⦿ **Technical Paper**

Two Dimensional Numerical Simulation of Liquid Sloshing+

Sin Young Kang* and Han Il Park**

(*Received April* 13, 1988)

액체 슬라싱에 관한 2차원 수치 시뮬레이션

강 신 영 · 박 한 일

Key Words : SOLA VOF Algorithm(솔라보프 알고리듬), Liquid Sloshing(액체 슬라싱), Numerical Simulation(수치 시뮬레이션)

초 록

자유표면 유동을 시뮬레이션할 수 있는 수치 알고리듬 중 가장 최근에 개발된 volume of fluid (VOF) 방법을 이용하여 뚜껑이 닫힌 사각 컨테이너 속의 액체 슬라싱을 시뮬레이션 하였다. 그 결과 유동이 작은 경우에는 실험치와 거의 일치 하였으나 과격한 액체유동으로 컨테이너벽에 순간적인 큰 충격을 유발하는 경우에는 알고리듬이 불안정하게 되어 장기간 시뮬레이션을 어렵게하는 문제점이 발견되었다. 이 문제점을 해결하기 위해 VOF 알고리듬 중 유량이동 알고리듬을 수정하여 원래의 알고리듬으로 시뮬레이션한 결과와 비교 분석하였다.

1. Introduction

The problem associated with liquid sloshing can be found in many engineering applications such as tank truck accidents, liquid fuel sloshing in space rockets, liquid cargo ships, and on a much larger scale, the oscillation of water in lakes and harbors occuring as the result of earthquake. The sloshing loads resulting from sloshing liquids may cause structural damage, create a destabilization effect and produce environmental hazards.

In the present study, the finite difference method(FDM) is proposed for the simulation of liquid sloshing in containers. One difficulty of simulating free surface flows lies in locating the free surface. Recently, Hirt and Nichols[1] developed an algorithm that uses the volume of fluid (VOF) technique to track a complicated free surface configuration by advecting either horizontal or vertical blocks of fluid between cells. In the VOF method a function F is defined whose value is unity at any point occupied by fluid and zero otherwise. The average value of F in a cell would represent the fractional volume of the cell occupied by fluid.

With this VOF technique, the liquid sloshing in partially filled enclosed prismatic tanks is simulated. The liquid is assumed to be homogeneous and to remain laminar. The numerical solution, however, was only able to obtain first few forcing cycles. This preludes the study of post-impact phenomena and the multi-period simulation of liquid sloshing which requires a sufficient number

+ 1987년도 한국해양공학회 추계학술대회 발표(1987년 11월)
* Member, Korea Maritime University
** Korea Maritime University

of statistically distributed impact pressure whose magnitude and durations vary randomly from cycle to cycle even though the external excitation is harmonic.

In this regard, the present paper proposes a modified advection algorithm which takes into account surface orientation and transports trapezoidal shapes from cell to cell. A few comparisons between numerical results and experimental data have been made (Fig. 1). Also, the results obtained using the modified scheme and the original scheme are presented and compared.

2. Governing Equations

Consider a two dimensional, rigid, rectangular, enclosed tank that is partially filled with a homogeneous viscous liquid. A moving coordinate system, fixed with respect to the tank, is used to describe the motion as shown in Fig. 2. The origin of the coordinate system is positioned at the lower left corner of the tank (small 0 in Fig. 2). The y-axis is along the left tank wall and the x-axis is the bottom. The moving coordinate is translating and rotating relative to an inertial

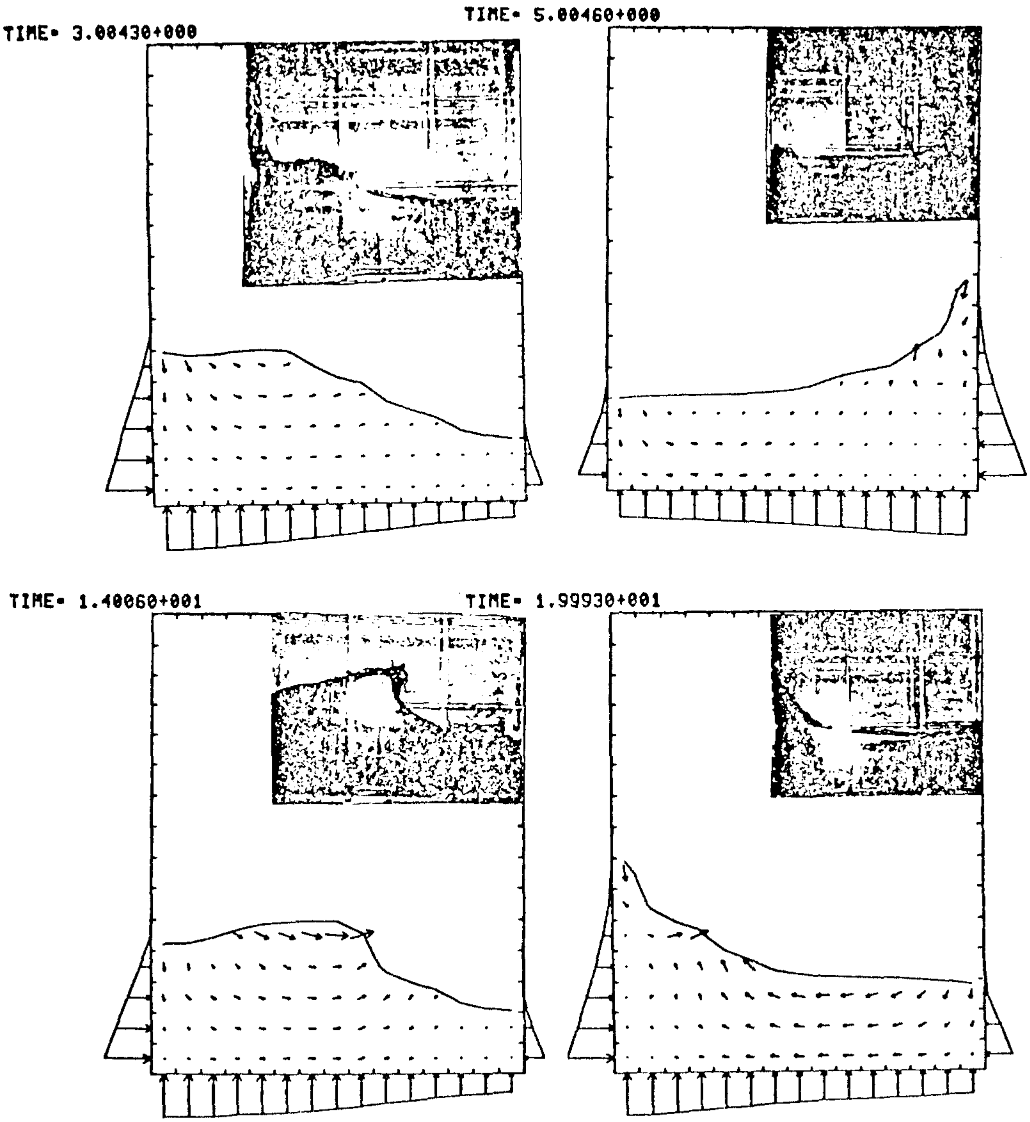


Fig. 1 Comparison of numerical solutions and experimental results

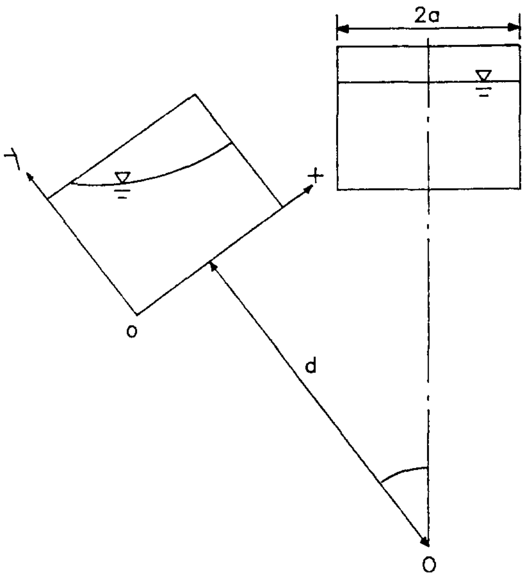


Fig. 2 Moving coordinate system

system.

The equations governing the motions of the liquid relative to the moving coordinate are as follows:

$$\frac{\partial u}{\partial x}+\frac{\partial v}{\partial y}=0 \tag{1}$$

$$\frac{\partial u}{\partial t}+u\frac{\partial u}{\partial x}+v\frac{\partial u}{\partial y}=-\frac{1}{\rho}\frac{\partial p}{\partial x}+g_x+\nu\left[\frac{\partial^2 u}{\partial x^2}+\frac{\partial^2 u}{\partial y^2}\right] \tag{2}$$

$$\frac{\partial v}{\partial t}+u\frac{\partial v}{\partial x}+v\frac{\partial v}{\partial y}=-\frac{1}{\rho}\frac{\partial p}{\partial y}+g_y+\nu\left[\frac{\partial^2 v}{\partial x^2}+\frac{\partial^2 v}{\partial y^2}\right] \tag{3}$$

where u and v are the velocities of the fluid in the x-and y-directions, p is the pressure, ν is the coefficent of kinematic viscosity, ρ is the density and (g_x, g_y) denote body accelerations in the x-and y-directions, respectively. For a roll motion about an axis on $(a, -d)$, g_x and g_y could be specified as follows[2]:

$$g_x=-g\sin\theta+\ddot{\theta}(y+d)+2\dot{\theta}v+\dot{\theta}^2(x-a) \tag{4}$$

$$g_y=-g\cos\theta-\ddot{\theta}(x-a)-2\dot{\theta}u+\dot{\theta}^2(y+d) \tag{5}$$

where g is the gravitational acceleration and θ, $\dot{\theta}$ and $\ddot{\theta}$ represent the angular displacement, angular velocity and angular acceleration. In computation, θ is specified as $\theta=\theta_0 \sin wt$ in which θ_0 and w are the amplitude and the frequency, respectively, of the roll motion (θ is shown in Fig. 2).

Finite difference representation of the governing differential equations is obtained as described in Reference (3). A uniform finite difference is used. The fluid region is surrounded by a single layer of fictitious cells which are used to set boundary conditions so that the same difference equations used in the interior of the mesh can also be used at the boundaries. A staggered grid is used with u-velocity at the middle of the vertical sides at a cell, v-velocity at the middle of the horizontal sides, and pressure p and the volume of fluid function F at the cell center.

3. Solution Procedure

Following the SOLA-VOF algorithm[3], the basic solution procedure consists of the following operation:

1) Start with an initial condition.

2) Calculate the first guesses of the velocities for the new time-level to the explicit finite difference form of equations (2) and (3).

3) Adjust the cell pressures iteratively. For an interior cell, the pressure is adjusted until equation (1) is satisfied to a prescribed level of accuracy. While for a surface cell, the cell pressure is adjusted until the dynamic free surface boundary condition is satisfied. The velocities are then adjusted to reflect the change in the cell pressure.

(4) Compute the volume of fluid function F for the next time-level to give the new fluid configuration. A donor-acceptor fluxing is used to compute the advection of F. During the process boundary conditions are imposed.

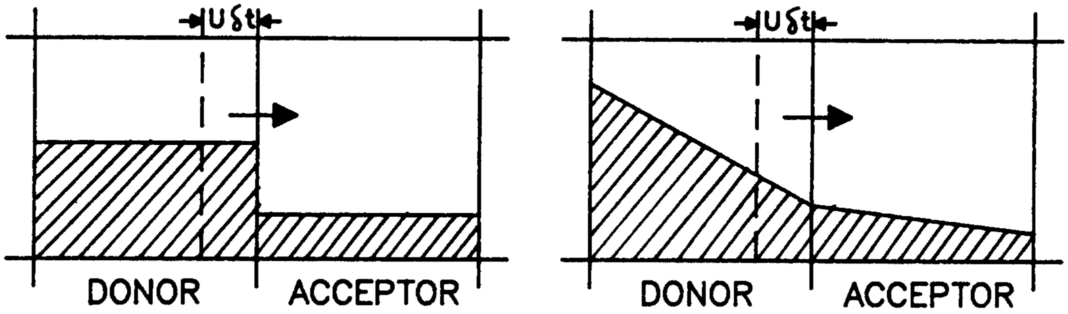
(5) Repeat the procedure for the next step of the time-march.

The advancement of the volume of fluid function in time is carried out by a donor-acceptor method. While the original SOLA-VOF program exchanges blocks of fluid between cells and fluid surface regardless of their orientation, the new algorithm takes into account surface orientation and transports trapezoidal shapes from cell to cell. Both schematics are illustrated in Fig. 3.

4. Comparison of Numerical Results

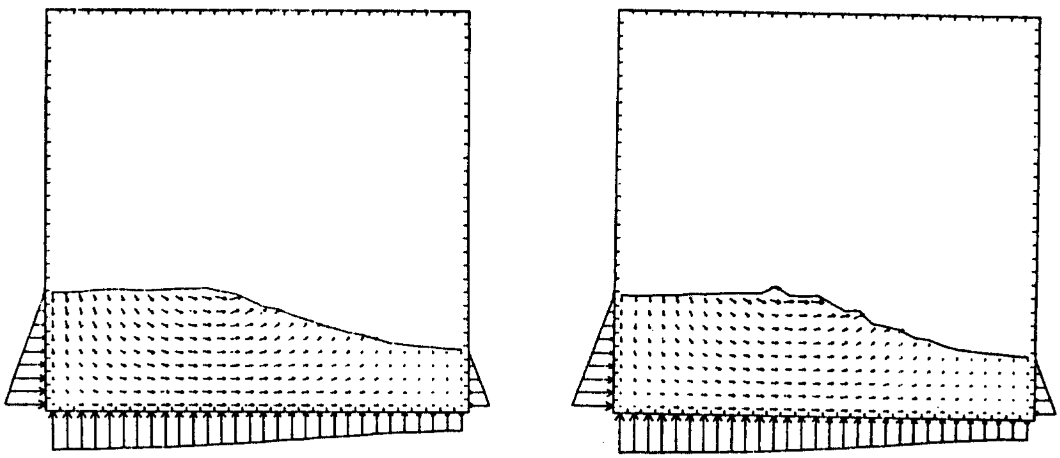
Liquid sloshing inside non-baffled and horizontally baffled tanks is chosen for the comparison of the original and modified schemes. At an early stage, prior to the initial impact, the results of these schemes appear identical. Especially for low amplitude excitations, the entire flow fields are almost identical. However, it is evident that the modified scheme show a smoother surface configuration than original scheme (see Fig. 4 and 5).

In the case of large amplitude sloshing (24 degree roll) excitation, the computation using the original scheme is forced to terminate because an excessive number of iterations is required in the pressure iteration subroutine. With the modified scheme such difficulty is less encountered.



(a) (b)

Fig. 3 Schematics used in calculating F advection
(a) Original scheme, (b) Modified scheme crosshatched regions are the amounts of F advected



(a) (b)

Fig. 4 Flow configuration and pressure field (unbaffled)
(a) Modified, (b) Original ($\theta_0=8°$, $\omega=1.0511$ rad/s, $\nu=1.2\times10^{-5}$ ft^2/s, $\rho=1.95$ $slug/ft^3$, $d=0$, $a=30ft$, $H=60$ ft, $\Delta T=0.1$ s, water level $0.25H$)

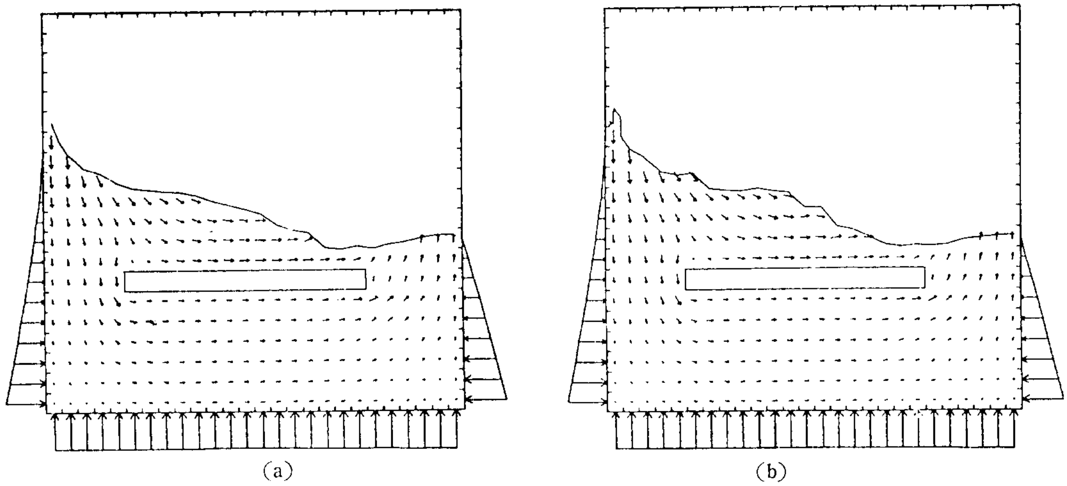


Fig. 5 Flow configuration and pressure field (baffled)
(*a*) Modified, (*b*) Original ($\theta_0=4°$, $\omega=5.6745$ *rad/s*, $\nu=0$, $\rho=1.95$ *slug/ft*^{3}, d=0, $a=1.0ft$, $H=2.0ft$, $\Delta T=0.0034s$, water level $0.5H$, baffle location: i(7-21), j(8))

5. Conclusion and Discussion

The use of VOF technique seems to pose little problem for the computation of gentle sloshing. The modified advection algorithm shows a smoother surface configuration while showing the possibility of long term numerical simulation of liquid sloshing. However, the current algorithm lacks the capability of simulating repeated violent impact and post-impact process resulting from large amplitude excitations. Violent impact with splashing fluid is a very complicated problem. The flow may become turbulent after initial impact. Moreover, unless a fine grid is used or a two phase model is developed, the splashing fluid becomes sub-grid scale. Therefore, a further research is required in this direction.

References

1) Nichols, B. D. C. W. Hirt and R. S. Hotchkiss, "SOLA-VOF: A Solution Algorithm for Trans-, ient Fluid with Multiple Free Boundaries", Los Alamos Scientific Laboratory Report, LA-8355, August 1980

2) Kang, S. Y. "Analysis of Liqid Impact on Moving Containers", Master thesis, Florida Antlantic University, 1984

3) Hirt C. W. and B. D. Nichols, "Volume of Fluid (VOF) Method for the Dynamics of Free Boundaries", Journal of Computational Physics, Vol. 39, pp. 201-225, 1981