Fifth International Conference on CFD in the Process Industries
CSIRO, Melbourne, Australia
13-15 December 2006

SIMULATION OF INTERFACIAL FLOWS USING FRONT TRACKING APPROACH

Manasa Ranjan Behera[1] and Murali. K[2]

[1] Department of Ocean Engineering, Indian Institute of Technology Madras, Chennai – 600036, INDIA

[2] Department of Ocean Engineering, Indian Institute of Technology Madras, Chennai – 600036, INDIA

ABSTRACT

The interfacial flow simulations using a robust interface-tracking method are presented. The method is based on one fluid formulation, where a single set of governing equations for the whole computational domain with variable material properties are solved. Interfacial terms, at the boundary separating the phases, are accounted for by adding the appropriate sources as δ functions. The method has been implemented for Liquid-Air interfacial flow problems, depicting the interface and topology change capturing capability. The representation of the moving interface and its dynamic restructuring, as well as the transfer of information between the moving front and the fixed grid, is discussed. This method has been applied to density stratified flows, and interfacial movements are then presented. An experimental study on salt wedge movement, conducted in the University of Dundee, has been simulated using the above algorithm. The numerical results are compared with the experimental results available in the literature.

NOMENCLATURE

D	Distribution function
F_{st}	Surface tension force
g	Gravitational acceleration
G	Gradient function
h	Grid size
I	Indicator function
k	Curvature
$\mathbf{n}$	Unit normal
p	Pressure
r	Distance between front and fixed grid points
t	Time in seconds
t^*	Non-dimensional time
$\mathbf{u}$	Velocity vector in the fixed grid
u_f	Front velocity
x^*	Non-dimensional wedge displacement
X	Position vector
ρ_g	Density of gas
ρ_l	Density of liquid
μ	Viscosity
σ	Surface tension
Ψ	Stream function
$\delta(\mathbf{x}-\mathbf{x}_f)$	Delta function

INTRODUCTION

Ocean and marine engineers, recently, started applying Computational Fluid Dynamics (CFD) to the problems in their fields. The main reason for this lag is the presence of free surface and other multi-fluid systems (MFSs) in the field of Ocean Engineering such as bubbles, density stratification, etc. Free surface flows feature most prominently in the marine environment and are characterised by wind-water interactions and unsteady waves. Density stratification in lakes, estuaries and in the oceans leads to circulation and mixing of large volumes of water resulting in multiphase flow. Modeling of these processes is of great importance for conservation of the ecological balance of these systems.

The multi-fluid systems could be numerically modeled with a single set of governing equations in the integral form (Navier-Stokes equations and mass conservation equation), as in the case of single-phase flow. However, straightforward application of these conservative schemes is severely limited when a sharp interface is present, because the fluids' interface leads to sharp discontinuities of the fluid properties, such as density and viscosity. In addition, the existence of surface tension on the interface would induce a pressure jump across the interface as well. To overcome these obstacles in tracking the sharp interface for the multiphase flows, various numerical methods, such as VOF method (Hirt and Nichols 1981), level set method (Son and Dhir 1998), front tracking method (Qian et al. 1998), and surface fitted method (Jacqmin 1999) have been proposed. Front tracking method provides clear geometrical information of the interface and can workout for large topology changes of the interacting interfaces.

In the present work, the state of the art of front tracking method, a hybrid approach of the front capturing and front tracking technique proposed by Tryggvason et al. (1998), is considered. In the front tracking method, a stationary, fixed grid is used for the fluid flow, and a set of adaptive elements on the front is used to mark the interface. The fluid properties, such as density and viscosity, are updated based on the position of the interface. Interfacial source terms, such as surface tension, are computed on the front and transferred to the flow solver grid by using a δ (dirac-delta) function on the interface between the phases. The unsteady Navier-Stokes equations are solved by a commercial flow solver FLUENT® on the structured grids. The interface or front is tracked explicitly by extracting the advection velocity from the flow solver grid. The advection of fluid properties is achieved following the motion of the interface or front. Hence, this method could evade numerical diffusion, and capture sharp interfaces and topology changes.

THEORY AND NUMERICAL ALGORITHM

Governing Equation

In this study, it is reasonable to treat both phases incompressible. Hence, the mass conservation on the whole domain (both fluid phases and the interface) may be expressed as,

$$\nabla \cdot u = 0 \tag{1}$$

The Navier-Stokes equations, governing the momentum balance in each fluid domain, including the interface, may be expressed as,

$$\frac{\partial \rho u}{\partial t} + \nabla \cdot (\rho u u) = -\nabla p + \nabla . \mu \nabla u + \sigma k n \delta(x - x_f) + (\rho_g - \rho_l) g \tag{2}$$

where, $\delta(x - x_f)$ is a delta function that is zero everywhere except at the interface, i.e, $x = x_f$; g is the gravitational acceleration, and subscript f refers the front. ρ_g and ρ_l refers to the density of gas and liquid phase respectively. The unsteady Navier-stokes equations were solved numerically using a fluid dynamics solver (FLUENT®).

Discontinuity Treatment on the Front

Solution of the Navier-Stokes equations, with the interface discontinuity, requires the fluid properties (density and viscosity) to be reconstructed for the whole solution domain. Although the density and viscosity of each fluid are constant, their abrupt jump across an interface may lead to either excessive numerical diffusion or numerical instability. The novelty of the method proposed by Tryggvason et al. (1998) is that the front is considered of having a finite thickness of the order of mesh size instead of zero thickness. In the transition zone, the fluid properties change smoothly / continuously from the value on one side of the interface to that on the other side. Hence, the material property distribution over whole domain may be reconstructed by using an indicator function $I(x,t)$, which has the value of one in the gas phase and zero in the liquid phase at any time t .

$$b(x,t) = b_l + (b_g - b_l) I(x,t) \tag{3}$$

in which b stands for either fluid density or viscosity. The subscripts *g* and *l* respectively refer to gaseous phase and liquid phase. The indicator function can be written in terms of an integral over the whole domain $\Omega(t)$, bounded by the phase interface $\Gamma(t)$,

$$I(x,t) = \int_{\Omega(t)} \delta(x - x')dv' \tag{4}$$

where $\delta(x - x')$ is a delta function that is one only where $x' = x$ and zero otherwise. The indicator function $I(x,t)$ can be reconstructed by solving Poisson equation.

$$\nabla^2 I = \nabla \cdot \int_{\Gamma(t)} n\delta(x - x')ds \tag{5}$$

To consider the small artificial thickness of the front, a distribution function, $D(x)$, is used to approximate the delta function. Thus, the sharp jump of fluid properties over the interface is to be distributed among the nearby grids. A gradient field $(G = \nabla I)$ is defined and the discretised form of the gradient function G is given as follows,

$$G(x) = \sum_f D(x - x_f) n_f \Delta s_f \tag{6}$$

where n_f is the normal unit vector to an interface element of area Δs_f whose centroid is at x_f . In this study the distribution function suggested by Tryggvason et al.(2001) can be used for a two-dimensional grid system,

$$D(x - x^{(l)}) = \begin{cases} (h - |r|)/h \ldots\ldots\ldots\ldots |r| < h \\ 0 \ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots |r| \geq h \end{cases} \tag{7}$$

where h is the grid size. Using the same approach, the surface tension on the front can also be easily distributed to the fixed grid, thus

$$F_{st}(x) = \sum_f D(x - x_f) \sigma k n_f \tag{8}$$

Front Tracking

Since the fluid velocity is updated on the fixed grid, the velocity on the moving front should be computed by interpolating from the fixed grid so that the front would move at the same velocity as that of the surrounding fluids.

$$u_f = \sum D(x_f - x) u(x) \tag{9}$$

Then, the front is advected normal to itself in a Lagrangian fashion,

$$x_f^{n+1} - x_f^{n} = \Delta t \; u_f \cdot n \tag{10}$$

Once the front position is updated, the property of marker elements on the interface may have also been modified. The element resolution on the interface has a strong effect on the information exchange between the front and the fixed grids (shown in fig. 1), and it further affects the accuracy of the simulation results. Therefore, the elements on the front should be adapted to keep a good geometric resolution.

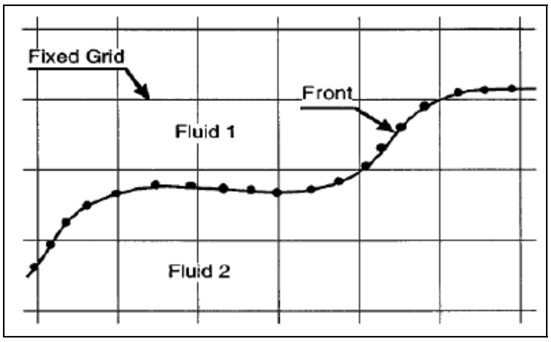

Figure 1: The governing equations are solved on a fixed grid but the phase boundary is represented by a moving "front," consisting of connected marker points.

Solution Procedure

Here the solution of the fluid flow is carried out using a commercial fluid flow solver FLUENT®. The external solver FLUENT® is used only to invoke the continuity and momentum equations to solve the flow domain at each time step. The advection of the interface from old position to the new position and distribution of the material properties over the domain with respect to the new interface position is done using the algorithm described above in sections 2.3 and 2.4. The flow of the solution process is given in the following steps.

(1) Using the fluid velocity field (u^n) and the interface (x_f^n), the moving velocity of the front marker points (u_f^n) is computed according to equation (9).

(2) Using the estimated interface velocity, the front is advected to the new position (x_f^{n+1}). Subsequently, the elements representing the front are examined for the adaptation and topology change.

(3) At the new interface positions, the redistribution of the interface property is performed with the reconstructed indicator function $I(x_f^{n+1})$. Hence, the new fluid property field, such as density (ρ^{n+1}), viscosity (μ^{n+1}) as well as the surface tension (F_{st}^{n+1}), is obtained.

(4) With appropriate boundary conditions, the momentum balance equation and continuity equation are solved by FLUENT®. By now, the fluid velocity (u^{n+1}) and pressure (p^{n+1}) would have been updated.

(5) The steps from (1) to (4) are repeated for the next time step prediction with the updated values till the convergence.

The steps 1 to 3 are performed using the present algorithm described above and then the updated data for the whole domain is fed into the FLUENT® solver to carry out the step 4.

RESULTS

The front tracking method is applied to two phase flows simulations to demonstrate its capacity to capture the moving interface position and the topology change.

Solenoidal Velocity Field

The flow simulated by Theodorakakos and Bergeles (2004) is considered for validation of the front tracking code. In this case, a square computational domain of 1m×1m is taken and a predefined velocity pattern is considered for the flow, which contains shear. An air bubble of radius 0.15m is centered at a location x = 0.50, y = 0.75 in the liquid (water) computational domain. A vertex is formulated with a predefined velocity field obtained by the stream function (ψ). Where the stream function is given by

$$\Psi = \frac{1}{\pi}\sin^2(\pi x)\sin^2(\pi y) \qquad (11)$$

When the bubble is placed in this solenoidal flow field, it stretches and spirals about the center of the domain, which is a stagnation. The flow is simulated for various times to get the transient air-water interface position (modified shape) of the bubble.

The results are compared in fig. 2. The simulated results shown in fig. 2(a) are in excellent comparison with the profile of Theodorakakos and Bergeles (2004) (Fig. 2(b)). In the present study, the simulations have been carried out for more time to check the stability of the front as it becomes thinner.

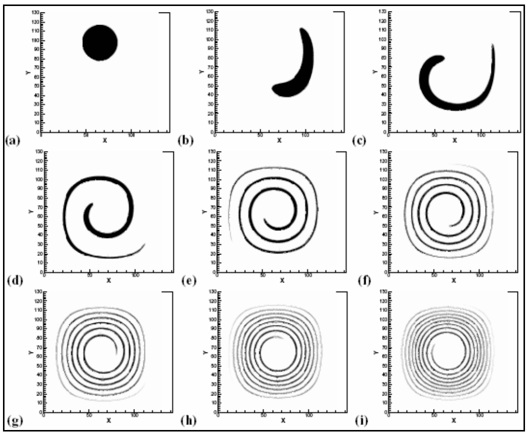

Figure 2(a): Present simulation for single vortex flow field case, for (a) t = 0s; (b) t = 0.5s; (c) t = 1s; (d) t = 2s; (e) t = 4s; (f) t = 6s; (g) t = 8s; (h) t = 10s; and (i) t=12s.

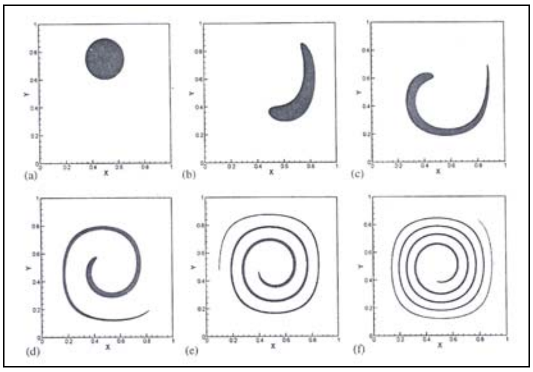

Figure 2(b): Simulation plots given by Theodorakakos and Bergeles (2004) for single vortex flow field case, for (a) t = 0.0s; (b) t = 0.5s; (c) t = 1.0s; (d) t = 2.0s; (e) t = 4.0s (f) t = 6.0s.

Density Stratified Flow

The interface tracking scheme developed in the present study has been applied to salt wedge problem based on the physical model studies conducted at the University of Dundee (Coates and Guo, 2003). A schematic of the physical model set-up of Coates and Guo (2003) is shown in Fig. 3. The transient displacement of the nose of the salt wedge due to an oncoming fresh water inflow has been simulated with the present interface tracking code. The physical model experiments are numerically simulated in the present study considering the flow to be two-dimensional.

In the numerical model (shown in Fig. 4), the whole domain is considered to be filled with three different fluids – the saline water, freshwater and air. Hence, there are two interfaces in the domain; one between the salt water and fresh water and the other between the freshwater and air. These three fluids with two interfaces make the system a three phase flow. The bed of the channel and the weir are simulated as solid boundary. The top boundary, above air, is simulated as atmospheric boundary, which provides the free surface condition for the freshwater flow. All the physical model setup conditions are reproduced here for numerical simulation.

The utility of the computational code has been demonstrated by comparing the numerical results with the experimental results and semi empirical results (Behera and Murali). This comparison is shown in Fig. 5. It is seen that the numerical results closely follow the experimental trend of the non-dimensional wedge displacement (x^*) with respect to non-dimensional time (t^*). The numerical experiments demonstrated that the present numerical scheme could be implemented for simulation of realistic multi-fluid systems.

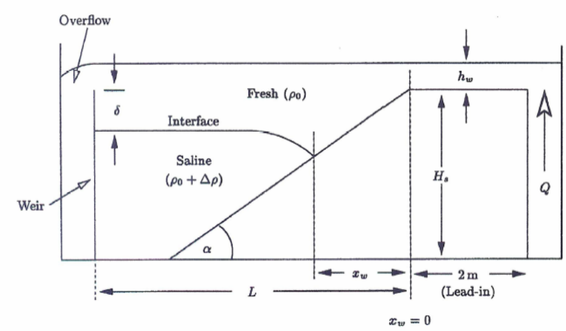

Figure 3: The sketch of the physical model defining the various model parameters. (Coates and Guo, 2003).

CONCLUSIONS

The front tracking method is used to model multiphase/interfacial flow phenomena. The studies indicate that this method can handle the sharp interface between different fluids and sufficiently robust to handle large topology changes as illustrated in the solenoidal velocity field problem. The physical model studies conducted, by Coates and Guo (2003), to analyse the salt wedge dynamics has been reproduced and numerically simulated to show the usefulness of the present scheme. The numerical results are compared with the semi empirical and experimental results, showing the usefulness of the model and accuracy of the numerical scheme. The study demonstrated that the scheme can be well applicable to various practical multiphase flow problems, with further improvement of the simulation tool.

REFERENCES

BEHERA, M. R., (2005), "Simulation of multi-fluid systems (MFS) using interface tracking techniques." *M.S Thesis*, Indian Institute of Technology Madras, India.

BEHERA, M. R., and Murali, K., (In Press), An improved semi-empirical model for displacement of salt water wedge in a bar-blocked estuary, *ASCE, Journal of Waterway, Port, Coastal, & Ocean Engineering*.

COATES, M. J. and GUO, Y., (2003), "The salt wedge position in a bar-blocked estuary subject to pulsed inflows", *Estuarine Coastal and Shelf Science,* **58**, 187-196.

COATES, M. J., GUO, Y., and DAVIES, P. A., (2001), "Laboratory model studies of flushing of trapped saltwater from a blocked tidal estuary." *IAHR Journal of Hydraulic Research* **39(6)**, 601–609.

DEBLER, W., and IMBERGER, J., (1996), "Flushing criteria in estuarine and laboratory experiments.", *Journal of Hydraulic Engineering ASCE,* **122(12)**, 728–734.

HIRT, C.W. and NICHOLS, B.D., (1981), "Volume of fluid (VOF) method for the dynamics of free boundaries" *J. Comput. Phys.* **39**, 201-225.

JACQMIN, D., (1999), "Calculation of two-phase Navier–Stokes flows using phase-field modeling", *J. Comput. Phys.* **155**, 96-127.

QIAN, J., TRYGGVASON, G. and LAW, C.K., (1998), "A front tracking method for the motion of premixed flames", *J. Comput. Phys.* **144**, 52-69.

SON, G. and DHIR, V.K., (1998), "Numerical simulation of film boiling near critical pressures with a level set method", *J. Heat Transfer*, **120**, 183-192.

THEODORAKAKOS, A. and BERGELES, G., (2004), "Simulation of sharp gas-liquid interface using VOF method and adaptive grid local refinement around the interface", *International Journal for Numerical Methods in Fluids*. **45**, 421-439.

TRYGGVASON, G., BUNNER, B., EBRAT, O. and TAUBER, W., (1998), "Computations of MultiPhase Flows by A Finite Difference/Front Tracking Method. I. Multi-fluid flows", *Von Karman lectures notes*, The Von Karman Institute.

TRYGGVASON, G., BUNNER, B., ESMAEELI, A., JURIC, D., AL-RAWAHI, N., TAUBER, W., HAN, J., NAS, S. and JAN, Y.-J., (2001), "A Front-Tracking Method for the Computations of Multiphase Flow", *J. Comput. Phys.*, **169**, 708-759.

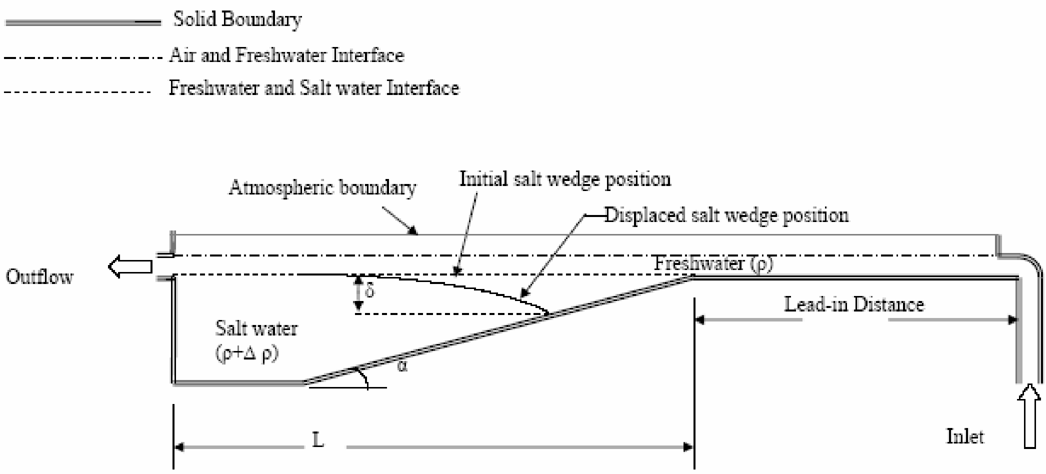

Figure 4: Schematic diagram of the physical model for numerical study (not to scale)

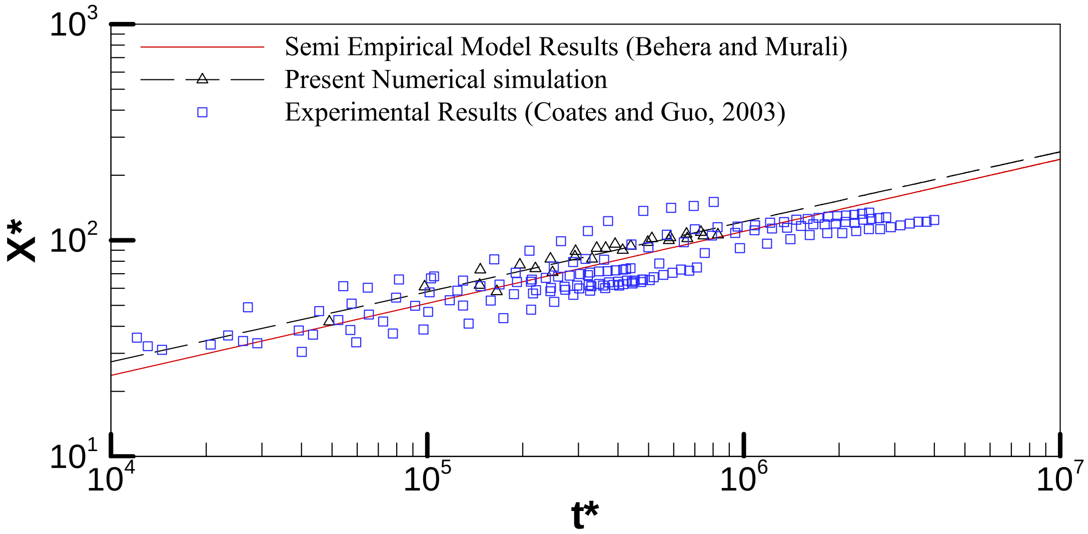

Figure 5: Comparison of semi empirical, numerical and experimental(Coates and Guo, 2003) results for non-dimensional wedge displacement (x*) with respect to non-dimensional time (t*).