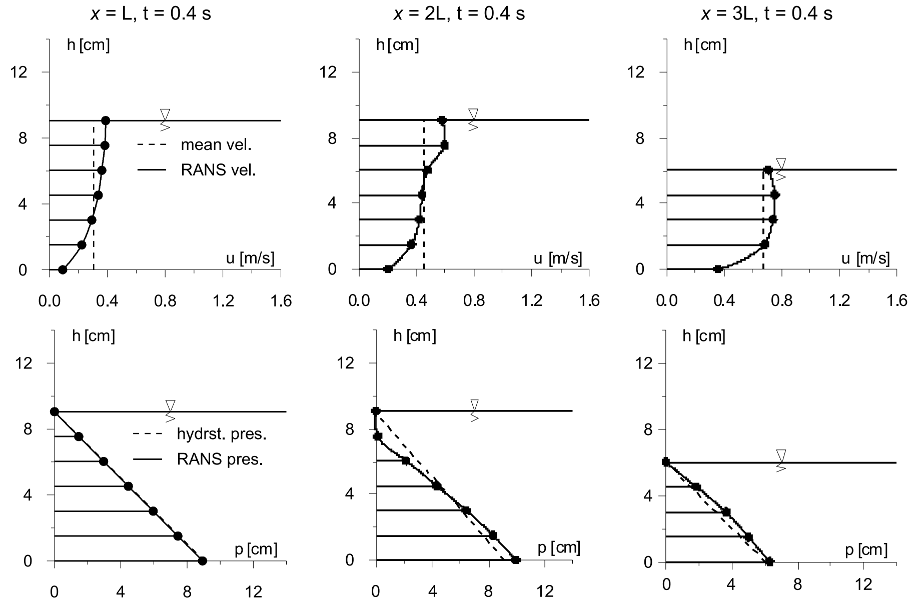


Fig. 7. Horizontal velocity and pressure distribution along depth computed with RANS model at some cross-sections for a) $t = 0.1$ s and $t = 0.2$ s, b) $t = 0.3$ s, c) $t = 0.4$ s, d) $t = 0.5$ s

c)



d)

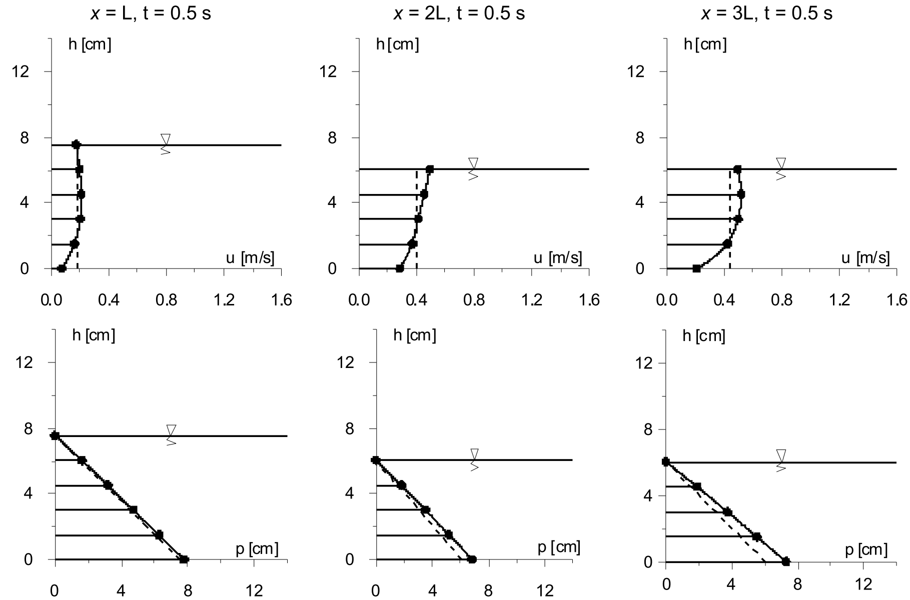


Fig. 7. (continued) Horizontal velocity and pressure distribution along depth computed with RANS model at some cross-sections for a) t = 0.1 s and t = 0.2 s, b) t = 0.3 s, c) t = 0.4 s, d) t = 0.5 s

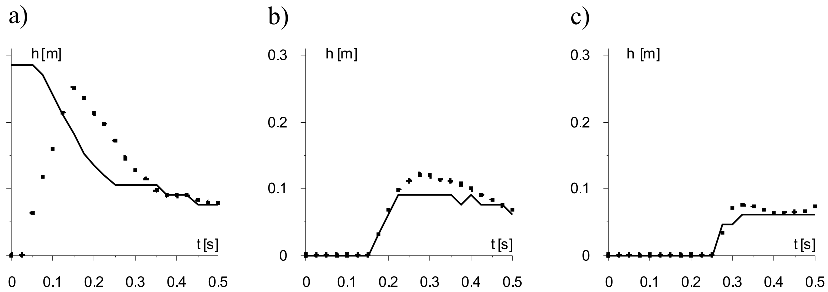


Fig. 8. Pressure (•) and depth (–) evolution at a) $x = L$ (dam cross-section), b) $x = 2L$, c) $x = 3L$

6. Summary and Conclusions

In the paper, rapidly varied open channel flow due to dam-break effect was analysed. The analysis was based on experimental (Koshizuka et al 1995) and numerical investigation of the water column collapse problem. In order to calculate the evolution of flow parameters two mathematical models were used – RANS and SV equations.

Concluding, it should be pointed out that SV model (assuming hydrostatic pressure and uniform velocity distribution along water depth) is not an adequate description of flow near the local hydraulic effects (i.e. steep wave fronts, hydraulic jumps, etc.). The better simulation results for short-term problems can be ensured using RANS equations. Unfortunately, this model is too complex (even in the two-dimensional vertical plane) to be applied for practical, free surface water engineering problems. Therefore, SV equations still remain the main mathematical model of open channel rapidly varied flow ensuring sufficiently good simulation results for long-term typical engineering application problems where inner structure of local hydraulic effects and short-term pressure and velocity distribution evolution can be neglected. However, if the detailed description of flow parameters of local hydraulic effect is necessary, the RANS type models must be used.

Acknowledgment

The authors wish to acknowledge the financial support offered by Polish Ministry of Education and Science for the research project 2 P06S 034 29.

References

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

Anderson J. D. (1995), *Computational Fluid Dynamics*, McGraw-Hill Inc., New York.

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

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

Fletcher C. A. J. (1991), *Computational Techniques for Fluid Dynamics 1. Fundamental and General Techniques*, Springer-Verlag, Berlin.

Granatowicz J., Szymkiewicz R. (1989), A comparison of the solution effectiveness of the Saint-Venant equations with finite element method and finite difference method, *Archives of Hydro-Engineering and Environmental Mechanics*, Vol. 36, No. 3–4, 199–210 (in Polish).

Koshizuka S., Tamako H., Oka Y. (1995), A Particle Method for Incompressible Viscous Flow with Fluid Fragmentation, *Computational Fluid Dynamics Journal*, Vol. 4, No. 1, 29–46.

LeVeque R. J. (2002), *Finite Volume Method for Hyperbolic Problems*, Cambridge University Press, New York.

Maronnier V., Picasso M., Rappaz J. (1999), Numerical simulation of free surface flows, *Journal of Computational Physics*, Vol. 155, 439–455.

Mohapatra P. K., Eswaran V., Bhallamudi S. M. (1999), Two-dimensional analysis of dam-break flow in vertical plane, *Journal of Hydraulic Engineering*, Vol. 25, No. 2, 183–192.

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

Potter D. (1982), *Computational Physics*, PWN, Warsaw (in Polish).

Remson I., Hornberger G., Molz F. (1971), *Numerical Methods in Subsurface Hydrology*, Wiley and Sons, New York.

Ritter A. (1892), Die Fortplanzung der Wasserwellen, *Zeitschrift des Vereines Deutscher Ingeniere*, 33(36), 947–954.

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

Sawicki J. (1998), *Free Surface Flows*, PWN, Warsaw (in Polish).

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

Szydłowski M. (2004), Implicit versus explicit finite volume schemes for extreme, free surface water flow modelling, *Archives of Hydro-Engineering and Environmental Mechanics*, Vol. 51, No. 3, 287–303.

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

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

Welch J. E., Harlow F. H., Shannon J. P., Daly B. J. (1966), *The MAC Method*, Technical Report La-3425, Los Alamos National Laboratory.

Zima P. (2005), The numerical simulation of two-dimensional vertical incompressible viscous flow, [in:] *Proceedings of Water Management and Hydraulic Engineering Ninth International Symposium,* Ottenstein-Austria, Sept. 4–7 2005, eds. H. P. Nachtnebel, C. J. Jugovic, Vienna, BOKU, Univ. Natural Resour. a. Appl. Sci., 455–462.

Zoppou C., Roberts S. (2003), Explicit schemes for dam-break simulations, *Journal of Hydraulic Engineering*, Vol. 129, No. 1, 11–34.

Zwart P. J., Raithby G. D., Raw M. J. (1999), The integrated space-time finite volume method and its application to moving boundary problems, *Journal of Computational Physics*, Vol. 154, 497–519.