The Numerical Simulation of Two-Dimensional Vertical Incompressible Viscous Flow

Piotr Zima

Abstract: The paper presents the numerical simulation of two-dimensional vertical incompressible viscous free surface flow. The model is based on the Navier-Stokes equations. To solve governing equation the pressure method (SIMPLE) was used. In this case the divergence of Navier-Stokes equations are considered and Poisson equation on pressure is obtained. To solve this problem the Lax-Wendroff scheme with staggered Euler grid was applied. This approach ensures second order accuracy of numerical scheme. The Poisson equation was solved using the Successive Over-Relaxation (SOR) method. The domain (where the fluid is present) was defined using Marker and Cell (MAC) method. In order to assess the accuracy of the models same simple test cases were analyzed. The results of laboratory experiments presented by S. Koshizuka et al. were used to verify the calculations. Collapse of a water column is analyzed using the presented method.

Keywords: two-dimensional vertical incompressible viscous flow, Navier-Stokes equation, water splash effects, numerical simulation

1. Introduction

In many hydraulic engineering problems the free surface rapidly varied flows are encountered, especially including water jets, hydraulic jumps, flow over and under gates, collapse of water wall. Numerical simulations of these problems are often made with the Navier-Stokes equations and continuity equation (mass conservation law). The Navier-Stokes equations (NSE) are the system of partial differential equations describing incompressible viscous flow (Sawicki, 1998). Unfortunately, an analytical solution of NSE, in most real cases, does not exist and they must be solved using numerical methods. A lot of classical methods for solving the water flow equations are well known and successfully applied
Zima (Fletcher 1991, Anderson 1995, Tannehill 1984). They are based on the three fundamental methods: finite difference method (FDM), finite element method (FEM) and finite volume method (FVM) (Gryboś, 1998). One of the most popular and often used method is SIMPLE method (Semi Implicit Method for Pressure Linked Equations)(Potter, 1982). In this algorithm we consider a divergence of NSE. It leads to the Poisson equation which describes pressure field evolution (pressure-correction equation). SIMPLE method will be used to solve two-dimensional vertical NSE on rectangular, staggered grid.

2. Governing equations

Governing equations for incompressible viscous flow are the continuity equation (1) and NSE (2) in the following form (Sawicki, 1998):

\[ D = \nabla u = 0 \]  
\[ \frac{Du}{Dt} = f - \frac{1}{\rho} \nabla p + \nu \Delta u \]

where: 
- \( D \) – velocity divergence, 
- \( u \) – velocity vector, 
- \( f \) – external forces vector, 
- \( p \) – pressure, 
- \( \rho \) – density, 
- \( \nu \) – kinetic viscosity factor.

Pressure-correction equation can be written:

\[ \Delta \bar{p} = -\nabla \cdot (u \cdot \nabla u) \]

where: \( \bar{p} \) – normalized pressure, \( \bar{p} = p / \rho \).

These equations are solved for the horizontal and vertical velocity variables \( u_x \) and \( u_y \) and the pressure \( p \). The external body forces vector includes the acceleration due to gravity \( g \) with components \( \rho g \), in the directions of the x-y cartesian coordinate system. We can rewrite above equations in differential form for both velocity components:

- continuity equation

\[ D = \frac{\partial u_x}{\partial x} + \frac{\partial u_y}{\partial y} = 0 \]

- momentum equations

\[
\begin{align*}
\frac{\partial u_x}{\partial t} + u_x \frac{\partial u_x}{\partial x} + u_y \frac{\partial u_x}{\partial y} &= g_x - \frac{1}{\rho} \frac{\partial p}{\partial x} + \nu \left( \frac{\partial^2 u_x}{\partial x^2} + \frac{\partial^2 u_x}{\partial y^2} \right) \\
\frac{\partial u_y}{\partial t} + u_x \frac{\partial u_y}{\partial x} + u_y \frac{\partial u_y}{\partial y} &= g_y - \frac{1}{\rho} \frac{\partial p}{\partial y} + \nu \left( \frac{\partial^2 u_y}{\partial x^2} + \frac{\partial^2 u_y}{\partial y^2} \right)
\end{align*}
\]
• pressure-correction equation

\[
\frac{\partial^2 \bar{p}}{\partial x^2} + \frac{\partial^2 \bar{p}}{\partial y^2} = - \left\{ \left( \frac{\partial \bar{u}_x}{\partial x} \right)^2 + 2 \left( \frac{\partial \bar{u}_x}{\partial y} \right) \left( \frac{\partial \bar{v}_y}{\partial x} \right) + \left( \frac{\partial \bar{v}_y}{\partial y} \right)^2 \right\}
\]  

(6)

For all test cases presented in this paper no turbulence model is incorporated into solution and surface tension effects are neglected.

3. Numerical methods

Equations (3) and (4) will be solved by FDM using SIMPLE method. The main idea of this method is splitting the governing equations solution in two steps:

• first step is prediction of the velocity field integrating equation (4), the explicit scheme is used to obtain values of velocity components with values of pressure from previous time step,
• second step is computation of the pressure-correction field, which then corrects the velocity to satisfy the zero divergence condition (4).

To determine the free surface location the flow domain (where the fluid is present) is defined using Marker and Cell (MAC) method (Welch, 1965).

To integrate the NSE system (4) and Poisson equation (6) in space by FDM the two-dimensional domain \( z-x \) should be discretized into a set of cells. To make this discretion the Euler staggered grid was used. In each cell variables \( u_x, u_y \) and \( p \) are located in different places (see Fig.1). There are four types of cells (Fig.2): full (F), boundary (B), surface (S) and empty (E).
The velocities are specified at the cell faces while the pressure is specified at the cell center. To discretize NSE on a staggered grid the Lax-Wendroff scheme was applied. This approach ensures second order accuracy of numerical scheme (Potter, 1982). The pressure-correction equation is solved using the Successive Over-Relaxation (SOR) method. The boundary conditions for these equations and their solution stability conditions must be satisfied.

The initial velocity field $\mathbf{u}$, pressure $p$ and the initial domain fill (initial position of the free surface) are specified by 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 $\mathbf{u}$ and Newton’s second law.

4. Numerical simulations and experimental verifications of two-dimensional vertical free surface flow

The numerical algorithm presented above was verified for typical Computational Fluid Dynamics (CFD) test case: collapse of water column (Dam-Break Flow - DBF) (Mohapatra, 1999). There were considered two problems: simulation with and without the obstacle on the domain bottom. Geometry of domain and initial position of water column is presented in Fig.3. The results of
laboratory experiments carried out by S. Koshizuka et al. (1995) were used to verify the calculations. In the experiment, the box made of glass with the scale \( L = 14.6 \) cm was used. The water column is supported by a vertical wall, which was drawn up rapidly (approximately 0.05 sec) for the beginning of collapse. Shapes of water column was recorded by a video camera. The numerical simulation was made with following parameters: number of cells 1891 \((dx = dy = 0.973)\), number \( n \) of particles 512, 1152 and 2048. Gravitation unit \( g_r = -9.806 \) m/s\(^2\) and kinetic viscosity factor \( \nu = 1.0 \times 10^{-3} \) were imposed. Viscous effect on the boundary was neglected.

The first presented problem is a simulation of collapsing of water column. The results of experiment and computing (particle locations and velocity vectors) for the example calculations are presented in Fig 4 (test case without obstacle on the bottom).

![Fig. 4 Experimental and calculated results of collapse of water column (number of particles: 512)](image)
The position of the water wave front of collapsed water column is shown in Fig 6. The experimental and computing results (maximal position of particles in relation $x/L$) are shown on the normalized time ($t_n = t \sqrt{2g/L}$) background. The significant disagreement comparison is observed to $t_n=2.0$. To this time, the delay between the calculated and observed wave front location is evident. The reason is lack of bottom friction (the viscous effect was neglected). After $t_n=2.0$ the conformity of experimental and calculation result is quite good. There is no dependence on number of particles $n$.

Fig. 5 Experimental and calculated results of collapse of water column ($n = 2048$) with an obstacle
The collapse of water column with obstacle was considered as a next test case. A obstacle is located on the bottom wall at \(2L\) far from the left wall. The size is \(h \times 2h\), where \(h = 2.4\) [cm]. Comparison of the calculated and experimental results are presented in Fig.6. After time \(t=0.2\) [s], the run water crashes on the obstacle and splashed toward the upper-right direction. In the calculation result, the water is splashing more horizontally. The reason of this difference is probably neglecting of reaction between air and water on the free surface (assumption – the pressure is constant on the free surface).

![Experimental and calculated results of water wave location](image)

\textbf{Fig. 6} Location of front of water wave - experimental and calculated results

5. Summary and conclusions

In this paper, the numerical simulation of the collapse of the water column (standard test case in CFD) was considered. This numerical simulation is based on the results obtained using two-dimensional vertical flow equations (NSE) that allow the computation of the velocity and pressure fields. The numerical solution is based on SIMPLE algorithm. The FDM with staggered Euler grid was applied. The pressure-correction equation was solved using the SOR method. To define the free surface and spatial arrangement of fluid the MAC method was used. The results of laboratory experiments carried out by S. Koshizuka et al. were used to verify the calculations.

Following are the conclusions drawn from presented in this paper study:

- presented method can be applied to numeric solutions of many hydraulic problems where the strong deformation, splashing or rapidly varied free surface is occur;
• presented numerical calculations results of collapse of water column confirm suitability of this model to two-dimensional vertical analysis of the DBF mechanics, especially for dry-bed conditions.

References
Welch J., Harlow F., Shannon J. (1965) The MAC Method, Los Alamos Scientific Laboratory of the University of California

Author
Dr Piotr Zima, Gdansk University of Technology, Faculty of Civil and Environmental Engineering, pzim@pg.gda.pl