ISSN 1392 - 1207. MECHANIKA. 2005. Nr.1(51)

Dam break flow simulation by the pseudo-concentration method

A. Kačeniauskas
Vilnius Gediminas Technical University, Saulėtekio 11, LT-10223 Vilnius, Lithuania, E-mail: arnka@fm.vtu.lt

1. Introduction

Dam break flow has been the subject of extensive research for a long time [1, 2]. The original problem has direct application in the industrial areas of fluid mechanics and environment protection. The breaking wave phenomena occurring in some cases of a dam break problem includes it into the class of complex applications such as solitary wave propagation, tank sloshing and water on a ship deck simulation. Some experimental measurements were performed on the dam break flow or collapse of a liquid column problem [2, 3]. Photographs showing the time evolution of the collapsing column as well as the wave returning after hitting a wall on the opposite side are available for the purpose of evaluating the numerical methodology on the basis of flow visualisation. Measurements of the exact interface shape are not available, but some secondary data such as the reduction of the water column height [3] can be employed for quantitative comparison of the obtained results. Several modifications of the broken dam problem have been extensively used as a classical test cases for numerical simulation of free surfaces and moving interfaces [4-6]. However, the universal, accurate and efficient numerical technique for breaking wave simulation attracts big attention of research community and software developers.

There are a lot of numerical methods advocated for solving moving interface problems. They might be classified into two categories, depending on their description of interface motion: interface tracking techniques (hereinafter ITT) and interface capturing techniques (hereinafter ICT). In the first category of interface simulating methods a moving interface is represented and tracked explicitly either by making it with special marker points, or by attaching it to a mesh surface, which is forced to move with the interface. The earliest work [7] was based on the Lagrangian description of motion. The mesh deforms severely as a free surface moves, making remeshing and rezoning necessary at each time step [8]. In the Arbitrary Lagrange-Eulerian approach [9] a mesh deforms in terms of an arbitrary velocity field, which is independent on the flow velocities, except at the moving interface. Various surface fitted methods [10, 11] for attaching the interface to the mesh surface were developed during the past decades using the finite element method (hereinafter FEM). Surface fitted methods [10, 11] use moving unstructured meshes and allow employing the full power and flexibility of the FEM. These methods are unable to cope naturally with interface interacting with itself by folding or rupturing. Only at a cost of complex implementation they simulate the discussed phenomena.

In the second category of interface simulating methods either massless particles or an indicator function marks gas or fluid on either side of the interface. The marker-and-cell method [5], the volume of fluid method (hereinafter VOF) [6] and the level set method (hereinafter LSM) [12] are well known methods using interface capturing idea and the Eulerian approach. The interface capturing methods require no geometry manipulations after the mesh is generated and can be applied to interfaces of a complex topology. The VOF methods are very efficient and practical [13], therefore, they are implemented in a lot of commercial codes using the finite volume method. The LSM introduced by Osher and Sethian [12] is based on finite difference schemes. The mathematical model of the LSM is very universal. This method automatically takes care of merging and breaking of the interface, but the numerical implementation of reinitialization procedures is quite complicated and requires large computational resources [14]. The first publications presenting attempts to combine the level set method and finite elements appeared recently [14-16].

Another interesting alternative for interface capturing is referred to as pseudo - concentration method (hereinafter PCM) [17], often used with the FEM. The method uses a pseudo - concentration function defined in the entire domain and solves directly a hyperbolic equation to determine the moving interface. The choice of function features depends on different numerical schemes employed in the solution procedure by different authors [18, 19]. Sometimes pseudo - concentration function is chosen to be very close to volume fraction function widely used in the VOF method [20] and finite volumes. In the most cases the PCM is more efficient than the LSM, because it uses simpler front reconstruction techniques. However, the choice of the numerical schema and the pseudo - concentration function remains state of the art problem.

2. Mathematical model of the flow

The laminar and Newtonian flow of viscous and incompressible fluids is described by the Navier-Stokes equations in the Eulerian reference frame

$$\rho\left(\frac{\partial u_i}{\partial t}+u_j\frac{\partial u_i}{\partial x_j}\right)=\rho F_i+\frac{\partial \sigma_{ij}}{\partial x_j} \tag{1}$$

$$\frac{\partial u_i}{\partial x_i}=0 \tag{2}$$

where u_i are the velocity components; ρ is the density; F_i are the gravity force components and σ_{ij} is stress tensor

$$\sigma_{ij}=-p\delta_{ij}+\mu\left(\frac{\partial u_i}{\partial x_j}+\frac{\partial u_j}{\partial x_i}\right) \tag{3}$$

here μ is dynamic viscosity coefficient; p is pressure and δ_{ij} is Kronecker delta. Slip boundary conditions for velocity are prescribed on rigid walls

$$u_i n_i = 0 \quad (4)$$

here n_i are components of a unit normal vector. This is usual choice of boundary conditions used for modelling of moving interface flows. The exception is made for the upper wall. No-slip boundary conditions and pressure

$$u_i = 0, p = 0 \quad (5)$$

are prescribed on the upper wall for modelling of breaking wave phenomena. The zero stress boundary conditions are prescribed on the open upper boundary

$$\sigma_{ij} n_j = 0 \quad (6)$$

The reference pressure is prescribed on the upper wall. The zero initial conditions are prescribed for the Eqs. (1)-(3) in the performed investigation.

The pseudo-concentration method [17] is developed for moving interface flows using the Eulerian approach and the interface capturing idea. The pseudo-concentration function φ serves as a marker identifying fluids A and B with densities ρ_A and ρ_B and viscosities μ_A and μ_B. In this context, the density and viscosity are defined as

$$\rho = \varphi\rho_A + (1-\varphi)\rho_B \quad (7)$$

$$\mu = \varphi\mu_A + (1-\varphi)\mu_B \quad (8)$$

while φ=1 for fluid A and φ=0 for fluid B. The evolution of the interface is governed by a time dependent convection equation

$$\frac{\partial\varphi}{\partial t} + u_j \frac{\partial\varphi}{\partial x_j} = 0 \quad (9)$$

The velocity u_j is obtained from the solution of the Navier-Stokes Eqs. (1, 2). The initial conditions defined on the entire solution domain should be prescribed for the Eq. (9).

3. Finite element formulation

In this paper, the solution domain is discretized by quadrilateral finite elements. Equal order bilinear shape functions are used for both the pressure and velocity components as well as for pseudo-concentration function. The space-time GLS finite element method is applied for the stabilization of Navier-Stokes Eqs. (1-2) as a general-purpose computational approach to solve a wide variety of incompressible flow problems [18]. The stabilization nature of the formulation prevents numerical oscillation for incompressible flows using equal-order interpolation functions for velocity and pressure and preserves the consistency of the standard Galerkin method when adapting remeshing is performed. The details of the applied weak formulation can be found in the reference [21].

The standard Galerkin method yields oscillatory solutions when it is applied to hyperbolic convection Eq. (9) in conjunction with classical time-stepping algorithms. The Galerkin Least Squares Method (GLS) [22, 23] belongs to the family of the stabilized methods based on adding a stabilization term to the Galerkin method. This stabilization term is the least square form of the residual of the equation evaluated elementwise and multiplied by a stabilization parameter. GLS method is naturally used together with space-time approach and the temporal derivatives are treated like the first spatial derivatives

$$\int_{Q_n} \boldsymbol{N}\left(\frac{\partial \boldsymbol{N}^T}{\partial t} + u_j \frac{\partial \boldsymbol{N}^T}{\partial x_j}\right) dQ\, \Phi + \sum_{e=1}^{N} \tau_e \int_{Q_n^e} \left(\frac{\partial \boldsymbol{N}}{\partial t} + u_i \frac{\partial \boldsymbol{N}}{\partial x_i}\right)\left(\frac{\partial \boldsymbol{N}^T}{\partial t} + u_j \frac{\partial \boldsymbol{N}^T}{\partial x_j}\right) dQ\, \Phi = 0 \quad (10)$$

The time derivatives are computed using central weighted space-time finite elements in every space-time slab Q_n. The stabilization parameter τ_e is calculated in every finite element Q_n^e

$$\tau_e = \frac{h_e}{2|u|_e} \quad (11)$$

here h_e is a measure of the finite element length; $|u|_e$ is a length of velocity vector in finite element e (e=1, …, N). The implicit numerical schema (10, 11) is unconditionally stable. The resulting coefficient matrices are unsymmetrical in spite of the symmetry of the stabilising term.

While the Eq. (10) moves the interface at a correct velocity, the pseudo-concentration function may become irregular after some period of time. In this work, the simple interface reconstruction technique is implemented

$$\varphi = min[max[\varphi, 0], 1] \quad (12)$$

It removes the overshoots and prevents the field from undesirable numerical oscillations.

4. Numerical results and discussions

The broken dam problem was modelled by the PCM implemented in the FEMTOOL software [21]. The geometry of solution domain is shown in Fig. 1. The dimensions of the reservoir and the water column (Fig. 1, a) correspond with those used in the experiment carried out by Koshizuka *et. al.* [3] The reservoir is made of glass, with a base length of 0.584 m. The water column, with a base length of 0.146 m and a height of 0.292 m (a=0.146 m), is initially supported on the right by a vertical plate drawn up rapidly at time t=0.0 s. The water falls under the influence of gravity (g=9.81 m/s^2) acting vertically downwards. The density of water ρ_A=1000 kg/m^3, the dynamic viscosity coefficient μ_A=0.01 kg/(m s). The density of air is taken to be ρ_B=1 kg/m^3, the dynamic viscosity coefficient μ_B=0.0001 kg/(m s). The slip boundary conditions (4) were applied on the bottom and sides of the reservoir. The stress boundary conditions (6) were prescribed on the upper open boundary. They may be changed to fixed pressure and zero normal gradients of the velocities.
The gravity causes the water column in the left of the reservoir to seek the lowest possible level of potential energy. Thus, the column will collapse and eventually come to rest. The initial stages of the flow are dominated by inertia forces with viscous effects increasing as the water comes

to rest. On such a large scale, the effect of surface tension forces is unimportant. The numerical results were validated by the quantitative comparison with experimental measurements obtained for the early stages of this experiment [2, 3]. Non-dimensional height of the collapsing water column at the left wall versus non-dimensional time is shown in Fig. 2. The predicted height of water column corresponds very well with experimental measurements. The computations were performed on the 120×50 and 240×100 finite element meshes. For both meshes the computed reduction of height was the same.

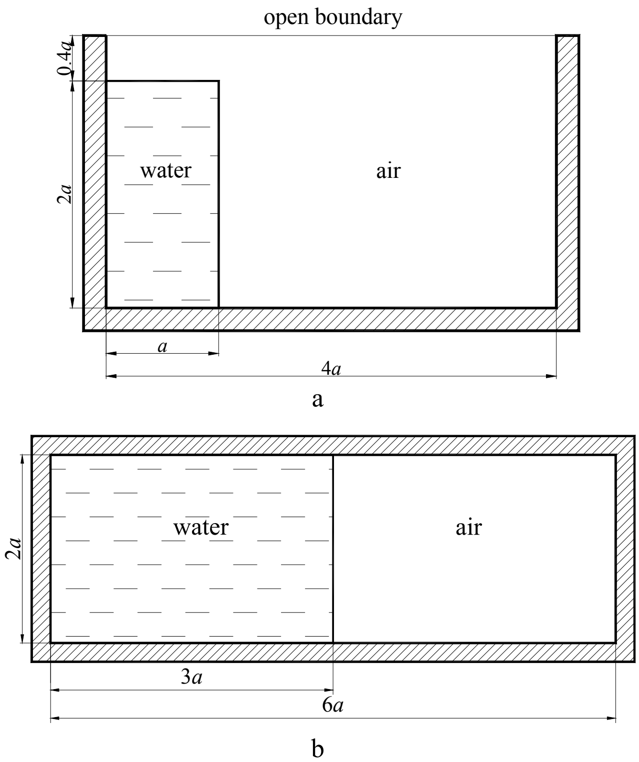

Fig. 1 Geometry of broken dam problems

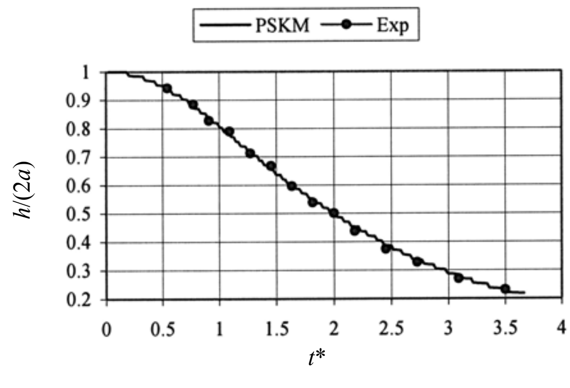

Fig. 2 Quantitative comparison of the numerical results (PSKM) and experimental measurements (Exp). Non-dimensional height of the collapsing water column $h/(2a)$ versus non-dimensional time $t^* = t\sqrt{g/a}$

The dam break flow in a confined domain (Fig. 1, b) was simulated in order to investigate the breaking wave phenomena [24]. A rectangular cavity with dimensions 0.09m×0.03m was considered (a=0.015 m). At initial time t=0.0 s, water is confined in the left half of the cavity. Later it is subject of vertical gravity and free to

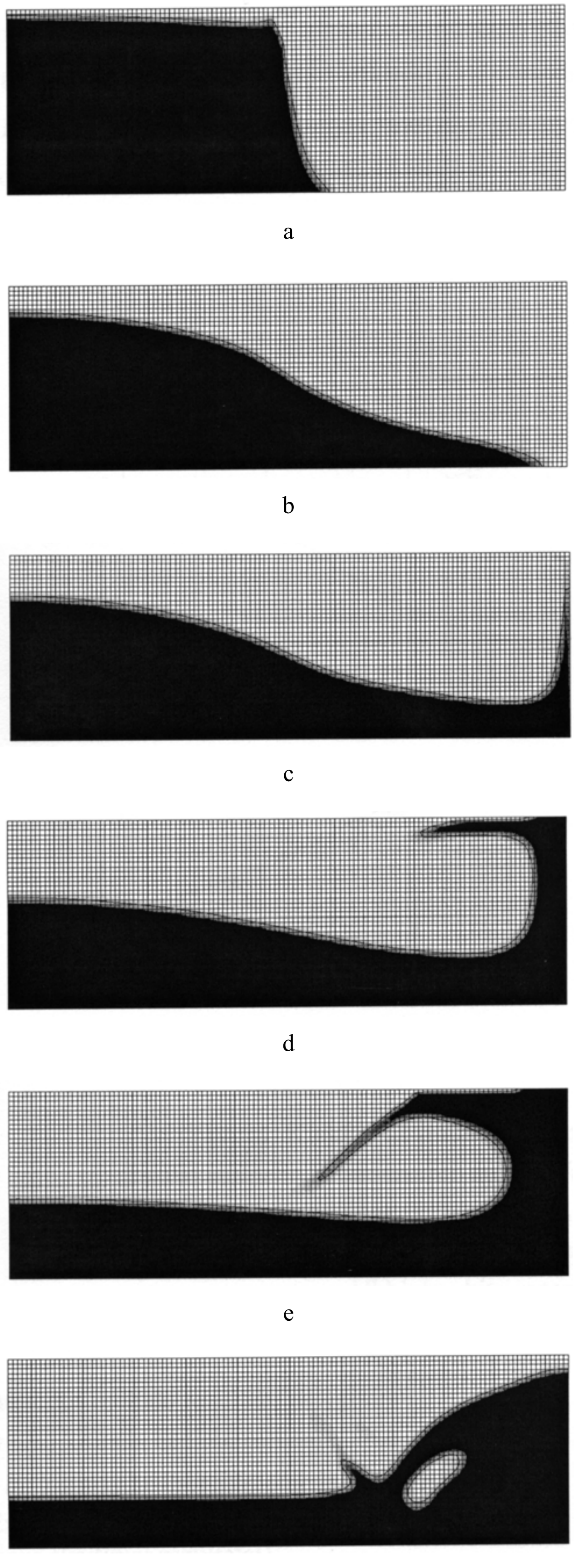

Fig. 3 Time evolution of the dam break flow, the pseudo-concentration function and stationary finite element mesh: a – t=0.025s; b – t=0.075s; c – t=0.100s; d – t=0.15s; e – t=0.200s; f – t=0.275s

move. The densities and viscosities of water and air are the same as in the previous example. The slip boundary conditions (4) were applied on the bottom and sides of the reservoir. The non-slip boundary conditions and pressure (7) were prescribed on the upper wall. The 120×40 and 240×100 finite element meshes are employed for computations.

Fig. 3 illustrates the breaking wave phenomena in the confined reservoir simulated by the PCM. The pseudo-concentration function value 0.5 represents the exact shape of moving interface. The accuracy of interface capturing techniques is limited by the mesh size, therefore, grey colours illustrate the transition region between different phases of the flow. The complexity of velocity fields occurring in the different stages of breaking wave phenomena can be easily captured using simple structured meshes. To predict the behaviour of the small bubbles correctly is more difficult task. At t=0.275 s the backward moving wave has folded over and a small amount of air is trapped. In experiments however, this air is present in the form of small bubbles. The current methodology has been derived for sharp interfaces, therefore, the mesh needs significant refinement to a resolution smaller than the bubble size.

Large density ratios might cause numerical oscillations at the interface [12, 14]. In this work, the problem was solved carefully computing density values (7) in a finite element *area*. The density is taken to be constant in the element and φ values in formula (7) were averaged

$$\varphi_{EL} = \frac{\sum_{i=1}^{n} G_i \varphi_i}{area} \qquad (13)$$

here n is the number of Gauss points; G_i are Gauss coefficients; φ_i are φ the values at Gauss points. The adopted strategy works very well when the interface is not very sharp and its thickness is greater then one element size. The observed mass loss is only 0.15%, which is good achievement for interface capturing techniques [25]. The implemented interface reconstruction technique (12) is very economic and the obtained results are enough accurate. The interface sharpening techniques preserving global mass conservation are the subject for future research.

5. Conclusions

Dam break flow simulation is performed by the interface capturing technique based on the pseudo-concentration method. The proposed numerical methodology consists of the GLS stabilised space-time finite elements, the pseudo - concentration function and the simple interface reconstruction technique. The numerical approach is validated by quantitative comparison with experimental measurements. The computed values of the reduction of water column height are in excellent agreement with the experimental data. The reliable GLS stabilization and efficient interface reconstruction technique avoids numerical oscillations modelling of breaking wave phenomena. The accurate numerical solution of the dam break problem including highly non-linear breaking waves proves that the proposed numerical technique is capable of simulating moving interfaces undergoing large topological changes.

References

1. **Stoker, J.J.** Water Waves.-New York: John Wiley & Sons, 1958.-415p.
2. **Martin, J.C., Moyce, W.J.** An Experimental study of the collapse of liquid columns on a rigid horizontal plane. -Philos. Trans. Roy. Soc.-London, 1952, NoA244, p.312-324.
3. **Koshizuka, S., Tamako, H., Oka, Y.** Aparticle method for incompressible viscous flow with fluid fragmentation. -J. Computational Fluid Dynamics, 1995, No4(1), p.29-46.
4. **Soulis, J.V.** Computation of two-dimensional dam-break flood flows. -Int. J. for Numerical Methods in Fluids, 1992, No14, p.631-664.
5. **Harlow, F.H., Welch, J.E.** Numerical calculation of time - dependent viscous incompressible flow. -J. Physics of Fluids, 1965, No8, p.2182-2189.
6. **Hirt, C.W., Nichols, B.D.** Volume of fluid (VOF) method for the dynamics of free boundaries. -J. Computational Physics, 1981, No39, p.201-225.
7. **Hirt, C.W., Cook, J.L., Butler, T.D.** A Lagrangian method for calculating the dynamics of an incompressible fluid with free surface. -J. Computational Physics, 1970, No5, p.103-124.
8. **Radovitzky, R., Ortiz, M.** Lagrangian finite element analysis of newtonian fluid flows. -Int. J. Numerical Methods in Engineering, 1998, No43, p.607-619.
9. **Hirt, C.W., Amsden, A.A., Cook, J.L.** An arbitrary lagrangian - eulerian computing method for all flow speeds. -J. Computational Physics, 1974, No14, p.227-253.
10. **Masud, A., Hughes, T.J.R.** A space - time Galerkin/ least - squares finite element formulation of the navier-stokes equations for moving domain problems. -Computational Methods in Applied Mechanics and Engineering, 1997, No146(1-2), p.91-126.
11. **Souli, M., Zolesio, J.P.** Arbitrary Lagrangian-Eulerian and free surface methods in fluid mechanics. -Computational Methods in Applied Mechanics and Engineering, 2001, No191, p.451-466.
12. **Osher, S., Sethian, J.A**. Fronts propagating with curvature-dependent speed: algorithms based on Hamilton-Jacobi formulation. -J. Computational Physics, 1988, No79, p.12-49.
13. **Harvie, D.J.E., Fletcher, D.F.** A new volume of fluid advection algorithm: the defined donating region scheme. -Int. J. for Numerical Methods in Fluids, 2001, No35(2), p.151-172.
14. **Kačeniauskas, A.** Two - phase flow modelling by the level set method and finite elements. -Energetika, 2000, No 4, p. 22-31.
15. **Quecedo, M, Pastor, M.** Application of the level set method to the finite element solution of two-phase flows. -Int. J. for Numerical Methods in Engineering, 2001, No50(3), p.645-663.
16. **Sharif, N.H., Wiberg, N.-E., Levenstam, M.** Free surface flow through rock-fill dams analyzed by FEM with level sėt approach. -Computational Mechanics, 2001, No27(3), p.233-243.
17. **Thompson, E.** Use Of Pseudo-concentration to follow creeping viscous flow during transient analysis. -Int. J. for Numerical Methods in Fluids, 1986, No7, p.749-761.

18. **Aliabadi, S., Tezduyar, T.E.** Stabilized - finite-element/interface - capturing technique for parallel computation of unsteady flows with interfaces. -Computational Methods in Applied Mechanics and Engineering, 2000, No190(3-4), p.243-261.
19. **Lewis, R.W., Ravindran, K.** Finite element simulation of metal casting. -Int. J. for Methods in Engineering, 2000, No47, p.29-59.
20. **Gao, D.M.** A three-dimensional hybrid finite element-volume tracking model for mould filling in processes. -Int. J. for Numerical Methods in Fluids, 1999, No29(7), p.877-895.
21. **Kačeniauskas, A., Rutschmann, P.** Parallel FEM software for CFD problems. -Informatica, 2004, No3(15), p.363-378.
22. **Hughes, T.J.R., Franca, L.P., Hulbert, G.M.** A new finite element formulation for computational fluid dynamics: viii. the Galerkin/least-squares method for advective-diffusive equations. -Computer Methods in Applied Mechanics and Engineering, 1989, No73, p.173-189.
23. **Kačeniauskas, A.** Parallel solution of convective transport equation by GLS stabilised space - time FEM. -Mechanika, 2003, No4(42), p.61-66.
24. **Dieterlen, R., Maronnier, V., Picasso, M., Rappaz, J.** Numerical simulation of free surface computational mechanics. New trends and applications, Eds. S. Idelson, E. Onate, E. Dvorkin, Barcelona, 1998.-13p.
25. **Kačeniauskas, A.** Interface capturing techniques for tank sloshing problems. -Mechanika, 2002, No5(37), p.14-17.

A. Kačeniauskas

SUGRIUVUSIOS UŽTVANKOS TĖKMĖS MODELIAVIMAS PSEUDOKONCENTRACIJOS METODU

R e z i u m ė

Straipsnyje sugriuvusios užtvankos tėkmė modeliuota baigtiniais elementais ir pseudokoncentracijos metodu. Uždavinio apibrėžimo sritis diskretizuota erdvės ir laiko baigtiniais elementais, o skaitinė schema stabilizuota Galiorkino mažiausiųjų kvadratų metodu. Dvipusis paviršius modeliuotas pseudokoncentracijos metodu ir paprasta paviršiaus rekonstrukcijos procedūra.

Keli sugriuvusios užtvankos uždavinio variantai išspręsti siekiant patikrinti skaitinę koncepciją. Apskaičiuotos vandens stulpelio aukščio mažėjimo vertės palygintos su fizikinių matavimų rezultatais. Lūžtančių bangų reiškinio modeliavimo rezultatai parodė, kad nagrinėtas skaitinis algoritmas gali modeliuoti ryškiai kintančios topologijos dvipusius paviršius.

A. Kačeniauskas

DAM BREAK FLOW SIMULATION BY THE PSEUDO-CONCENTRATION METHOD

S u m m a r y

In this paper, dam break flow simulation is performed by the finite elements and the pseudo-concentration method. Solution domain is discretized by the space-time finite elements, while numerical schemes are stabilised by the Galerkin least squares method. The moving interface is modelled by the pseudo-concentration method and the simple interface reconstruction technique.

Several cases of the broken dam problem are solved in order validate the numerical approach. The computed values of the reduction of water column height are compared with the experimental measurements. The simulated breaking wave phenomena shows that investigated numerical technique is capable of modelling moving interfaces undergoing large topological changes.

A. Каченяускас

ПРИМЕНЕНИЕ МЕТОДА ПСЕВДО-КОНЦЕНТРАЦИИ ДЛЯ МОДЕЛИРОВАНИЯ ПОТОКА ЧЕРЕЗ РУХНУВШУЮ ПЛОТИНУ

Р е з ю м е

В статье поток через рухнувшую плотину моделируется при помощи конечных элементов и метода псевдо - концентрации. Область определения задачи дискретизируется пространственно-временными конечными элементами, а численная схема стабилизируется методом наименьших квадратов Галеркина. Двухсторонняя поверхность моделируется применяя метод псевдо - концентрации и простую процедуру реконструкции поверхности.

Несколько вариантов задачи рухнувшей плотины решено, чтобы проверить численную методику. Вычисленные значения убывающей высоты столбца воды сравнены с результатами экспериментов. Результаты моделирования ломающихся волн показали, что разработанная методика позволяет моделировать поверхности подвергающиеся большим топологическим изменениям.

Received August 28, 2004