

FREE SURFACE FLOW SIMULATION USING A TWO-DIMENSIONAL MODEL BASED ON MARKER AND CELL METHOD

Wei Chek Moon

School of Civil Engineering
Engineering Campus, Universiti Sains
Malaysia
Penang, Malaysia
vmoonpower@yahoo.com.my

Yeh Gal Chan

School of Civil Engineering
Engineering Campus, Universiti Sains
Malaysia
Penang, Malaysia
yehgalchan@hotmail.com

How Tion Puay

River Engineering and Urban Drainage
Research Centre (REDAC)
Engineering Campus, Universiti Sains
Malaysia
Penang, Malaysia
redac_puay@usm.my

Abstract— The interest in the numerical simulation of free surface flows is increasing due to the rapid development of computer technology. A two-dimensional numerical model based on the Marker and Cell (MAC) method is developed in this study. In the current development, the advection term in the Navier-Stokes equation is solved using the third-order Quadratic Upstream Interpolation for Convective Kinematics (QUICK) scheme. For verification and validation purposes, the performance of the numerical model is checked by simulating the classical dam-break flow problem. Satisfactory agreements are observed between the numerical and the previous experimental results in term of the development of free surface profile, the propagation of front wave, the attenuation of depth at the origin and the impact force on the downstream wall.

Keywords—numerical simulation, MAC method, QUICK scheme, free surface flow

I. INTRODUCTION

The water flow phenomena is complex and difficult to solve analytically without assumptions. With the emergence and rapid development of computer technology, the Computational Fluid Dynamics (CFD) model is used to solve the problem numerically. Several numerical methods for free surface flow problem have been developed in the field of CFD over the past few decades. For example, Marker and Cell (MAC) method [1], Volume of Fluid (VOF) method [2], level set method [3] and others. Each numerical technique has its own advantages and weaknesses in term of accuracy, simplicity of the algorithm, flexibility and speed of convergence. A so-called comprehensive numerical model is capable of simulating the flow of various cases robustly and predicting the flow parameters qualitatively and quantitatively.

MAC method was invented back in the early sixties at the Los Alamos Laboratories, with the considerably greater computing power that we now enjoy. However, it is witnessing a revival and, for free surface fluid flow problems, showing itself the equal of any of the competing methods. Many researchers have been using the MAC method widely to discretize equations for simulation works [4, 5]. One of the key features of the MAC method is the use of Lagrangian virtual particles, whose coordinates are stored, and which move from a cell to the next according to the latest computed velocity field. It is deemed to contain fluid, if a cell contains a particle, thus providing flow visualization of the free surface.

Numerical simulation of free surface flow has been the subject of extensive research for a long time [6, 7]. A higher order scheme is needed to produce accurate results with less diffusion. This study aims to develop a two-dimensional model based on MAC method with the application of higher order scheme to solve the advection term. The numerical results in term of the development of free surface profile, the propagation of front wave, the attenuation of depth at the origin and the impact force are discussed and compared with the available experimental results obtained from literature reviews.

II. NUMERICAL PROCEDURE

A. Governing equations

The governing equations for the numerical model are the momentum equation (derived from the conservation of linear momentum) and the continuity equation (derived from the conservation of mass) which are shown in (1) and (2). Based on the staggered grid system as shown in Fig. 1, the momentum equations in x and y directions are discretized with the finite difference method (FDM) and solved to satisfy the continuity equation. For an incompressible fluid, the fluid density is constant.

$$\frac{\partial V}{\partial t} + (V \cdot \nabla)V = -\frac{1}{\rho} \nabla P + \nu \nabla^2 V + g \quad (1)$$

$$\nabla \cdot V = 0 \quad (2)$$

Where ρ is the density of the fluid, P is the pressure, V is the velocity vector, ν is the kinematic viscosity and g is the gravitational acceleration vector.

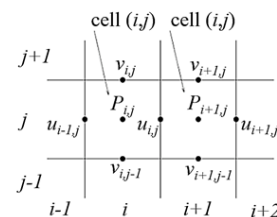


Fig. 1. Staggered grid system.

B. Numerical algorithm

MAC method calculations utilize an Eulerian finite-difference mesh on rectangular cells, together with time advancement through finite intervals. Fig. 2 shows the one-time step algorithm.

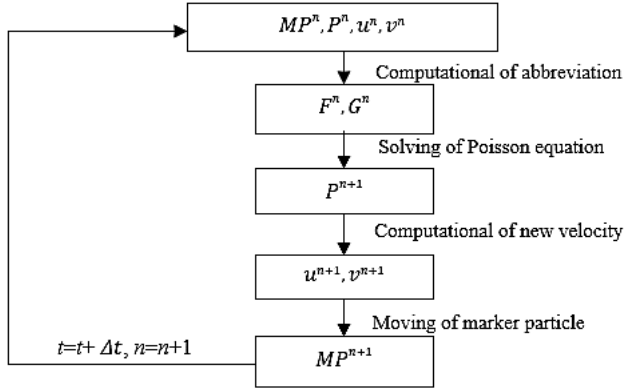


Fig. 2. General algorithm of MAC method.

By using the explicit Euler scheme, the temporal terms $\partial u / \partial t$ and $\partial v / \partial t$ in the momentum equations can be written as

$$u^{n+1} = F - \Delta t \frac{\partial P^{n+1}}{\partial x} \quad (3)$$

$$v^{n+1} = G - \Delta t \frac{\partial P^{n+1}}{\partial y} \quad (4)$$

Here,

$$F = u^n + \Delta t \left[v \left(\frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 u}{\partial y^2} \right) - \frac{\partial uu}{\partial x} - \frac{\partial uv}{\partial y} + g_x \right]^n \quad (5)$$

$$G = v^n + \Delta t \left[v \left(\frac{\partial^2 v}{\partial x^2} + \frac{\partial^2 v}{\partial y^2} \right) - \frac{\partial v}{\partial y} - \frac{\partial uv}{\partial x} + g_y \right]^n \quad (6)$$

All the terms in F and G are evaluated at time step n , whereas the pressure gradients are evaluated at time step $n+1$. By modifying the governing equations with the condition of zero velocity divergence, Poisson equation for the pressure P^{n+1} as shown in (7) is obtained. In this study, the Poisson equation is solved implicitly by Gauss-Seidel method.

$$\frac{\partial^2 P^{n+1}}{\partial x^2} + \frac{\partial^2 P^{n+1}}{\partial y^2} = \frac{1}{\Delta t} \left(\frac{\partial F^n}{\partial x} + \frac{\partial G^n}{\partial y} \right) \quad (7)$$

Once the new pressure field is known, the final velocity field that satisfies the zero velocity divergence condition is then updated. In this study, the first-order upwind scheme

which is originally devised in the MAC method is replaced by the third-order Quadratic Upstream Interpolation for Convective Kinematics (QUICK) scheme [8] to solve the advection terms in (1). For example, the velocity u in direction x at cell i is evaluated as follows.

$$u_i^{n+1} = \begin{cases} u_i^n - \Delta t u_i^n \frac{3u_{i+1}^n + 3u_i^n - 7u_{i-1}^n + u_{i-2}^n}{8\Delta x}, & \text{if } u_i^n > 0 \\ u_i^n - \Delta t u_i^n \frac{-3u_{i-1}^n - 3u_i^n + 7u_{i+1}^n - u_{i+2}^n}{8\Delta x}, & \text{otherwise} \end{cases} \quad (8)$$

At each step, appropriate boundary conditions are imposed at the imaginary cells and the free surface boundaries. In each free surface cell, the pseudo pressure is equal to the true pressure which is necessary to satisfy the free surface normal stress condition. Otherwise, only the physical boundary conditions on the velocity field itself are required. Lastly, based on the new velocity field, the massless marker particles (MPs) imbedded in the fluid are moved, giving the new fluid configuration. Marker particles are moved with a velocity that is a weighted average of the nearest cell velocities. For example, the velocity of marker particle in x direction is calculated as an interpolated value of the four nearest velocities u at cells shown in Fig. 3.

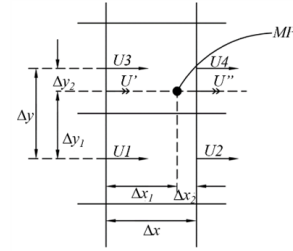


Fig. 3. Determination of velocity of marker particle in x direction.

C. Numerical model setup

Prior to assess the correctness of the developed algorithms in the numerical model, the numerical model is first verified with the dam-break flow problem, which is a famous benchmark testing case. The numerical simulation setup is shown in Fig. 4, where a fixed volume of water is released on an initially dry horizontal channel. The propagation of wave front, L and the attenuation of depth at the origin, h at the origin are compared against Koshizuka et al. [9] and Martin and Moyce [10].

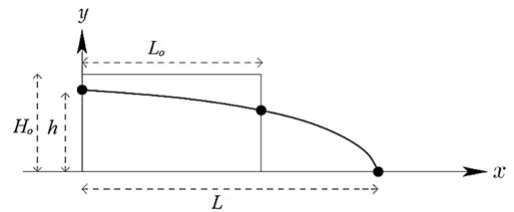


Fig. 4. Dam-break flow of finite volume fluid.

The accuracy of the numerical model is checked via the validation with the previous experiment conducted by Lobovsky et al. [11]. The numerical model domain and the initial conditions are set based on Fig. 5. Non-slip condition is used for the floor and walls. Other parameters used in the numerical model are shown in Table I.

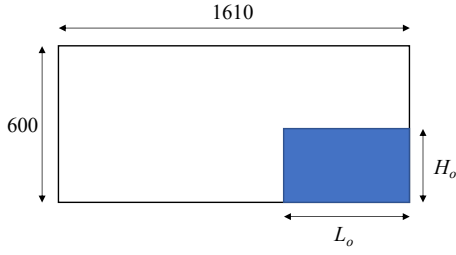


Fig. 5. Initial geometry of water column (dimensions are in millimetres).

TABLE I. NUMERICAL SIMULATION CONDITION FOR DAM-BREAK FLOW PROBLEM

Parameter	Value
Initial water column width, L_o (m)	0.30
Initial water column height, H_o (m)	0.60
Kinematic viscosity, ν (m^2s^{-1})	1.0×10^{-6}
Density, ρ (kgm^{-3})	1000.00
Grid size, Δx (m)	0.01
Grid size, Δy (m)	0.01
Time step, Δt (s)	1.0×10^{-6}
Gravitational acceleration, g (ms^{-2})	-9.81

III. RESULTS AND DISCUSSION

A. Verification of numerical model

Figs. 6 and 7 display the dimensionless front wave propagation and the dimensionless attenuation of depth at the origin, respectively. Based on Fig. 6, there is a small discrepancy between the numerical and experimental results at the later stage of the flow. This might be due to the fact that in the experiment, there is friction on the floor and thus dissipation plays a role, causing a loss in momentum and thus deceleration. Furthermore, the removal of dam gate in the experiment may account for the delay of water release on top of the column, especially when the water column height to column base ratio is high, where the friction between dam gate and water particle cannot be neglected. These problems are not accounted for in the numerical simulation. On the contrary, the numerical results by MAC+QUICK model show good agreement with the experimental results by Koshizuka et al. [9] and Martin and Moyce [10] in terms of the rate of attenuation of depth at the origin (Fig. 7).

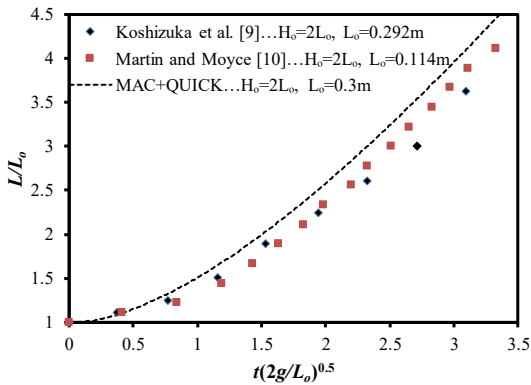


Fig. 6. Front wave propagation.

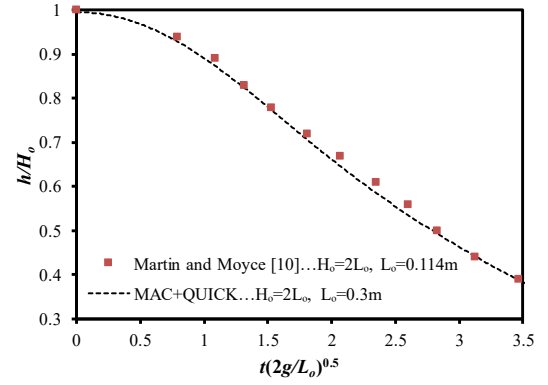


Fig. 7. Attenuation of depth at the origin.

B. Validation of numerical model

a) Comparison of Flow Profile

Fig. 8 depicts the flow profiles at different times of the simulation compared with the experimental results by Lobovsky et al. [11]. For the purpose of comparison between the different orders of the numerical schemes, the results of a single-phase model by a proprietary software, FLOW-3D which adopts the first-order scheme are also added in Fig. 8.

As shown in Fig. 8, several stages are displayed upon the removal of the dam gate. The first stage is the advancing downstream wave front, followed by the wave impact on the downstream wall and the consequent run-up. Subsequently, a plunging breaker is formed in the back flow that develops in a combination of bore propagation and a mixing layer on top of the laminar layer of fluid yet advancing towards the wall. A lot of vorticity is generated at this stage. This corresponds to findings by Landrini et al. [12] who analysed this phenomenon numerically.

From the comparison, both numerical models exhibit the capability of producing almost identical flow surface profile as experiment at the early stage of dam-break flow. However, it is interesting to note that the FLOW-3D model with a lower order numerical accurate scheme produces a lower surge speed. This can be seen at time $t = 0.45$ s, the simulated wave for the case with the first-order scheme is about to hit the downstream wall, whereas the splashed-up wave is observed in the experiment and simulation by the MAC+QUICK model (Fig. 8).

After the flow hits the downstream wall, the MAC+QUICK model could not reproduce similar flow profile as in the experiment after the time $t = 0.862$ s (Fig. 8). This might be due to the lack of turbulence model in the numerical model as well as the mesh resolution factor. Another plausible reason is the finite number of particles representing the fluid in the MAC method, which might create the false regions of void in flows with high shear [13]. The free surface boundary conditions are also problematic, where the conditions are applied in an approximate way that often leads to instability at the free surface [1]. As the MAC method implements the linear interpolation scheme for efficiency consideration, large errors could arise near the highly distorted free surface, especially for breaking wave. Therefore, it cannot correctly represent the physical situation.

b) Comparison of pressure on downstream wall

In the experiment by Lobovsky et al. [11], the impact pressures on the vertical wall at the downstream wall at four

different heights were measured (Fig. 9). The recorded pressure P is non-dimensionalized with regards to the hydrostatic pressure $\rho g H_0$ and plotted versus the non-dimensional time t^* which is defined as $t(g/H_0)^{0.5}$.

A comparison of the pressure peaks and their corresponding occurrence time was made between the experimental and numerical results. A slight discrepancy

between the numerical results by different scheme order of accuracy is observed in terms of the arrival time of the wave at the downstream wall. It is interesting to note that the front wave speed for the lower order scheme is slower than that in the experiment. On the other hand, excellence performance is shown by the MAC+QUICK model, where the simulated wave hits the downstream wall at the same moment as in the experiment.

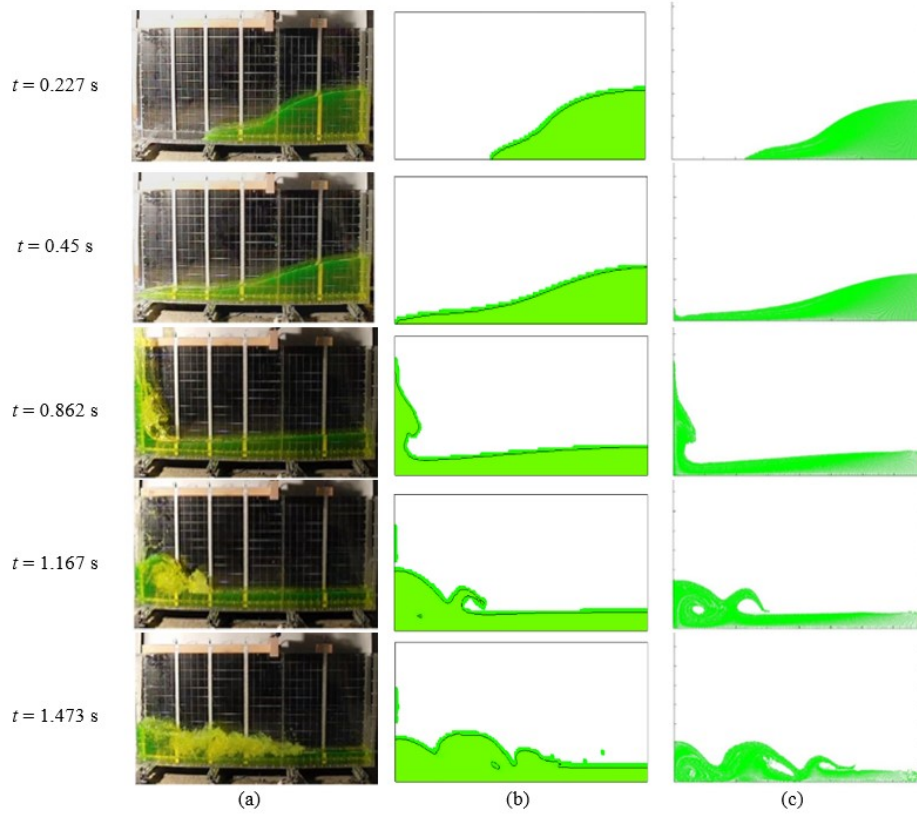


Fig. 8. Comparison of surface profile of dam-break problem between (a) experiment by Lobovsky et al. [11], (b) simulation by FLOW-3D and (c) simulation by MAC+QUICK model.

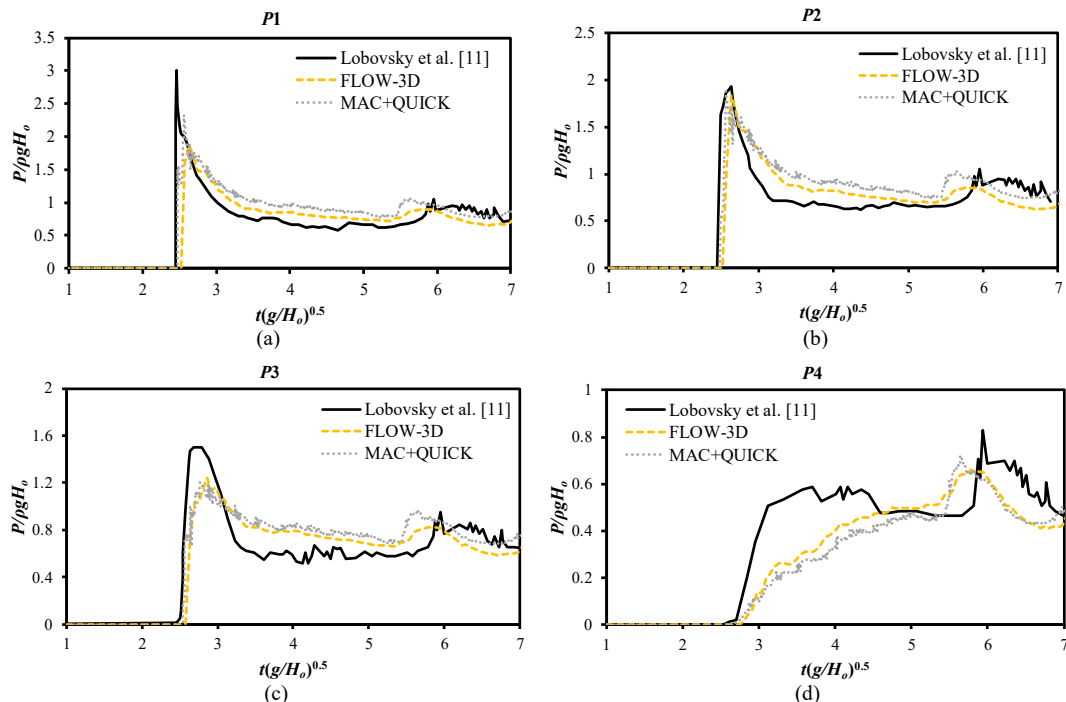


Fig. 9. Pressure time histories of sensors (a) P1, (b) P2, (c) P3 and (d) P4 at 3 mm, 15 mm, 30 mm and 80 mm from flume's bottom, respectively.

In terms of pressure peaks, it can be noticed in Fig. 9 that the bottom sensor $P1$ (3 mm from the flume's bottom) records the highest value, which is approximately three times the hydrostatic pressure (in the experiment). Conversely, the pressure sensors at the higher position ($P2$ and $P3$ at 15 and 30 mm from the flume's bottom, respectively) experience the impact induced by the run-up of the flow. As for sensor $P4$ (80 mm from the flume's bottom), it does not record a direct impact event since it is located at the highest position. As compared to the experimental results, the MAC+QUICK model with the third-order scheme also shows closer results than the FLOW-3D with the first-order scheme, although there is some discrepancy in pressure peaks for $P1$.

As shown in Fig. 9, there is a second increment in pressure recorded in the experiment at around $t^* = 6$. This pressure increment is induced by the splashed-up wave collapsed downwards, causing an impact on the pressure sensor. However, both numerical models fail to reproduce the similar phenomenon as the flow becomes turbulent and the entrainment of air voids occurs. The abovementioned increment is obtained earlier in time for both numerical models as compared to the experiment. As for the results by FLOW-3D model, the induced peak pressure by the collapsed wave is much lower than that in the experiment.

IV. CONCLUSIONS

In this study, a two-dimensional numerical model has been developed to simulate dam-break problem based on the MAC method. Modifications have been made to increase the accuracy of the model by using the QUICK scheme to solve the advection terms in the momentum equation. The model has been shown to perform satisfactorily. However, the model could not reproduce the flow profile of dam-break problem precisely after the wave breaking and overturning. Further investigations on the applicability of the present model for other applications are necessary, in consideration of the limitations of MAC method.

REFERENCES

- [1] F. H. Harlow and J. E. Welch, "Numerical calculation of time-dependent viscous incompressible flow of fluid with free surface," *Physics of Fluids*, vol. 8, no. 12, pp. 2182-2189, 1965.
- [2] C. W. Hirt and B. D. Nicholas, "Volume of fluid (VOF) method for the dynamics of free boundaries," *Journal of Computational Physics* vol. 39, no. 1, pp. 201-225, 1981.
- [3] S. Osher and J. A. Sethian, "Fronts propagating with curvature-dependent speed: algorithms based on Hamilton-Jacobi formulations," *Journal of Computational Physics*, vol. 79, no. 1, pp. 12-49, 1988.
- [4] T. Kokado, T. Hosoda and T. Miyagawa, "Methods for evaluating rheological coefficients of self-compacting concrete by numerical analysis," *Concrete Library of JSCE*, no. 38, pp. 51-70, 2001.
- [5] F. J. Trunk, J. D. Dent and T. E. Lang, "Computer modeling of large rock slides," *Journal of Geotechnical Engineering*, vol. 112, no. 3, pp. 348-360, 1986.
- [6] S. Marrone, A. Colagrossi, A. Di Mascio and D. Le Touzé, "Analysis of free surface flows through energy considerations: Single-phase versus two-phase modeling," *Physical Review E*, vol. 93, no. 5, pp. 053113-1-053113-13, 2016.
- [7] D. Violeau and B. D. Rogers, "Smoothed particle hydrodynamics (SPH) for free-surface flows: past, present and future," *Journal of Hydraulic Research*, vol. 54, no. 1, pp. 1-26, 2016.
- [8] B. P. Leonard, "A stable and accurate convective modelling procedure based on quadratic upstream interpolation," *Comput. Methods in Appl. Mech. Eng.*, vol. 19, no. 1, pp. 59-98, 1979.
- [9] S. Koshizuka and Y. Oka, "Moving-particle semi-implicit method for fragmentation of incompressible fluid," *Nuclear Science and Engineering*, vol. 123, no. 3, pp. 421-434, 1996.
- [10] J. C. Martin and W. J. Moyce, "Part IV: An experimental study of the collapse of liquid columns on a rigid horizontal plane," *Series A, Mathematical and Physical Sciences*, vol. 244, no. 882, pp. 312-324, 1952.
- [11] L. Lobovsky, E. Botia-Vera, F. Castellana, J. Mas-Soler and A. Souto-Iglesias, "Experimental investigation of dynamic pressure loads during dam break," *Journal of Fluids and Structures*, vol. 48, pp. 407-434, 2014.
- [12] M. Landrini, A. Colagrossi, M. Greco and M. P. Tulin, "Gridless simulations of splashing process and near-shore bore propagation," *Journal of Fluid Mechanics*, vol. 591, pp. 183-213, 2007.
- [13] M. Rudman, "Volume-tracking methods for interfacial flow calculations," *International Journal for Numerical Methods in Fluids*, vol. 24, no. 7, pp. 671-691, 1997.