14th Annual (International) Mechanical Engineering Conference – May 2006
Isfahan University of Technology, Isfahan, Iran

COMPARISON OF FINITE VOLUME AND PENDULUM MODELS FOR SIMULATION OF SLOSHING

M.H. Djavareshkian[1], M. Khalili[2]
Mechanical Engineering Department, University of Tabriz, Iran
E-mail: djavaresh@Tabrizu.ac.ir

Abstract
In this paper, a numerical procedure and an analytical work have simulated Sloshing of liquid in a tanker. With the numerical procedure, one of the most appropriate Finite Volume methods called VOF (Volume Of Fluid) method, which is a pressure-based algorithm for solving Navier-Stokes equations, has been presented for tracking the flow in containers. In this simulation, a non-orthogonal mesh with collocated formulation has been used. For the analytical work, the Pendulum Model has also been applied to simulation of the sloshing of liquid. This model is a cost effective tool for the simulation of sloshing. The results of this simulation for a tanker truck containing liquid fuel with different free surfaces have been compared with each other and with published results of another numerical method.

Keywords: VOF, Sloshing, Pendulum Model, Finite Volume

Introduction
Every year, there are many fatal accidents associated with the role over of tanker trucks carrying fuel. The tendency to roll over is even higher when the tanker truck is partial filled and the fuel inside the tanker is free to slosh. Typically a half-empty tanker truck has ten times more tendency to role over than a passenger car. Sloshing also happens in the gas tank of automobiles in the wing of an aircraft were the fuel is stored and in water towers during an earthquake. In these problems the liquid is subjected to external forces and as a result the free surface of the liquid becomes unstable and sloshing occurs. The sloshing amplifies the maximum fluid force exerted on the structures and subjects them to instability. To understand the fluid–structure interaction during sloshing, accurate knowledge of the free surface behavior is the most important factor.

Until now, a lot of researches have been done about sloshing. Amabili [1] and Warnitchai et al [2] solved the sloshing problems by analytical methods and verified the results with some experiences. Martin et al [3] and Abramson [4] did some experiences about sloshing. Another possible way is Finite Element method Aliabadi et al [5,6] and Johnson et al [7] used it for studying the free surface movements. The most useful method of computational works in sloshing flows is Finite volume method used by Nichols et al [8,9], Maxwell [10], Deville [11], Hirt et al [12,13] and Liu Jun [14].

In above computational works, numerical simulation of free-surface flow problems is based on the solution of a complex set of partial differential equations governing the conservation of mass and momentum, commonly known as the Navier-Stokes equations. It needs to use large amounts of memory and a lot of computer processes. So Hirt et al [15] designed the fractional Volume Of Fluid (VOF) method which on one hand provides the same information as is available in other strong methods of Finite Volume, and on the other hand requires only one storage word for each grid cell. This method is a kind of volume-tracking techniques applied to a fixed Eulerian mesh and has been used by others such as Liu Jun [16] for studying the free surface movement and its results in different problems.

Whereas each method has special abilities for solving some CFD problems, Finite Volume methods are generally the best ways for free surface and sloshing problems, and from different methods in this group, the most successful method is the VOF technique because of its simplicity and

1- Assistant Prof .
2- MSc. Student

robustness and so is appropriate for stratified or free surface flows.
In this paper, the VOF method is employed for tracking the flow in containers. In this simulation, a non-orthogonal mesh with collocated pressure-base formulation has been used. The pendulum model has also been applied to simulation of the sloshing of liquid. This model is a cost effective tool for the simulation of sloshing. The results of this simulation for a tanker truck containing liquid fuel with different free surfaces have been compared with each other and other available results.

Governing Equations

The basic equations, which describe conservation of mass, momentum and scalar quantities, can be expressed in Cartesian tensor form as

$$\frac{\partial \rho}{\partial t} + \frac{\partial (\rho u_j)}{\partial x_j} = 0 \qquad (1)$$

$$\frac{\partial (\rho u_i)}{\partial t} + \frac{\partial (\rho u_i u_j - T_{ij})}{\partial x_j} = S_i^u \qquad (2)$$

$$\frac{\partial (\rho \phi)}{\partial t} + \frac{\partial (\rho u_j \phi - q_j)}{\partial x_j} = S^\phi \qquad (3)$$

The stress tensor and scalar flux vector are usually expressed in terms of basic dependent variable. The stress tensor for a Newtonian incompressible fluid is

$$T_{ij} = -p\delta_{ij} + 2\mu \vec{D} \qquad (4)$$

The scalar flux vector usually given by the Fourier-type law:

$$q_j = \Gamma_\phi \left(\frac{\partial \phi}{\partial x_j}\right) \qquad (5)$$

Discretization

The discretization of the above differential equations is carried out using a finite-volume approach. First, the solution domain is divided into a finite number of discrete volumes or cells, where all variables are stored at their geometric centers (see e.g. Fig.1). The equations are then integrated over all the control volumes by using the Gaussian theorem. The discrete expressions are presented affected with reference to only one face of the control volume, namely, e, for the sake of brevity.

For any variable ϕ (which may also stand for the velocity components), the result of the integration yields

$$\frac{\delta \upsilon}{\delta t}[(\rho\phi)_p^{n+1} - (\rho\phi)_p^n] + I_e - I_w + I_n - I_s = S_\phi \delta \upsilon \qquad (6)$$

where I's are the combined cell-face convection I^c and diffusion I^D fluxes. The diffusion flux is approximated by central differences and can be written for cell-face e of the control volume in Fig. (1) as:

$$I_e^D = D_e(\phi_p - \phi_E) - S_e^\phi \qquad (7)$$

where S_e^ϕ stands for cross derivative arising from mesh nonorthogonality. The discretization of the convective flux, however, requires special attention and is the subject of the various schemes developed. A representation of the convective flux for cell-face e is:

$$I_e^c = (\rho.V.A)_e \phi_e = F_e \phi_e \qquad (8)$$

the value of ϕ_e is not known and should be estimated by interpolation, from the values at neighboring grid points. The final form of the discretized equation from each approximation is given as:

$$A_P.\phi_P = \sum_{m=E,W,N,S} A_m.\phi_m + S'_\phi + S_{dc} \qquad (9)$$

where A's are the convection-diffusion coefficients. The term S'_ϕ in Eq. (9) contains quantities arising from non-orthogonality, numerical dissipation terms and external sources, and $(\rho\delta\upsilon/\delta t)\phi_P$ of the old time-step/iteration level (for time dependent equation). For the momentum equations it is easy to separate out the pressure-gradient source from the convected momentum fluxes. S_{dc} is the contribution due to the adapted deferred correction procedure.

Volume Of Fluid (VOF) Method

In this method, the actual location of the interface is determined by some additional computation based on the distribution of markers within the cell. This procedure is somewhat inconsistent with the customary practice to use only one value in each cell of a grid for each dependent variable defining the fluid state. This recognition led Hirt and Nichols [15] to design the fractional volume of fluid (VOF) method.
A function F be defined, of which the value is unity at any point occupied by fluid and zero otherwise. The average value of F in a cell then represents the fractional volume of the cell occupied by fluid. The interface occurs in the cells with fractional values. The volume fractions are updated during the

calculation according to the appropriate advection equation

$$\frac{\partial F}{\partial t}+u\frac{\partial F}{\partial x}+v\frac{\partial F}{\partial y}=0 \qquad (10)$$

This equation states that F moves with the fluid, and is the partial differential equation analogy of marker particles. In a Lagrangian grid, the equation reduces to the statement that F remains constant in each cell, thus serves solely as a flag identifying cells that contain fluid. In a Eulerian grid, the equation indicates that the flux of F moves with fluid through cell boundaries.

Pendulum Method

For solving the problem with an analytical method, we must simplify the geometries and flow characteristics. The pendulum model in case of sloshing is the best analytical way. In this case, we assume the sloshing shape, as oscillating of a solid point mass around a hinge. Figure 2 is a circular cylinder with radius *R*, contains some water with depth of *R-d*.

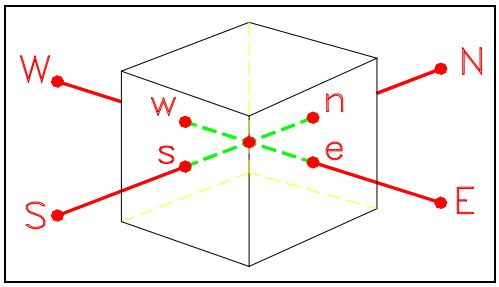


Fig. 1 - Typical grid – point and its equivalent pendulum model cluster and control volume

We can simulate sloshing of it with a simple pendulum connected to the center of cylinder with an imaginary arm with length of X_c. This length is calculated by:

$$\int_d^R x.y.dx=\int_d^R x.\sqrt{1-x^2}.dx=X_c.A=$$
$$X_c.\int_d^R y.dx=X_c.\int_d^R \sqrt{1-x^2}.dx \qquad (11)$$

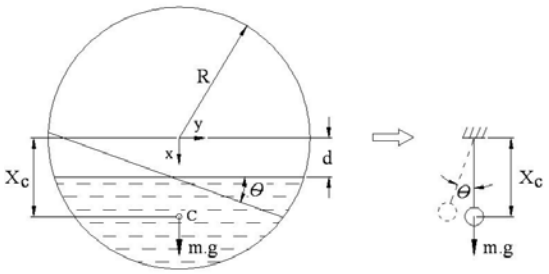


Fig. 2 - Sloshing in a half-empty cylinder

General equation of oscillating of this pendulum is:

$$I.\ddot{\theta}+C.R.\dot{\theta}+m.g.X_c.\theta=0 \qquad (12)$$

where I is the second moment of area relative to center of cylinder and *C* is damping factor. These parameters are given as below:

$$I=I_{yy}+I_{xx}=2\rho.t.[\int_d^R x^2.\sqrt{1-x^2}.dx+$$
$$\int_0^{\sqrt{R^2-d^2}} y^2.(x-d).dy] \qquad (13)$$

$$C=\mu.R \qquad (14)$$

where *t* is the thickness of the circle (length of cylinder) and μ is the viscosity of the fluid. The initial conditions of above mentioned oscillation equation are:

$$\theta\big|_{t=0}=\theta_0 \qquad (15)$$

$$\dot{\theta}\big|_{t=0}=0 \qquad (16)$$

where θ_0 is the angle of free surface after damping of pendulum and this surface then will be calculated from summation of different accelerations applied to system such as turning acceleration and gravity.
Here we integrate the oscillation equation using the Rung-Kutta order 4 and then we can find θ at each time. By finding the angle of free surface at each time, it is clear that the horizontal force exerted to wall from the liquid, is the *x*-component of total force calculated by multiplying the hydrostatic pressure by the wet area. By connecting this data on a diagram we have the horizontal force due to sloshing in each time.

Solution Algorithm

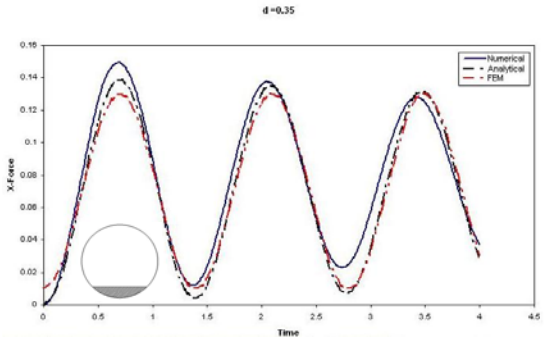
Most contemporary pressure-based methods employ a sequential iteration technique in which the different conservation equations are solved one after another. The common approach taken in enforcing continuity is by combining the equation for continuity with those of momentum to derive an equation for pressure or pressure-correction. The PISO algorithm is used in this work.

Results

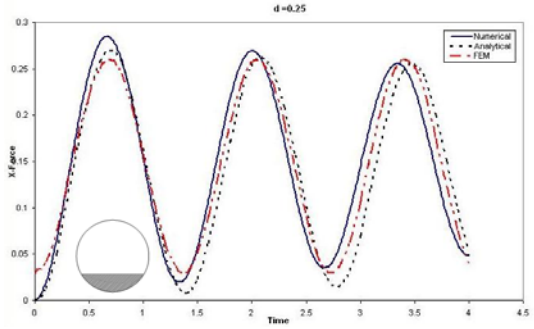
The results of the numerical simulations using the finite volume method, VOF, and pendulum models for simulation of a tanker truck having a circular cross-section with one unit in diameter are compared. The tanker-truck with a horizontal circular cylinder and 4m length in the axial direction of it, is moving in a road with velocity of 10m/s. The road has a circle with 250m radius. Two different accelerations applied to the system, gravity and centrifugal accelerations. Before entering of tanker in curved road, vertical gravity was the only component of acceleration and then at time t=0 a horizontal acceleration is added to system and sloshing occurs.

The series of the pictures in Fig. 3 show the results obtained using both the Numerical method (solid line) and the Analytical method (dotted line) for water elevation of 0.35, 0.25, 0.15 and 0.05 m measured from the center of the tanker. The results of these two methods also have been compared with results of Finite-Element method which have published before [7]. Also in each picture, the circle shows the cross-section of the tanker and the elevation of the water.

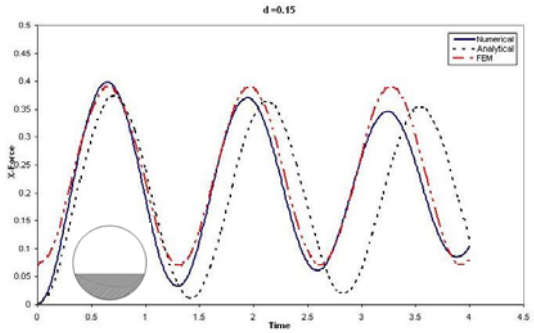
As it is expected, both methods have relatively good agreements when the water inside the tanker is low (d=0.35). The frequency of the sloshing is almost the same for all methods at low and moderate amount of the water inside the tanker (d=0.35 and d=0.25m). The amplitude of the force is in good agreements only at relatively low water elevations. The differences between amplitude and frequency of sloshing of different methods are significant when there is a significant amount of water inside the taker (d=0.05).



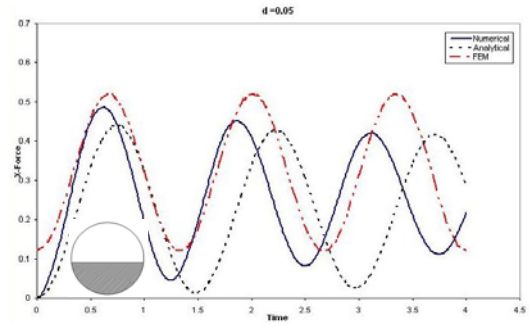
a) d=0.35



b) d=0.25



c) d=0.15



d) d=0.05

Figure 3- Horizontal force applied to the tanker during turning

Conclusion

In this paper, a finite volume, VOF, method is used to simulate sloshing in containers. The results of this simulation are also compared with a simple pendulum model and FEM. For sloshing applications in tanker trucks during turning, the results obtained from these methods are qualitatively comparable for low and moderate amount of fuel inside the tanker. On the other hand, in the presence of significant amount of water inside the tanker, the models result in different solutions. The numerical solutions (VOF) are obtained for a realistic centrifugal acceleration of tanker trucks during turning.

References

1- Amabili,M., "Free vibration of partially filled, horizontal cylindrical shells", 1995
2- Wanitchai,P. and Pinkaew,T., "Modeling of liquid sloshing in rectangular tanks with flow-dampening devices", 1997
3- Martin, J.C., and Moyce, W.J., An Experimental Study of the Collapse of Liquid Columns on a Rigid Horizontal Plane, Phil. Trans. Roy. Soc. London 244, pp.312-324, 1952
4- Abramson,H.N., "The Dynamics Behavior of Liquids in Moving Containers", ANSA SP-106, 1966
5- Aliabadi,S. and Tezduyar,T., "Stabilized-finite-element/interface-capturing technique for parallel computation of unsteady flows with interfaces", Compt. Method Appl. , Mech. Engg. 190, pp.243-61, 2000
6- Aliabadi,S., Johnson.A. and Abedi,A., "comparison of finite element and pendulum models for simulation of sloshing", Comp. & Fluid,32 pp. 535-545, 2003
7- Johnson,A. and Aliabadi,S., "Application of automatic mesh generation and mesh multiplication technique to very large scale free-surface flow simulations" Proceeding of the 7th International Conference on Numerical Grid Generation in Computational Field Simulations, Whistler, Biritish Columbia, Canada, September 25-28, 2000
8- Nichols,B.D. and Hirt,C.W., "Improved Free Surface Boundary Conditions for Numerical Incompressible-Fluid Calculations", , *J. Comp. Phys.*, 8, pp.434-448, 1971
9- Nichols,B.D. and Hirt,C.W., "Methods for Calculating Multidimensional, Transient Free Surface Flows Past Bodies", Los Alamos Scientific Laboratory Report LA-UR-75-1932, 1975
10- Maxwell,T.T., "Numerical Modeling of Free-Surface Flows", PhD Thesis, University of London, 1977
11- Deville,M.O., "Numerical Experiments on the MAC Code for a Slow Flow", *J. Comp. Phys*, 15,pp. 362-374, 1974
12- Hirt,C.W., and Shannon,J.P., "Free-Surface Stress Conditions for Incompressible Flow Calculations", , *J. Comp. Phys*., 2, pp.403-411, 1968
13- Hirt,C.W. , Nichols,B.D. , and Romero,N.C., "SOLA-A Numerical Solution Algorithm for Transient Fluid Flows", Los Alamos Scientific Laboratory Report LA-5852 1975
14- Liu Jun, "Application of Variable Blockage Method to Steady Flows over Curved Beds", C.F.D.U. report pDR/25, Imperial College 1985
15- Hirt,C.W. and Nichols,B.D., "Volume Of Fluid (VOF) Method for the Dynamics of Free Boundaries", , *J Computer Phys* 39, pp.201-25, 1981
16- Liu Jun, "Computer Modeling of Flows with a Free Surface", 1986