9th ANKARA INTERNATIONAL AEROSPACE CONFERENCE **AIAC-2017-135**
20-22 September 2017 - METU, Ankara TURKEY

AN ARBITRARY LAGRANGIAN-EULERIAN (ALE) FRAMEWORK WITH EXACT MASS CONSERVATION FOR MULTIPHASE FLUID FLOWS

Cagatay Guventurk * and Mehmet Sahin†
Istanbul Technical University
Istanbul, Turkey

ABSTRACT

An arbitrary Lagrangian Eulerian (ALE) framework is presented to solve incompressible multiphase fluid flow problems with exact mass conservation. The incompressible Navier-Stokes equations are discretized using the stable side-centered unstructured finite volume method where the continuity equation is satisfied exactly within each element (div-stable discretization). The pressure field is treated to be discontinuous across the interface with the discontinuous treatment of density and viscosity. The surface tension term is considered as a force tangent to the interface and computed by utilizing cubic Legendre polynomials. For the application of the interface kinematic boundary condition, a special attention is given to satisfy both local and global discrete geometric conservation law (DGCL) in order to conserve the total mass of both species at machine precision. The mesh vertices are deformed due to the normal displacement of interface by solving the linear elasticity equations. The resulting algebraic equations are solved in a fully coupled manner and a one-level restricted additive Schwarz preconditioner with a block-incomplete factorization is utilized within each partitioned sub-domain. The method is validated by simulating a single rising bubble in a viscous fluid due to buoyancy in two-dimensions. The mass of the bubble is conserved at machine precision and discontinuous pressure field is obtained in order to avoid errors due to the incompressibility condition in the vicinity of the interface.

INTRODUCTION

Multiphase fluid flows are one of the important applications of moving boundary problems and they are often encountered in nature and industrial applications such as melting and solidification, fiber coating, targeted drug delivery, drop formation, food processing, glass and metal forming processes, etc. The interfacial dynamics plays an important role in these processes and determines the outcome. Nevertheless, the numerical simulation of multiphase flows still poses a major research challenge. One of the main difficulties of two immiscible fluids is that the material properties such as density, viscosity, etc. are discontinuous across the interface and the location of the interface is not a priori known. In addition, there is a need to take into account the surface tension. It should be noted that the pressure jump is not only a consequence of the surface tension, but also occurs due to the viscosity jump with non-zero normal derivative of the normal velocity component [Idelsohn et al., 2010].

*GRA in Mechanical Engineering Department, Email: cagatay.guventurk@gmail.com
†Assoc. Prof. in Astronautical Engineering Department, Email: msahin@itu.edu.tr

The early numerical methods to solve multiphase fluid flows are based on fixed Cartesian grids such as the Marker-and-Cell (MAC) method [Harlow and Welch, 1965], the volume of fluid (VOF) method [Hirt and Nichols, 1981], the level set (LS) method [Osher and Sethian, 1988] and the moment-of-fluid (MOF) method [Dyadechko and Shashkov, 2005]. In these approaches, the density and viscosity jumps and the surface tension at the interface are smoothed across the interface in order to avoid numerical instabilities near the interface [Sussman et al., 1994]. Therefore, the interface is no longer sharp, but has a finite thickness. However, this approximation smears out the sharp interface and is only about first order accurate. In addition, the smoothing must be over a few grid cells, resulting in a relatively fine mesh or dynamic adaptive mesh refinement (AMR). Furthermore, the large variations in the transport properties across the fluid-fluid interface lead to relatively stiff linear algebraic equations [Uzgoren et al., 2007]. In order to overcome the limitations of high mesh resolution within the finite element framework, eXtended Finite Element Method (XFEM) is employed that naturally allows to handle interfacial discontinuities by using discontinuous shape functions across the interface [Moës et al., 1999]. More recent numerical algorithms for multiphase fluid flows include the Lattice Boltzmann method (LBM)[Gunstensen et al., 1991], the smoothed particle hydrodynamics (SPH) [Monaghan, 1994], etc.

Another class of numerical methods for multiphase fluids is based on the arbitrary Lagrangian Eulerian (ALE) approach [Hirt et al., 1974]. In the ALE method, the mesh follows the interface between the two fluids and the governing equations are discretized on unstructured moving meshes. Therefore, the ALE formulation allows an accurate representation of surface forces like surface tension, membrane effects, etc. and the interface is sharply defined. However, the exact mass conservation for the ALE based multiphase fluid simulations is a rather difficul challenge to overcome. As far as the authors' knowledge go, we are not aware of any ALE algorithm with the exact mass conservation for incompressible multiphase fluid flows (see, for example, [Walkley et al., 2005; Perot and Nallapati, 2003]), which is critical for the accuracy of long-term numerical simulation of multiphase problems. In this paper, a novel ALE framework with exact mass conservation is proposed for the numerical simulation of multiphase flows. The incompressible Navier-Stokes equations are discretized using the stable side-centered unstructured finite volume method [Erzincanli and Sahin, 2013]. The continuity equation is satisfied within each element and the summation of the continuity equations can be exactly reduced to the domain/sub-domain boundary, which is important for the local and global mass conservation. In addition, a special attention is given to satisfy the both local and global discrete geometric conservation law (DGCL) at machine precision. Hence, the mass of the bubble and exterior fluid are conserved at machine precision, which is the main contribution of the current paper. The pressure field is also treated to be discontinuous across the interface with the discontinuous treatment of density and viscosity in order to avoid errors due to the incompressibility condition in the vicinity of the interface. The surface tension term at the interface is treated as a force tangent to the interface and computed by utilizing cubic Legendre polynomials [Tryggvason et al., 2001]. The displacement of mesh vertices within the fluid domain and the tangential displacement on the interface are solved by utilizing the linear elasticity equations. The resulting algebraic equations are solved in a fully coupled manner (monolithic approach) since the mesh deformation algorithm may lead to inadmissible small elements, which require an extremely small time step due to the CFL restriction. In this paper, the original system of equations is preconditioned with an upper triangular right preconditioner, which results in a scaled discrete Laplacian instead of a zero block in the original system due to the divergence-free constraint. Then, a one-level restricted additive Schwarz preconditioner with a block-incomplete factorization within each partitioned sub-domain is utilized for the resulting fully coupled system. The implementation of the preconditioned Krylov subspace algorithm, matrix-matrix multiplication and the restricted additive Schwarz preconditioner are carried out using the PETSc software package developed at the Argonne National Laboratories in order to improve the parallel performance. The computational domain is decomposed into a set of sub-domains using the METIS library.

NUMERICAL RESULTS

In this section, the ALE algorithm is applied to the two-dimensional [Hysing et al., 2009] rising bubble benchmark problem in order to assess the accuracy of the proposed approach. All the time dependent numerical calculations are carried out using the second-order backward difference (BDF2) with a time step of 0.0025 on the meshes M1-M4. The approximate mesh sizes are $1/40$, $1/80$, $1/160$ and $1/320$. Physical parameters and dimensionless numbers are taken from the work of Hysing et al. [2009] and the values are provided in Table 1. It should be mentioned that the surface tension force is computed by utilizing cubic Legendre polynomials [Tryggvason et al., 2001]. At the fluid-fluid interface, the discrete form of the local geometric conservation law (DGCL) is enforced for the kinematic boundary condition in the normal direction, which leads to an exact mass conservation when it is combined with the div-stable discretization. In the tangential direction, the linear elasticity equation is solved.

Table 1: Physical parameters and dimensionless numbers.

ρ_1	ρ_2	μ_1	μ_2	g	σ	Re	Eo	ρ_1/ρ_2	μ_1/μ_2
1000	100	10	1	0.98	24.5	35	10	10	10

For the present test case, a circular bubble with a radius of 0.25 is placed in a rectangular domain $[0,1]\times[0,2]$. Initially the bubble is stationary and then it is allowed to rise by the buoyancy force. The time evaluation of the bubble shape is provided on mesh M1 in Figure 1 with the mesh deformation up to t=3. The contours shows the velocity vector magnitude. The final bubble geometry on mesh M4 is compared with the results of TP2D and MooNMD codes provided in [Hysing et al., 2009] in Figure 2-[a] and the comparison shows very good agreement. In addition, the time variation of the bubble area is compared with the result of TP2D in Figure 2-[b]. Although the result of TP2D shows high frequency oscillations in the bubble area due to the employed level set method, the present approach exactly conserves the bubble area at machine precision, which is independent from mesh resolution and time step.

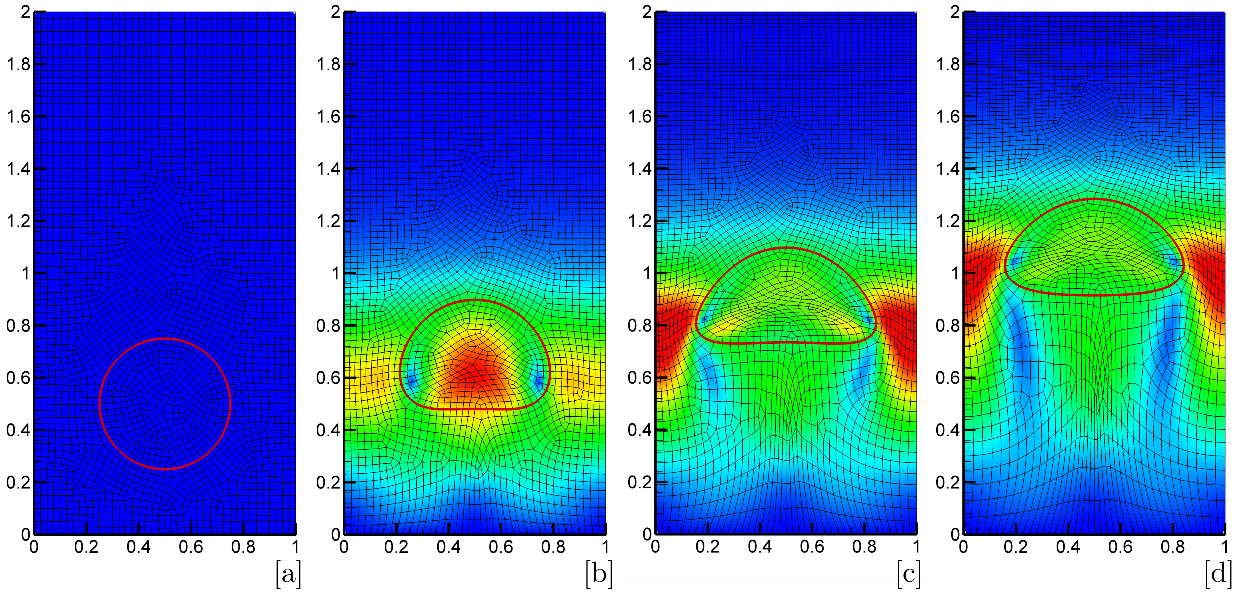

Figure 1: Mesh deformation on mesh M1 at time levels t=0 [a], t=1 [b], t=2 [c] and t=3 [d] with velocity magnitude contours.

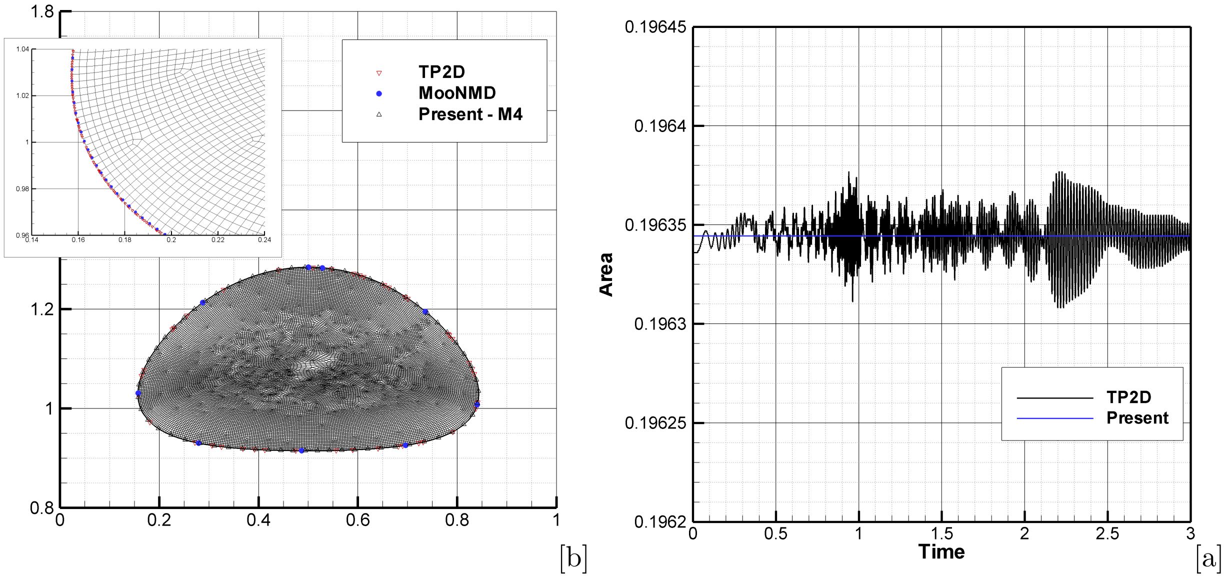

Figure 2: Comparison of bubble shapes on mesh M4 with TP2D (red) and MooNMD (blue) at t=3 [a] and the comparison of the bubble area time variation with the result of TP2D [b].

ACKNOWLEDGMENTS

The authors also gratefully acknowledge the computing resources provided by the National Center for High Performance Computing of Turkey (UYBHM) under grant number 10752009 and the computing facilities at TUBITAK ULAKBIM, High Performance and Grid Computing Center.

References

Dyadechko, V. and Shashkov, M. (2005). Moment-of-fluid interface reconstruction. Technical Report LA-UR-05-7571, Los Alamos National Laboratory.

Erzincanli, B. and Sahin, M. (2013). An arbitrary lagrangian-eulerian formulation for solving moving boundary problems with large displacements and rotations. *J. Comput. Phys.*, 255(C):660–679.

Gunstensen, A. K., Rothman, D. H., Zaleski, S., and Zanetti, G. (1991). Lattice boltzmann model of immiscible fluids. *Phys. Rev. A*, 43:4320–4327.

Harlow, F. H. and Welch, J. E. (1965). Numerical Calculation of Time-Dependent Viscous Incompressible Flow of Fluid with Free Surface. *Physics of Fluids*, 8:2182–2189.

Hirt, C., Amsden, A., and Cook, J. (1974). An arbitrary lagrangian-eulerian computing method for all flow speeds. *Journal of Computational Physics*, 14(3):227 – 253.

Hirt, C. and Nichols, B. (1981). Volume of fluid (vof) method for the dynamics of free boundaries. *Journal of Computational Physics*, 39(1):201 – 225.

Hysing, S., Turek, S., Kuzmin, D., Parolini, N., Burman, E., Ganesan, S., and Tobiska, L. (2009). Quantitative benchmark computations of two-dimensional bubble dynamics. *International Journal for Numerical Methods in Fluids*, 60(11):1259–1288.

Idelsohn, S. R., Mier-Torrecilla, M., Nigro, N., and Oñate, E. (2010). On the analysis of heterogeneous fluids with jumps in the viscosity using a discontinuous pressure field. *Computational Mechanics*, 46(1):115–124.

Moës, N., Dolbow, J., and Belytschko, T. (1999). A finite element method for crack growth without remeshing. *International Journal for Numerical Methods in Engineering*, 46(1):131–150.

Monaghan, J. (1994). Simulating free surface flows with sph. *Journal of Computational Physics*, 110(2):399 – 406.

Osher, S. and Sethian, J. A. (1988). Fronts propagating with curvature-dependent speed: Algorithms based on hamilton-jacobi formulations. *J. Comput. Phys.*, 79(1):12–49.

Perot, B. and Nallapati, R. (2003). A moving unstructured staggered mesh method for the simulation of incompressible free-surface flows. *J. Comput. Phys.*, 184(1):192–214.

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

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. *Journal of Computational Physics*, 169(2):708 – 759.

Uzgoren, E., Sim, J., and Shyy, W. (2007). Computations of multiphase fluid flows using marker-based adaptive, multilevel cartesian grid method. In *45th AIAA Aerospace Sciences Meeting and Exhibit*. American Institute of Aeronautics and Astronautics.

Walkley, M. A., Gaskell, P. H., Jimack, P. K., Kelmanson, M. A., and Summers, J. L. (2005). Finite element simulation of three-dimensional free-surface flow problems with dynamic contact lines. *International Journal for Numerical Methods in Fluids*, 47(10-11):1353–1359.