A SKEW-SYMMETRIC FORMULATION FOR THE SIMULATION OF MULTIPHASE TURBULENT FLOWS

Daniel Fuster
CNRS (UMR 7190), Université Pierre et Marie Curie, Institut Jean le Rond d'Alembert, France.

Summary We propose an energy preserving formulation for the simulation of multiphase turbulent flows. The new formulation reduces the numerical diffusion with respect to previous formulations dealing with multiple phases, which makes this method especially appealing for the simulation of turbulent multiphase flows. In this work we discuss the accuracy and conservation properties of the method in various scenarios with large density and viscosity jumps across the interface. The method is also devised to reduce the *parasitic* currents induced by surface tension effects.

INTRODUCTION

Nowadays the simulation of turbulent flows is the object of intense research. Although numerical techniques and models have been already widely investigated in the literature for single phase flows, current studies are still far from a complete understanding of real processes encountered in a countless number of applications involving multiple phases, such as rocket engines injectors, diesel engines or chemical processes.

The selection of the most suitable numerical scheme for the simulation of complex problems involving different phases is not simple. One must evaluate the pros and cons of each numerical scheme for the particular problem that needs to be solved. In this work, we propose a new numerical scheme for the simulation of turbulent multiphase flows. The method can be seen as a generalization to multiphase flows of the discretization procedure followed by Verstappen et al [4] for single phase flows. This method has been shown to strictly conserve energy irrespective of the mesh resolution in cases with no physical mechanisms of energy dissipation [4]. Thus, we can interpret this method as a numerical scheme devised to reduce numerical dissipation at expenses of the local truncation error. This property is especially appealing for turbulent multiphase flows for under-resolved simulations, where one wants to model the *effective* viscosity from physical backgrounds without any interference from the numerical discretization (e.g. numerical viscosity). When dealing with multiple phases, one must choose a method to track the interface. Among all the methods available in the literature, the Volume Of Fluid (VOF) method is especially suitable for the simulation of under-resolved simulations because it enforces mass conservation and it handles natural topological changes at the interface in the subgrid level. We chose the Gerris Flow Solver [2] as a platform to implement the new numerical scheme. This code has been well validated for the simulation of multiphase flows in various problems involving interfaces. Moreover, the method has been designed to reduce the appearance of *parasitic* currents induced by surface tension forces [2], which is a desirable characteristic of the numerical scheme if one wants to reduce any artificial energy dissipation/generation on the system.

NUMERICAL SCHEME

The underlying idea of the skew-preserving formulations is that the discretization scheme must satisfy certain properties in order to preserve mass and energy. Such properties are satisfied at expenses of increasing the local truncation error with respect to other formulations. In incompressible fluids, mass conservation is typically imposed by satisfying the incompressibility condition for the velocity field at each time step. However, this condition is not sufficient to guarantee conservation of other properties such as the total energy. From a mathematical point of view it can be proved that, for an inviscid fluid without any external force, the evolution of the total system's energy is strictly zero because certain properties of the continuous differential operators hold [4]. Based on these properties for the continuous operators, Verstappen et al [4] have presented a 2nd and 4th order conservative scheme forcing the discrete operators to satisfy the conditions satisfied for continuous operators. Interestingly, this formulation has been developed for arbitrary non-uniform structured grids.

In this work, we extend this formulation to flows with arbitrary density and viscosity jumps including surface tension effects. We adopt the 2nd order scheme described in [4], obtaining the proper expressions for the laplacian, advection and diffusive operators for variable density and viscosity flows. The main advantage of a 2nd order formulation with respect to higher order methods is that it allows for a relatively simple extension of the formulation to octrees. Thus, the Adaptive Mesh Refinement Techniques (AMR) implemented in Gerris [2] can be used also with this formulation.

Surface tension is incorporated in the formulation by exploiting the equality obtained from the momentum equation in absence of motion following the procedure used in the current implementation in Gerris. We emphasize that it is important to pay especial attention to the temporal discretization scheme in order to completely remove *parasitic* currents for a

[a] Corresponding author. E-mail: fuster@dalembert.upmc.fr.

droplet in equilibrium.

NUMERICAL RESULTS

In order to test the performance of the method in flows with multiple phases we consider the following problem. A single bubble with diameter $D = 2/3$ is placed at $(0, 0)$ inside a rectangular domain with dimensions $L/D = [-1.5 : 1.5], [-1.5 : 3]$. The gas density is set to $\rho_g = 1.226$ and the liquid density is $\rho_l = 1000$. Gravity ($g = -9.81$) and surface tension ($\sigma = 728$) are included. The bubble rise and deformation is controlled by buoyancy and tension forces. The most representative nondimensional number is the Bond number ($\mathrm{Bo} = 5.983$). This problem has been discussed by [1] and later used by Popinet [2] as a validation of the surface tension implementation.

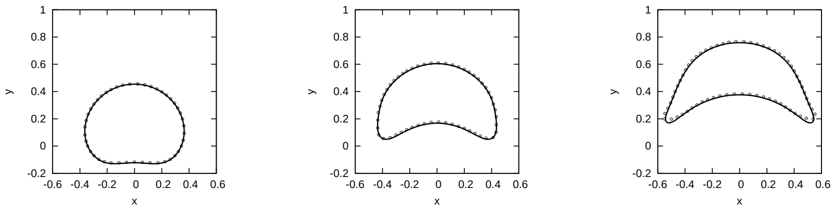
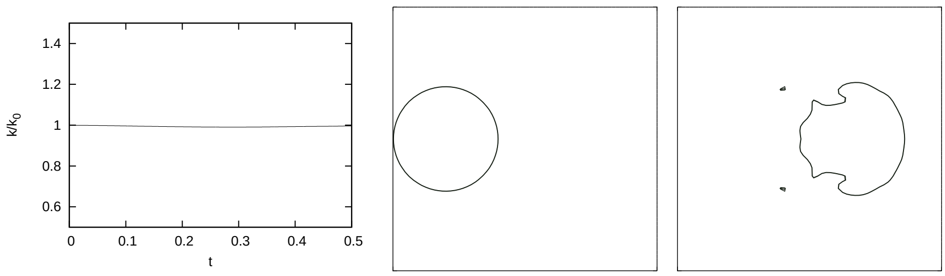


Figure 1. Bubble shape at t=0.2,0.35 and 0.5 obtained with the Godunov scheme implemented in Gerris ($\odot$) and with the new skew symmetric formulation (-)

Figure (1) compares the interface shape obtained with standard advection scheme implemented in Gerris. As expected, for well resolved simulations, both formulations recover the same solution.

The method has been also proved to provide accurate solutions for an inviscid oscillating droplet [3]. In addition, several independent tests have shown that the method ensures energy conservation irrespective of the density ratio when the interface is well resolved (e.g. Figure 2).



(a) Kinetic energy temporal evolution (b) Interface at $t = 0$. (c) Interface at $t = 0.5$.

Figure 2. Temporal evolution of the kinetic energy in a two-dimensional periodic box of an inviscid droplet with $U_0 = 1$ in an inviscid gas initially at rest. $\rho_g/\rho_l = 10^{-3}$ $\sigma = 0$.

CONCLUSIONS

We present an extension of a skew-symmetric formulation for the simulation of multiphase flows. The new formulation ensures energy conservation in absence of any source of physical dissipation irrespective of the density jump across the interface. It also prevents the appearance of *parasite* currents induced by surface tension effects.

References

[1] M. M. Francois, Cummins S. J., Dendy E. D., Kothe D. B., Sicilian J. M., and Williams M. W. A balanced-force algorithm for continuous and sharp interfacial surface tension models within a volume tracking framework. *Journal of Computational Physics*, 213:141–173, 2006.

[2] S. Popinet. An accurate adaptive solver for surface-tension-driven interfacial flows. *Journal of Computational Physics*, 288:5838–5866, 2009.

[3] D.J. Torres and J.U. Brackbill. The point-set method: front-tracking without connectivity. *Journal of Computational Physics*, 165(2):620–644, 2000.

[4] R. Verstappen and A.E.P. Veldman. Symmetry-preserving discretization of turbulent flow. *Journal of Computational Physics*, 187(1):343–368, 2003.