Level set method for incompressible two phase flow

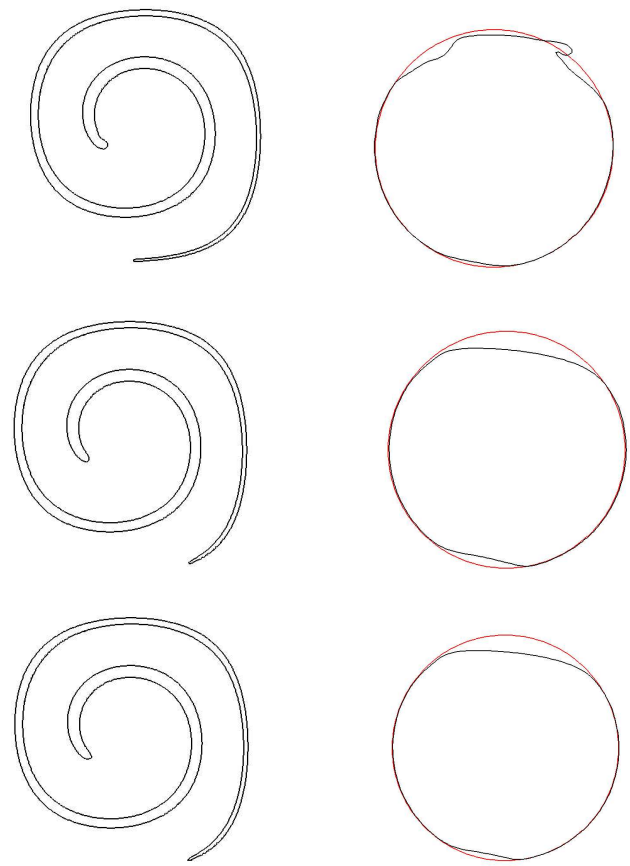
Christos N. Kavouklis, kchris@t7.lanl.gov
Hyung Taek Ahn, htahn@lanl.gov
Mikhail Shashkov, shashkov@lanl.gov

A recent trend in CFD is the study of flows involving more than one fluid. Of particular importance in this kind of flow is the correct representation of the interface among different fluids at each step of the simulation process. In this work, we are using the level set method to correctly locate the interface shared by two different fluids. In the level set methodology the fluid interface is represented as the zero level set of a smooth function that satisfies the following advection equation:

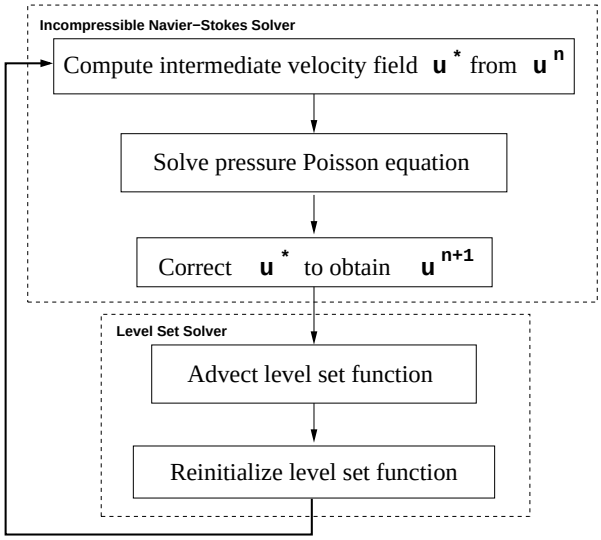
$$\phi_t + \mathbf{u} \cdot \nabla\phi = 0$$

where $\mathbf{u}$ is the interface velocity [1]. The level set function is a signed distance (i.e. $\|\nabla\phi\| = 1$ and ϕ attains opposite signs on different fluids). This allows for straightforward computation of geometric and physical quantities of interest, such as normal vector, interface curvature and surface tension. The most attractive feature of the method is that the interface does not need to be tracked explicitly and that topological changes such as merging of interfaces or splitting of material domains in multiple components are easily handled. We also emphasize that the method is readily extendable to three dimensions.

Our starting point in this investigation is the implementation of a 2D pressure-correction finite difference incompressible *Navier-Stokes* solver. The computational grid we are using is the classical MAC grid for which pressure and the level set function are defined at element centers, while velocities are computed at element edges. The second part of this research is devoted to the application of our numerical scheme to two phase flow. The interface at each simulation step is evaluated according to the level set equation. Unfor-



Maximum stretch profiles (left column) and initial/final time profiles (right column) for the reversible vortex case with a 256^2 mesh. The computed profile is coloured with black, while the initial profile is red coloured. Central differencing for the level set equation (first row) results in an oscillatory solution although the symmetric difference between the initial and final profile is minimal. The ENO (second row) and WENO (third row) schemes yield a slightly larger symmetric difference, however they both exhibit substantially smoother profiles with WENO being more accurate.



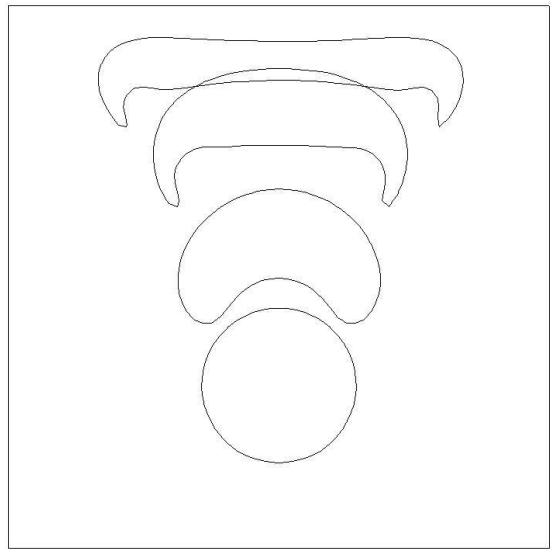
Flowchart of the coupled Navier-Stokes solver with the level set scheme.

tunately, ϕ no longer remains a signed distance after several timesteps. To remedy this deficiency, the following reinitialization equation [1] is introduced:

$$\phi_\tau = S(\phi_0)\,(1 - \|\nabla\phi\|)$$

where $S(\phi_0) = sign(\phi_0)$ and $\phi(\mathbf{x},t;0) = \phi_0(\mathbf{x})$. The latter is solved to steady-state to ensure that the signed distance property of the level set function is preserved. Temporal discretization of both the level set and reinitialization equation utilizes a standard second order Runge-Kutta scheme. To descritize spatial derivatives of the level set equation we are employing central, upwind, third order ENO and fifth order WENO schemes [2]. For the reinitialization equation a second order ENO scheme has been used. We also note that discontinuous data across the fluids' interface, such as density, viscosity and the Heaviside and Dirac delta functions are mollified to obtain non-oscillatory solutions.

Several representative case studies are included. Specifically, we consider the reversible vortex problem, for which the velocity field is known *a priori*, to demonstrate the efficacy of the method in resolving thin filament structures. Finally, the coupled code has been utilized to simulate motion of an air bubble in a fluid subject to a gravitational field.



Evolution of a rising gas bubble in a fluid. The bubble shapes at non-dimensional time $t = 0, 1.5, 3.0, 4.5$ are depicted. The Reynolds number for this case is $Re = 10,000$ and the gas to fluid density and viscosity ratios are $\rho_g/\rho_f = 0.2$, $\mu_g/\mu_f = 0.015$, respectively. The computational domain is a unit square and a 32^2 mesh is employed.

Acknowledgements

Los Alamos Report LA-UR-07-6963. Funded by the Department of Energy at Los Alamos National Laboratory under contracts DE-AC52-06NA25396 and the DOE Office of Science Advanced Computing Research (ASCR) program in Applied Mathematical Sciences.

References

[1] M. Sussman, P. Smereka, S. Osher, A level set approach for computing solutions to incompressible two-phase flow, Journal of Computational Physics **114**, 146-159 (1994).

[2] G. Jiang, D. Peng, Weighted ENO schemes for Hamilton-Jacobi equations, SIAM Journal of Scientific Computing **21**, 2126-2143 (2000).