ATLANTIS PRESS

Advances in Engineering, volume 100

International Conference on Manufacturing Engineering and Intelligent Materials (ICMEIM 2017)

Numerical Investigation of Collapsing Liquid Cylinder

Jian Hu, Lei Guo, Chong Geng, Ting Guo, and Shi-li Sun*

College of Shipbuilding Engineering, Harbin Engineering University, Harbin, 150001, China

*Corresponding author

Keywords: Multiphase Flow, Volume of Fluid, Finite Volume Method, IDEAL Algorithm

Abstract. This paper presents an two-dimensional numerical model for incompressible multiphase flow. The Navier-Stokes equations are discretised by the finite volume method. IDEAL algorithm is implemented for solving the Navier-Stokes equations. PLIC-VOF method is used to track the free surface. The validity of the numerical model is examined through collapsing cylinder of water.

Introduction

Multiphase flows are frequently encountered in industrial and engineering processes, such as dam break flow, liquid sloshing, droplet impact. In the past decades, researchers have proposed various methods to simulate complex multiphase flows problems. Volume of fluid methods introduces the fluid volume fraction function F that indicates the fractional volume of the cell occupied by fluid[1,2]. The advection equation of F is solved using geometrical advection scheme, the accuracy and capabilities of the modern VOF algorithms greatly exceeds that of the older VOF algorithms such as the Hirt and Nichols VOF method.

This paper aims to achieve an accurate two-dimensional numerical model for incompressible multiphase flow. The Navier-Stokes equations are discretised by the finite volume method with a staggered-mesh layout. IDEAL algorithm is implemented for solving the Navier-Stokes equations. PLIC-VOF method is used to track the free surface. Various calculations are compared to experimental results.

Governing Equations and Numerical Model

This paper mainly studied incompressible gas-liquid two-phase flow with free surface. Each phase in the flow field must satisfy the regularity of incompressible flow, therefore, the whole flow field can be conceptually considered a single-phase flow and physical parameters are step changed on the free surface. Assuming that the velocity of two-phase is continuous on the free surface, the two-phase flow can be considered as continuous flow in inhomogeneous medium.

Continuity Equation

A continuity equation is an equation that describes the transport of some quantity, which is a local form of conservation laws in hydrodynamics. For the simplified case of gas-liquid two-phase flow without mass transfer, the continuity equations of gas and liquid can be simplified and cast into a one-field form which covering two-phase.

For incompressible fluids, the continuity equation is now expressed as:

$$\nabla \cdot \mathbf{V} = 0 . \tag{1}$$

Where $\mathbf{V} = (u, v, w)$ is the velocity, ρ is the density, $\nabla \cdot$ is the gradient operator.

Navier-Stokes Equations

Navier-Stokes equations arise from applying Newton's second law to fluid dynamics. By introducing Newton's law of viscosity and Stokes equations, the momentum equation can be written as:

$$\frac{\partial \mathbf{V}}{\partial t}+\nabla\cdot(\mathbf{V}\otimes\mathbf{V})=\nabla\cdot(\nu\nabla\otimes\mathbf{V})+\mathbf{S}_{\mathbf{V}} \quad (2)$$

where ν is the kinetic viscosity, $\mathbf{S}_{\mathbf{V}}$ is the source term defined as:

$$\mathbf{S}_{\mathbf{V}}=-\frac{1}{\rho}\nabla P+\mathbf{g}+\frac{1}{\rho}\mathbf{f}_{\sigma}+\frac{1}{\rho}(\nabla\otimes\mathbf{V})\cdot(\nabla\mu) \quad (3)$$

where $\mathbf{V}=(u,v,w)$ is the velocity, ρ is the density, μ is the dynamic viscosity, P is the pressure, $\nabla\cdot$ is the divergence operator, ∇ is the gradient operator, $\mathbf{g}$ is the gravity, $\mathbf{f}_{\sigma}$ is the surface tension.

VOF Method

In the volume of fluid method, which first proposed by Hirt and Nichols[1], the interface is tracked using a liquid volume fraction function F which is defined as:

$$F(\mathbf{x},t)=\frac{V_l(I,J)}{V(I,J)} \quad (4)$$

According to the difinition of F , the averaged properties are defined by:

$$\rho=F\rho_l+(1-F)\rho_g$$
$$\mu=F\mu_l+(1-F)\mu_g \quad (5)$$

In the incompressible flow field, the time dependence F is governed by the advection equation:

$$\frac{\partial F}{\partial t}+\nabla\cdot(\mathbf{V}F)=0 \quad (6)$$

which holds everywhere.

Special geometry-based methods have been proposed by many researchers. In a later section, the method is explained in detail.

Surface Tension

The surface tension coefficient σ is defined as the necessary work to increase a unit area of free surface. The magnitude σ is determined by the nature of the fluids, the value is positive for immiscible fluids and negative for miscible fluids. The Young-Laplace equation describes the pressure jump ΔP across a curved surface that caused by surface tension which is a function of the surface curvature:

$$\Delta P=\sigma\kappa \quad (7)$$

Where σ is the surface tension coefficient, κ is the surface curvature.

Discrete Equations and Solving Method

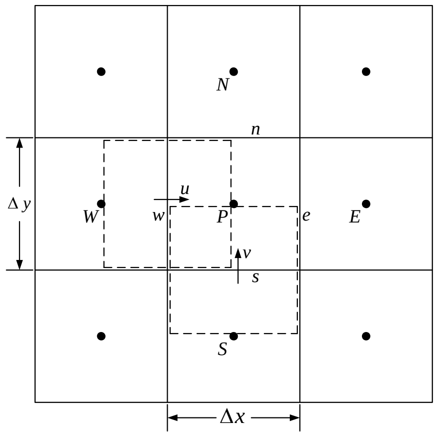


Figure 1. Staggered mesh

The finite volume method is applied in the discretisation of the governing equations. A uniform grid with square cells, which is known as structured grid, is used in the discretisation. Velocities are defined on cell-faces, whereas the other field variables are located at cell centers. Cells have equal width and height. Fig. 1 illustrates the mesh.

The finite volume discretisation of the continuity equation is based on the integral form of equation over the control volume:

$$\oint_V \nabla \cdot \mathbf{V} dV = 0 \quad . \tag{8}$$

Gauss' theorem can be applied to equation to give:

$$\sum_{f=1}^{n} A_f \cdot \mathbf{V}_f = 0 \quad . \tag{9}$$

The final form of momentum equations discretised with the fully implicit method for an orthogonal mesh is:

$$\frac{\mathbf{V}_P^{t+\delta t} - \mathbf{V}_P^t}{\delta t} V_P + \sum_{f=1}^{n} \rho_f F_f \mathbf{V}_P^{t+\delta t} = \sum_{f=1}^{n} \nu_f A_f \cdot (\nabla \otimes \mathbf{V})_f^{t+\delta t} + \mathbf{S}_{\mathbf{V}_P} V_P \quad . \tag{10}$$

One of the methods for solving the Navier-Stokes equations is called the pressure-correction method, the well-known SIMPLE-family algorithms belong to this method. In this paper, a modified SIMPLE algorithm called IDEAL algorithm[3] is adopted.

Free Surface Reconstruction

The advection equation of liquid volume fraction function F could be discretised and solved similarly to the momentum equation using finite volume method. However, numerical diffusion would excessively smooth the solution and blurring the interface.

In this paper, PLIC method is implemented. In the PLIC method, the normal vector to the interface must be determined first, a straight line is constructed by the obtained normal vector, and dividing the cell into two parts, each of which contains the proper volume of one of the two fluids. The estimation of the normal vector is based on the F, which is the only available phase information. Usually, the normal vector of cell (i, j) is determined by the nine cells only. It is important to find accurate approximation of the normal vector[4], there are many ways to achieve the normal vector, such as Green-Gauss gradient, volume-average gradient, least-squares gradient, minimization-principle gradient, Youngs' gradient. In this paper, the normal vector $\mathbf{n}_{i,j}$ is derived from the gradient of F.

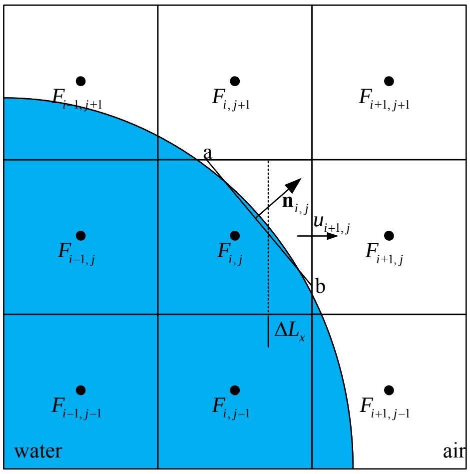
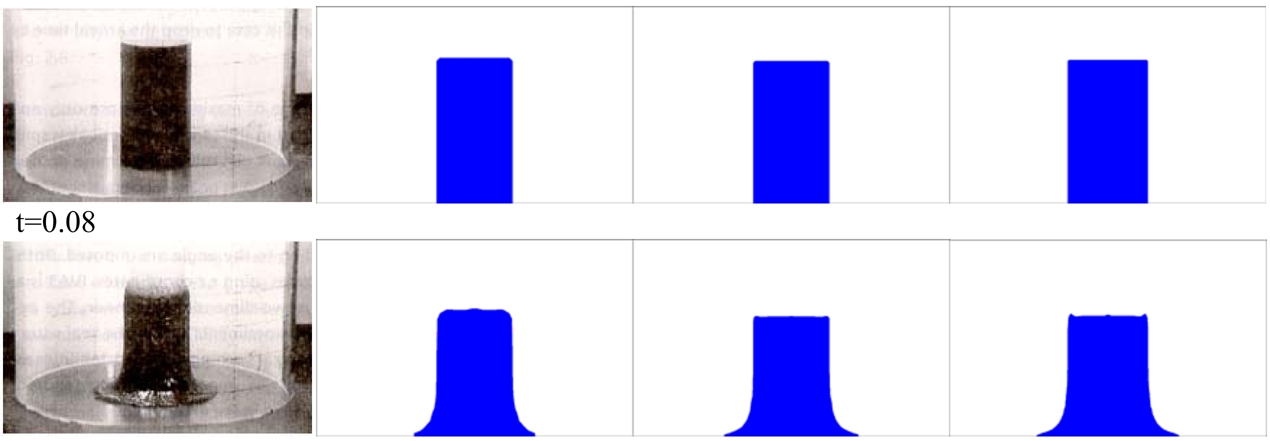


Figure 3. Reconstruction of the interface with a straight line

Once the normal vector is determined, the intercept points a and b match with F are easy to calculate. Then advection Equation needs to be solved, a separate treatment for every cell face, as proposed by Youngs[2], is implemented is this paper.

Results

t=0s



t=0.2s

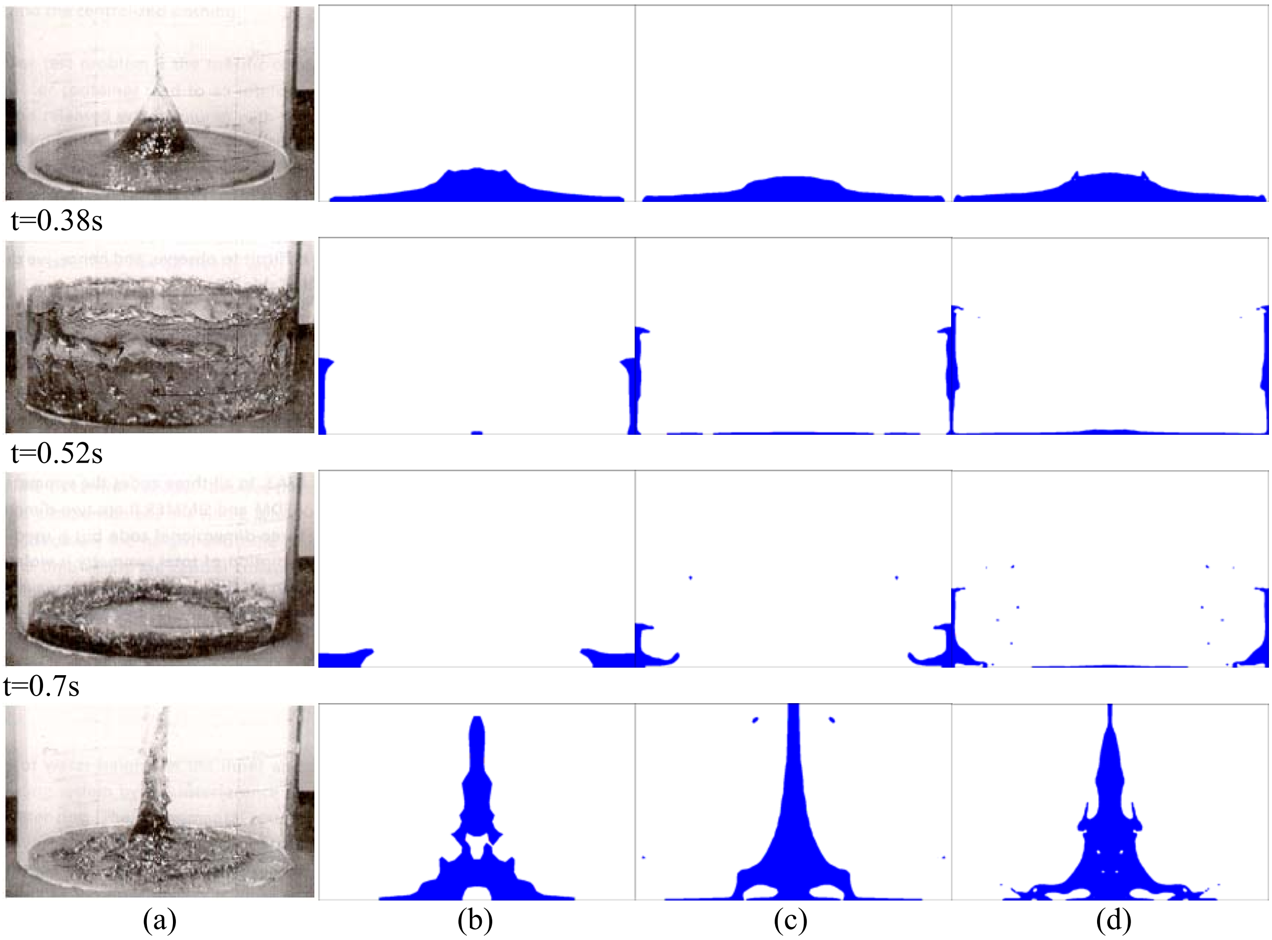


Figure 4. Comparison of collapsing cylinder of water: (a)experiment[18] (b) 50X150 cells

(c) 100X300 cells (d) 150x450 cells

The surface reconstruction method can be very well tested with dam break problems, however, this paper simulated the collapsing cylinder of water as Munz and Maschek[5] performed. Fig. 4 shows the images of their experiment: they released a cylindrical column of water of diameter 110mm and height 200mm in a perspex pot with diameter 444mm. Due to the gravity, the water collapse suddenly and spread radially on the flat bottom to the side wall of the pot, where it slosh upwards, fall back and collapse back to the center create a jet flow. The simulation using 2D-axisymmetric mesh with 50X150, 100X300 and 150X450 cells respectively. Table 1 compares the characteristic times and heights of the experiment with the simulations. The simulation can reconstruct the main behaviors of the flow, the characteristic times and heights are in good agreement with experiment.

Table 1. Characteristic times and heights for the collapsing cylinder of water

	Time of arrival at the sidewall of the pot, $t_1(s)$	Time of maximum sloshing height at the wall, $t_2(s)$	Maximum sloshing height, $h_2(mm)$	Time of maximum collapse height, $t_3(s)$	Maximum collapse height, $h_3(mm)$
Experiment[18]	0.20±0.02	0.42±0.02	160±10	0.88±0.04	400±50
50X150 cells	0.21	0.36	110	0.86	335
100X300 cells	0.21	0.38	151	0.91	425
150X450 cells	0.20	0.39	183	0.95	455

Conclusions

In this paper, a two-dimensional numerical model for incompressible multiphase flow is established and tested. The Navier-Stokes equations are discretised by the finite volume method with a staggered-mesh layout. IDEAL algorithm is implemented for solving the Navier-Stokes equations. PLIC-VOF method is used to track the free surface. Simulations of a collapsing cylinder of water agree well with experiment. The simulations show a sensitivity to the mesh size, which need further research and improvement.

Acknowledgement

The authors are grateful for the support of the National Natural Science Foundation of China (Grant Nos. 51678045, 51579052).

References

[1] C.W. Hirt, B.D. Nichols, Volume of fluid (VOF) method for the dynamics of free boundaries. Journal of computational physics. 39 (1981) 201-225.

[2] D.L. Youngs, Time-dependent multi-material flow with large fluid distortion. Numerical methods for fluid dynamics. 24 (1982) 273-285.

[3] D. Sun, Z. Qu, Y. He, et al., An efficient segregated algorithm for incompressible fluid flow and heat transfer problems—IDEAL (inner doubly iterative efficient algorithm for linked equations) Part I: Mathematical formulation and solution procedure. Numerical Heat Transfer, Part B: Fundamentals. 53 (2008) 1-17.

[4] D. Kothe, W. Rider, S. Mosso, et al., Volume tracking of interfaces having surface tension in two and three dimensions. AIAA paper. 96 (1996) 1-18.

[5] C.-D. Munz, W. Maschek, Comparison of results of two-phase fluid dynamics codes and sloshing experiments. KERNFORSCHUNGSZENTRUM KARLSRUHE-PUBLICATIONS-KFK. (1992).