Journal of Computational Physics 274 (2014) 19–49

Contents lists available at ScienceDirect

Journal of Computational Physics

www.elsevier.com/locate/jcp

A low diffusive Lagrange-remap scheme for the simulation of violent air–water free-surface flows

Aude Bernard-Champmartin [a,b,1], Florian De Vuyst [a]

[a] CMLA-ENS Cachan, 61 avenue du Président Wilson, BP 94235, Cachan, France
[b] LRC MESO, ENS CACHAN, 61 avenue du Président Wilson, 94235 Cachan Cedex, France

ARTICLE INFO

Article history:
Received 10 March 2013
Received in revised form 23 April 2014
Accepted 22 May 2014
Available online 2 June 2014

Keywords:
Numerical method
Multiphase flow
Air–water flow
Free boundary
Interface capturing
Compressible fluid
Finite volume
Lagrange-remap solver
Solver for parallel computing
Wave breaking
Sloshing
Impact problem

ABSTRACT

In 2002, Després and Lagoutière [17] proposed a low-diffusive advection scheme for pure transport equation problems, which is particularly accurate for step-shaped solutions, and thus suited for interface tracking procedure by a color function. This has been extended by Kokh and Lagoutière [28] in the context of compressible multifluid flows using a five-equation model. In this paper, we explore a simplified variant approach for gas–liquid three-equation models. The Eulerian numerical scheme has two ingredients: a robust remapped Lagrange solver for the solution of the volume-averaged equations, and a low diffusive compressive scheme for the advection of the gas mass fraction. Numerical experiments show the performance of the computational approach on various flow reference problems: dam break, sloshing of a tank filled with water, water–water impact and finally a case of Rayleigh–Taylor instability. One of the advantages of the present interface capturing solver is its natural implementation on parallel processors or computers.

1. Introduction

Simulation of free surface flows knows an increasing interest as an essential predictive tool for innovative engineering designs into many fields of applications, and a complementary analysis tool compared to physical experiments. This includes for instance the safety study of water dams, tsunamis, the extraction of offshore petroleum, the sizing of Liquified Natural Gas (LNG) carriers, processes of phase separation, waste water treatment, flocculation processes, bio-engineering, medical applications, etc. The evolution of the interfaces between phases and the consecutive complex dynamics needs to be simulated for the understanding of the flows and the process optimization in the industrial case. For gas–liquid applications involving fast dynamics, beyond quantities of interest, numerical models have to be able to capture most of the flow features that drive the dynamics, e.g.:

- wave formation and wave breaking;
- wall wave impacts, local pressure peaks and pressure loadings;

E-mail addresses: champmar@cmla.ens-cachan.fr (A. Bernard-Champmartin), devuyst@cmla.ens-cachan.fr (F. De Vuyst).
[1] Present addresses: INRIA Sophia Antipolis Méditerranée, 2004, route des Lucioles - BP 93, 06902, Valbonne, France; LRC MESO, ENS CACHAN, 61 avenue du Président Wilson, 94235 Cachan Cedex, France.

http://dx.doi.org/10.1016/j.jcp.2014.05.032
0021-9991/

- formation of air pockets;
- ejection, fragmentation of liquid droplets;
- Archimedes buoyancy effect with rising of bubbles and fall of droplets;
- effects of gas compressibility inducing a gas-to-liquid response by a pressure wave, etc.

In this paper, we consider immiscible gas–liquid two-phase flow problems. The strong ratio of mass density between gas and liquid (typically 1:1000) is known to be a source of numerical stiffness and numerical instability. Therefore robust computational approaches supporting high density ratio have to be considered. Among the family of conservative Finite Volume methods (FVM), the Lagrange-remapped solvers (see e.g. [42,45,6,4,25,2]) provide both robustness and stability with achievement of mathematical properties of positiveness and entropy compatibility.

Lagrange-remap numerical schemes (also referred to as Euler–Lagrange schemes) are a particular family of Eulerian FVM where, at each time step, the equations are solved according to a Lagrangian evolution with a mesh that is convected by the flow itself, then the "Lagrange solutions" are remapped on the initial mesh into a conservative way by estimating both fluxed mass and momentum. Because of the Lagrangian step of these methods, code coupling or coupling of different physics is made easier against conventional FV methods. Moreover, the Lagrangian description is very practical for multi-material flows of multi-phase flows because we are a natural control of the fluxed quantities material-by-material. Lagrange and Lagrange-remap solvers still know strong developments today with major contributions as e.g. energy-preserving compatible schemes for staggered methods [11], collocated variables and cell-centered entropy-satisfying schemes, see [18,32,12].

In this paper, we rather consider a simpler staggered Lagrange-remap solver with a direction-by-direction remapping. More precisely, the 2D multidimensional Lagrange step (operator $\mathcal{L}_{xy}^{\Delta t}$) completely solves the fluid equations while an operator splitting alternating direction (AD) for the projection (operators $\mathcal{R}_x$ and $\mathcal{R}_y$ respectively) is used to interpolate on the reference Cartesian Eulerian grid into a conservative manner, involving convective flux balances. Symmetrized operator splitting can be used to ensure second order accuracy. Both linear and nonlinear numerical stabilities are ensured by the use of standard pseudo-viscosity (viscous pressure) terms, detailed into Appendix B at the end of this paper. Actually, we use this simple Euler solver because of its simplicity of code implementation and because it can be vectorized/parallelized into a natural manner. Moreover, the aim of this paper is not about the hydrodynamics solver: the article mostly focuses on numerical antidiffusive methods for interface capturing, as part of a global multifluid hydrodynamics solver. At the present time, the antidiffusive approaches assume a direction-by-direction remapping which leads to a simpler derivation of the antidiffusive fluxes.

The issue of an interface tracking/capturing algorithm providing expected properties like robustness, accuracy, conservation of volume and mass while not being too much computationally intensive is still the object of today's active research. Pure Lagrangian approaches like Smoothed Particle Hydrodynamics (SPH) methods naturally captures the moving interfaces because each macro-particle moves with the flow. Each particle is also attached to a given material with its own equation of state (EOS). For liquid–gas flows, we have liquid particles and gas particles and the interface is nothing else but the discrete interface separating liquid particles from gas particles. In the last decade, we have seen in the literature major contributions of improvement in the SPH world with improved accuracy, stability, and ability to tackle multiphase flow problems with high density ratios for violent flow applications, see for example [13,33,24]. Despite these improvements, SPH still know some issues. Moreover parallelization techniques for SPH are quite technical and require a great expertise (as dealing with strong density ratio between phases). That's why we rather choose a more classical computational approach.

Notice also that there are also mesh-based Lagrangian methods [32,12]. But for flows with large deformations, cells may become degenerate, and both regularization or remapping procedures are needed.

For Eulerian methods, the family of interface tracking methods try to reconstruct the free boundary according to some tracking procedure (level set methods [38] for example). Interface reconstruction methods try to reconstruct a moving interface according to some incomplete information: volume-of-fluid (VOF) methods [36,46] or MOF (Moment-of-Fluid) methods [20]. The family of interface capturing methods involves at the continuous level the transport equation of an indicator function that distinguishes the location of the different materials:

$$\partial_t z + \boldsymbol{u} \cdot \nabla z = 0, \quad z \in \{0, 1\}$$

(where u is the velocity and z the color function or a phase indicator), expressing that the interface property between the two phases is advected with the local fluid velocity. From the numerical point of view, finite volume cells may host different materials. The so-called "mixed cells" need an additional closure but on the other hand the mass fraction $c_g \in [0, 1]$ of the gas fluid may be used as the numerical indicator function:

$$\partial_t c_g + \boldsymbol{u} \cdot \nabla c_g = 0, \quad c_g \in [0, 1].$$

It can be set up in conservative form in order to conserve the gas mass:

$$\partial_t (\rho c_g) + \nabla \cdot (\rho c_g \boldsymbol{u}) = 0$$

with ρ representing the mean cell "mixture" density, but of course it induces a "diffuse" interface, which has to be kept as less diffusive as possible. Let us emphasize that this diffuse feature is only a pure numerical artefact. This kind of methods uses more or less sophistication levels including high-order schemes, compressive flux limiters, artificial compression stages, local adaptive mesh refinement (AMR) [7], *a posteriori* methods in which an anti-diffusion phase is added after the projection