13th Australasian Fluid Mechanics Conference
Monash University, Melbourne, Australia
13-18 December 1998

Computing Flows with Moving Interfaces: Application in Mould Filling

Zili Zhu and Nick Stokes

CSIRO Mathematical & Information Sciences
Clayton, Victoria, AUSTRALIA

ABSTRACT

We will present a method for predicting fluid flows with gas and liquid phases separated by moving interfaces. The momentum and continuity equations of both the gaseous and the liquid phases are coupled together. The governing equations for fluid of such large density difference ratios are solved through an artificial-compressibility based algorithm. For tracking the moving interfaces, we rely on Eulerian approach by advecting the interface or interfaces on a fixed mesh. The treatment of both the liquid and gas together is an approach that can naturally extend to the simulation of heat transfer.

INTRODUCTION

Flows with moving internal interfaces can be found in many engineering applications. Mould-filling process is a typical case of such fluid flows, where one phase is liquid metal, the other phase is gas, and the two phases are separated by moving interfaces (or fronts). The computation of the gaseous and liquid phases together is important, in terms of not only fluid mechanics but also heat transfer in mould-filling processes. The difficult tasks in predicting liquid-gas flows with interfaces are: 1). predicting the locations of moving interfaces, 2). solving fluid flows with extremely large density difference ratio.

In this paper, the liquid is treated as incompressible flow, while the gaseous phase can be either incompressible or an ideal gas. Navier-Stokes equations for both the liquid and the gas are solved together at every time step. The mass continuity equation is used to compute pressure in a typical approach of artificial-compressibility algorithm. As the density difference is extremely large between the gas and liquid phases, the solver for the Navier-Stokes momentum equation needs to be very robust. In the current numerical algorithm, we couple together the momentum equation in each direction with an equation that is derived from the continuity equation. As the momentum equation is solved separately in each direction, this numerical scheme can be extended to three dimensional cases without any modifications and without large requirement for computer memory.

For tracking the moving interfaces in the fluid, we rely on Eulerian approach by advecting the interface or interfaces on a fixed mesh. This technique avoids the difficulties often associated with the Lagrangian approach in which re-meshing still has to be performed when the interfaces become very complex geometrically. In the Eulerian approach, we need only to solve the transport equation of a scaler function F. Since only the region near the interfaces is important, the advection equation for this function needs to be solved accurately only in the narrow region surrounding the interfaces, while in the area far away from the interfaces, the scalar function does not need to be solved accurately. Such a concept is referred to as cursoring (Medale and Jaeger, 1997).

The entire algorithm is implemented in finite-element formulation, and the numerical scheme is coded through the PDE package *Fastflo*.

GOVERNING EQUATIONS

In this paper, the mesh grid is fixed and is independent from the moving interfaces, and that the moving interfaces (boundaries) will be transported by the velocity of the fluid flow. Within the region of each phase enclosed by the interfaces, the governing equations are traditional continuity and momentum equations for this phase. For the gaseous phase, ideal gas law is the equation of state. The governing equations for each phase are not solved separately. The governing equations of all phases in different regions of the fixed flow domain will be coupled and solved together. There is no need for specifying boundary conditions along the moving interfaces, as the stress tensors are in balance across the interfaces when surface tension is not considered in the present formulation.

The non-dimensional mass conservation equations in the liquid regions are:

$$\rho_l = const \tag{1}$$

$$\nabla \cdot (\rho_l \mathbf{V}_l) = 0 \tag{2}$$

The mass conservation equations in the gaseous regions are:

$$\rho_g = \frac{U_0^2}{RT} P \tag{3}$$

$$\frac{\partial \rho_g}{\partial t} + \nabla \cdot (\rho_g \mathbf{V}_g) = 0 \tag{4}$$

where l and g indicate the liquid and gaseous phases of the fluid respectively. The vector velocity $\mathbf{V}$ has its component U_i in the i coordinate direction. ρ_k is the density of the kth phase. P is non-dimensional pressure in the form of $P = p/(\rho_0 U_0^2)$, while U_0 is the reference velocity and ρ_0 is the reference density.

The momentum equation in the liquid region is written as:

$$\rho_l \frac{\partial \mathbf{V}_l}{\partial t} + (\rho_l \mathbf{V}_l \cdot \nabla)\mathbf{V}_l + \nabla P - \nabla \cdot [\mu_t^l (\nabla \mathbf{V}_l + \nabla \mathbf{V}_l^T)] = 0 \tag{5}$$

The momentum equation in the gaseous regions is:

$$\rho_g \frac{\partial \mathbf{V}_g}{\partial t} + (\rho_g \mathbf{V}_g \cdot \nabla)\mathbf{V}_g + \nabla P - \nabla \cdot [\mu_t^g (\nabla \mathbf{V}_g + \nabla \mathbf{V}_g^T)] = 0 \tag{6}$$

μ_t^g is the viscosity of the gaseous phase, while μ_t^l is the viscosity of the liquid phase. If turbulence is present, μ_t can be calculated through a $k - \epsilon$ turbulence model.

The momentum equations (5) for the liquid region(s) will not be solved separately from the momentum equations (6) for the gaseous region(s). Equations (5) and (6) are solved together as if they are a single momentum equation of which the density is dependent on the liquid region(s) and gaseous region(s).

The locations of moving interfaces are determined through a scalar F. F is transported in an Eulerian approach on the fixed mesh which covers both gaseous and liquid regions. Scalar F varies from -1 in the gaseous regions to $+1$ in the liquid regions. Within a band area around the moving interfaces, F changes linearly from the gaseous to the liquid regions.

The transport equation for F is:

$$\frac{\partial F}{\partial t} + \mathbf{V} \cdot \nabla F + \nabla \cdot (D_m \nabla F) = 0 \tag{7}$$

where D_m is the molecule diffusivity of the fluid. In fact, for stability, D_m can be set large in areas far away from the moving interfaces. This will not influence the accuracy of predicting the moving fronts, as the transport equation only needs to be accurately solved within a narrow band covering the fronts.

NUMERICAL FORMULATIONS

In the current numerical scheme, there is only one velocity, one density and one pressure at each nodal point of the discretized flow domain. The scalar F at each nodal point is used to determine if this nodal point is in gaseous or liquid regions. When $F > F_c$, this nodal point is regarded as liquid, when $F < F_c$, this nodal point is regarded as gaseous. F_c is normally set to 0, however, F_c can be altered during solution process to adjust for total mass conservation.

In the gaseous region(s) of the flow domain, the continuity equation is:

$$\rho_g = \frac{U_0^2}{RT} P$$

$$\frac{\partial P}{\partial \tau} = \frac{RT}{U_0^2}\frac{\partial \rho}{\partial \tau} = -\frac{RT}{U_0^2} \nabla \cdot \rho_g \mathbf{V}$$

where τ is pseudo time. In the liquid region(s) of the flow domain, using artificial-compressibility approach, we can write the continuity equation as:

$$\rho_l = constant$$

$$\frac{\partial P}{\partial \tau} = \beta \frac{\partial \rho}{\partial \tau} = -\beta \nabla \cdot \rho_l \mathbf{V}$$

The continuity equation for both gas and liquid can therefore take a unified form of:

$$\frac{1}{\rho}\delta P - \nabla \cdot (\frac{\beta d\tau^2}{2\rho} \nabla \delta P) = -\beta d\tau \nabla \cdot \mathbf{V} + \nabla \cdot (\frac{\beta d\tau^2}{2\rho} \nabla P) + \nabla \cdot (\frac{\beta d\tau^2}{2} \mathbf{V} \cdot \nabla \mathbf{V}) \tag{8}$$

where β changes its value depending on in the gaseous or liquid regions.

To obtain velocity of gas or liquid phases, we solve the momentum equation in each coordinate direction one by one. However, due to the large density difference between the gas and liquid phases, for stability purpose, the momentum equation in each direction needs to be coupled with an equation that is derived from the unified continuity equation (8). The resulting momentum equation system in the i direction for both the gaseous and liquid phases is expressed in a unified form as:

$$\rho \frac{\partial U_i}{\partial t} + (\rho \mathbf{V} \cdot \nabla)U_i + \frac{\partial P}{\partial x_i} - \nabla \cdot [\mu_t \nabla U_i] = 0$$

$$\frac{\partial U_i}{\partial x_i} - \nabla \cdot (\frac{d\tau}{2\rho} \nabla \delta P) = -\nabla \cdot \mathbf{V}$$

For the transport equation of scalar F, we use MacCormack two-step predictor-corrector scheme.

SOME COMPUTATIONAL RESULTS:

Case 1: Broken Dam Problem

The numerical method will be tested by comparing predicted results of this method with experimental data of a broken-dam problem (Martin and Moyce, 1952).

The broken dam case is an interesting problem to solve, as there are experiment data to compare with, and in addition, the influence of the gas (air) on the liquid (water) can be studied carefully. There are also results from other numerical methods reported in the literature (Hirt and Nichols, 1981, Lewis et al, 1997).

In our current computation, a water column of $0.05715m$ wide and $0.05715m$ high is allowed to collapse within a rectangular cavity of $0.17145m$ wide and $0.07m$ in height. No-slip boundary conditions are set up for the three solid walls: the bottom section and two side walls. For the top section of the rectangular cavity, free-slip boundary conditions are used for the air velocity. The liquid is water with density $\rho_l = 1000kg/m^3$, and the gas is air of density $\rho_g = 1kg/m^3$. We use normal gravity force with $g = 9.8m/s^2$. The viscosity of air is $\mu_g = 1.31 \times 10^{-3}Ns/m^2$, and the viscosity of water is $\mu_l = 1.79 \times 10^{-5}Ns/m^2$.

Figure 1 shows the comparison between the numerical results and experimental data. The horizontal axis is the time in $t\sqrt{(g/H)}$ and the vertical axis is the position of water front X/H, here $H = 0.05715m$. This figure shows that the current numerical method can well predict the front location within a 7% accuracy.

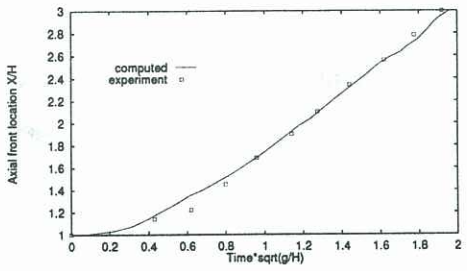
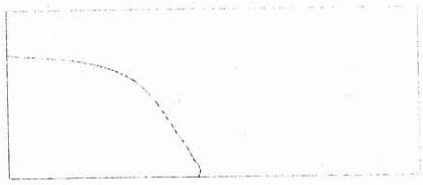


Figure 1: Moving water front position.

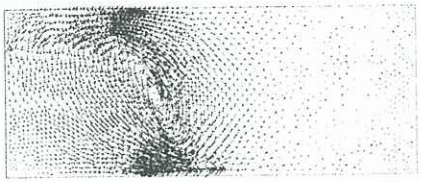
Figure 2 illustrates the liquid free-surface positions and corresponding velocity vectors of both air and water. From this figure, we can see that the moving water free-surface is realistically predicted.

Case 2: Mould Filling

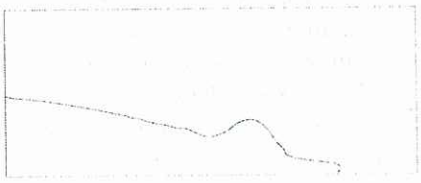
We use the current method to predict a typical mould-filling process as shown here in Figure 3.



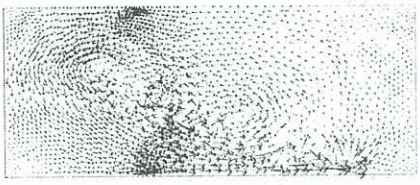
The moving front of liquid at Time $= 0.045s$



Velocity vectors of liquid and gas at Time $= 0.045s$



The moving front of liquid at Time $= 0.1s$



Velocity vectors of liquid and gas at Time $= 0.1s$

Figure 2: Broken-Dam problem.

The liquid velocity at the inlet to the mould is $U_0 = 1m/s$, the length of the rectangular mould is $L_0 = 0.3m$ while at the mouth to the rectangular mould, velocity magnitude becomes more than three times larger than the incoming velocity. In the current computation, the incoming liquid has a density of $\rho = 1000kg/m^3$, while the density of gas is $\rho = 1kg/m^3$. The gravity acceleration rate is still set to $g = 9.8m/s^2$. For stability, ten times of the viscosities of air and water are used in this case: i.e. $\mu_g = 2.0 \times 10^{-4}Ns/m^2$, $\mu_l = 1.0 \times 10^{-2}Ns/m^2$.

As shown in Figure 3, the moving liquid front in the rectangular mould filled with air is realistically simulated. This figure also illustrates the velocity vectors of both the air and water at two different time interval during the filling process. It takes more than 0.6 second for the water to reach the mould air vent.

CONCLUSION

In this paper, we have presented a numerical

scheme for simulating flows with internal moving interfaces. The flows can be a mixture of gas and liquid separated by internal interfaces moving in a fixed mesh. The moving fronts are tracked through the transportation of a scaler. The gaseous and liquid phases are computed as if they are of the same fluid, except that the equation of state takes different forms with different density and viscosity in the gaseous and liquid regions.

We have used the current numerical algorithm to simulate a broken-dam problem, and the computed results have been compared with experimental data. The comparison has demonstrated that the present method can satisfactorily predict flows with moving inter-facial fronts. We have also used the numerical scheme to simulate a typical mould-filling process. The complex mould-filling process can be successfully simulated by the present algorithm, and this test case has shown the ability of the current numerical method to predict flows with complex moving inter-facial fronts.

ACKNOWLEDGEMENT

This research work is carried out as part of the development of the general finite-element PDE package *Fastflo*.

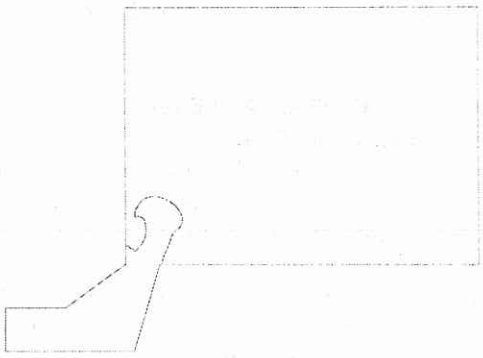
REFERENCES

Martin, J.C. and Moyce, W.J., "An Experimental Study of the Collapse of Liquid Columns on a Rigid Horizontal Plane", **em Phil. Tans. R. Soc. Lond.** A, **244**, 312-324, 1952.

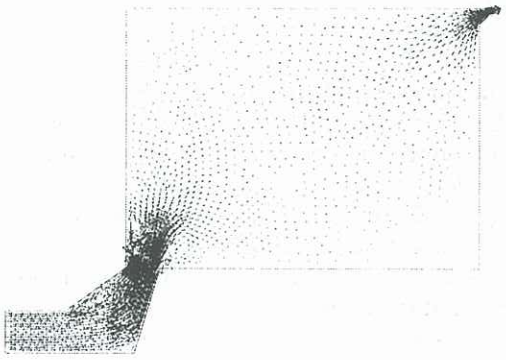
Medale, M. and Jaeger, M., "Numerical Simulation of Incompressible Flows with Moving Interfaces", **Int. J. Num. Methods in Fluids**, Vol.24, 615-638, 1997.

Hirt, C.W. and Nichols, B.D. "Volume of Fluid (VOF) Method for the Dynamics of Free Boundaries", **J. Comut. Phys.**, Vol.39, 201-225, 1981.

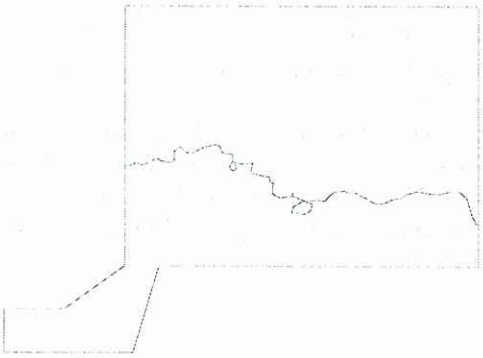
Lewis, R.W., Navti, S.E. and Taylor, C., "A Mixed Lagrangian-Eulerian Approach to Modelling Fluid Flow during Mould Filling", **Int. J. Num. Methods in Fluids**, Vol.25, 931-952, 1997.



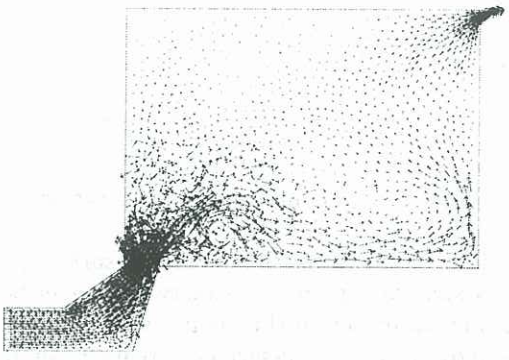
The moving front of liquid at Time = $0.09s$



Velocity vectors of liquid and gas at Time = $0.09s$



The moving front of liquid at Time = $0.34s$



Velocity vectors of liquid and gas at Time = $0.34s$

Figure 3: Mould filling problem