European Journal of Mechanics B/Fluids 42 (2013) 10–19

Contents lists available at ScienceDirect

European Journal of Mechanics B/Fluids

journal homepage: www.elsevier.com/locate/ejmflu

# A totally Eulerian finite volume solver for multi-material fluid flows III: The low Mach number case

Jean-Philippe Braeunig [a], Daniel Chauveheid [b,a,*], Jean-Michel Ghidaglia [b,c]

[a] CEA, DAM, DIF, F-91297 Arpajon, France

[b] CMLA, ENS Cachan and CNRS UMR 8635, 61 avenue du Président Wilson, F-94235 Cachan Cedex, France

[c] LRC Meso, CEA, DAM and ENS Cachan, France

ARTICLE INFO

*Article history:*
Received 4 February 2013
Received in revised form
11 May 2013
Accepted 20 June 2013
Available online 28 June 2013

*Keywords:*
Multi-material solvers
Upwind finite volume schemes
Implicit schemes
Low Mach number
Preconditioning
Kelvin–Helmholtz instability

ABSTRACT

We build an extension of the FVCF-ENIP multi-material solver for the computation of low Mach number fluid flows. The loss of accuracy generally noticed for Godunov type solvers when the Mach number goes to zero is handled through the renormalization of the viscosity matrix of the numerical flux. An implicit scheme is proposed in order to achieve numerical stability with reasonable CFL numbers. Its linear unconditional stability is proved in the case of a one-dimensional isentropic Euler system. Various numerical tests attest the efficiency of the method.



## 1. Introduction

Due to the tremendous computational power available through high performance computing, simulation is getting more and more importance in various domains such as process manufacturing, process engineering, and security in industry. In particular, simulation users would like computational fluid dynamics (CFD) tools to be able to handle a very wide range of flows. Among them, there are multi-fluid or even multi-material flows. These of course include compressible flows. For example, in the classical case of air and water flows, the compressibility of water being much smaller than that of air, a solver for such flows should be able to handle both moderate and low Mach number flows.

The use of an incompressible solver does not fulfill every expectation. Compressible phenomena associated with important thermal effects, as well as shock waves, may occur at very low Mach number flows, that is, when the fluid velocity is low compared to the speed of sound. There are then two strategies. The first is to extend a solver for incompressible flows (zero Mach number, $\mathcal{M} = 0$) to moderate and low Mach numbers flows, while the second is to extend a solver for compressible flows designed for high and moderate Mach numbers to low Mach numbers, let us say $\mathcal{M} \leq 0.1$. The second strategy, which we adopt in this article, should lead to a more general simulation tool, since it allows for the full range of Mach numbers.

Most of the modern compressible fluid flows solvers, in particular those designed for dealing with complex geometries, are based on upwinding to achieve numerical stability: Godunov [1] and Roe [2] schemes, see e.g. Godlewski and Raviart [3], are the well-known pioneering works for compressible external flows in aeronautics. However, as analyzed by Turkel [4,5], upwinding induces too much diffusion as the Mach number tends to 0. This phenomenon was further analyzed by Guillard and Viozat [6] and Guillard and Murrone [7]. All these authors propose a modification of Roe's scheme in order to capture the right solution. Turkel has proposed a non-consistent in time modification of Roe's scheme which allows one however to capture a consistent stationary solution, while Guillard and his co-authors have extended Turkel's approach to obtain fully consistent solvers. More recently, Dellacherie [8] and Li [9] have extended these works in the context of single fluid flows.

An asymptotic development of the numerical schemes allows one to identify the continuous equations satisfied by their solutions. These limit equations show pressure fluctuations of the order of the Mach number, while, at the continuous level, another asymptotic analysis underlines the fact that the pressure

* Corresponding author at: CEA, DAM, DIF, F-91297 Arpajon, France. Tel.: +33 147407611.

*E-mail addresses:* jean-philippe.braeunig@cea.fr (J.-P. Braeunig), daniel.chauveheid@cmla.ens-cachan.fr (D. Chauveheid), jmg@cmla.ens-cachan.fr (J.-M. Ghidaglia).

0997-7546/$ – see front matter © 2013 Elsevier Masson SAS. All rights reserved.
http://dx.doi.org/10.1016/j.euromechflu.2013.06.007

fluctuations scale with the square of the Mach number [6–9]. Therefore, cell-centered schemes (like Godunov's or Roe's) fail in rendering accurate solutions as the Mach number becomes too small. Most solutions to this problem use preconditioning techniques (stemming from Turkel's work) that consist in modifying the numerical fluxes so as to improve the condition number of the viscosity matrix [4–7], that is, for an invertible matrix $A$, the number $\kappa$ such as $\kappa = \|A\|\,\|A^{-1}\|$, for a given matrix norm $\|\cdot\|$. However, these modifications directly impact the stability of the considered scheme. In particular, explicit preconditioned schemes are unstable in the sense that the Courant–Friedrichs–Lewy condition is highly restrictive and gives rise to time steps of the order of the square of the Mach number [10,11].

In this paper, we consider multi-fluid or multi-material non-miscible flows governed by the compressible Euler equations. Our goal is to extend the finite volume with the characteristic flux-enhanced natural interface positioning method [12,13] (FVCF-ENIP method in the following) to low Mach number flows. This aforementioned method is a hybrid Eulerian–Lagrangian method, with sharp interface reconstruction. The fluids present are not miscible: they all satisfy the compressible Euler equations but each has its own equation of state. Hence we consider the compressible Euler equations, written in conservative form (mass, momentum and total energy) as

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \mathbf{u}) = 0, \tag{1}$$

$$\frac{\partial (\rho \mathbf{u})}{\partial t} + \nabla \cdot (\rho \mathbf{u} \otimes \mathbf{u} + pI) = \rho\, g, \tag{2}$$

$$\frac{\partial (\rho E)}{\partial t} + \nabla \cdot ((\rho E + p)\mathbf{u}) = \rho\, g \cdot \mathbf{u}, \tag{3}$$

with the usual notation: $\rho$ denotes the density, $\mathbf{u}$ the velocity field, $E$ the specific total energy, sum of $e$, the specific internal energy and $\frac{1}{2}\|\mathbf{u}\|^2$, the specific kinetic energy and $g$ is the gravity vector. The system is closed by equations of state:

$$\rho = \mathcal{R}(p\,, T), \qquad e = \mathcal{E}(p\,, T), \qquad s = \mathcal{S}(p\,, T), \tag{4}$$

where $p$ and $T$ are the pressure and the temperature. Here, $s$ denotes the specific entropy, which satisfies Gibbs' relation $Tds = de - \frac{p}{\rho^2} d\rho$.

The content of the paper is as follows. We recall in Section 2 the original FVCF-ENIP method. Then, according to the aforementioned fact concerning the stability condition for explicit renormalized low Mach schemes, we derive a fully implicit multi-material scheme in Section 3. Then this scheme is modified using a renormalization technique for which we prove linear unconditional stability (Appendix A) in the simplified case of a one-dimensional (1D) isentropic Euler system.

We illustrate the efficiency of our method in Section 4 by presenting various numerical results. Among them, an important test case in this context is the simulation of the Kelvin–Helmholtz instability (Youngs and Williams [14]). Concerning this case, we also prove analytically that the growth rate of the instability does not depend on the value of the Mach number (Appendix B). The paper ends with some concluding remarks and suggestions for future work in Section 5.

## 2. The original FVCF-ENIP method

Eqs. (1)–(3) are rewritten in the following form:

$$\frac{\partial v}{\partial t} + \nabla \cdot F(v) = S(v), \tag{5}$$

where $v$ denotes the conserved variables: $v = {}^t(\rho,\ \rho\mathbf{u},\ \rho E)$, and $F(v)$ is the physical flux such that $F(v)\cdot\nu = (\rho(\mathbf{u}\cdot\nu),\ \rho(\mathbf{u}\cdot\nu)\mathbf{u} + p\nu,\ (\rho E + p)(\mathbf{u}\cdot\nu))'$ is the normal flux in the direction $\nu \in \mathbb{S}^{d-1}$,

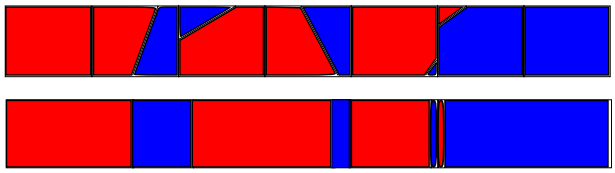

**Fig. 1.** Construction of a condensate.

$d$ being the physical space dimension, and finally $S(v)$ is a source term, such as gravity, for example.

As explained above, the underlying method that we use is the FVCF-ENIP method. This method allows one to compute non-miscible multi-material flows with sharp interface reconstruction, and it was designed in order to be strictly conservative for the conserved variables, that is, mass, momentum, and total energy. We refer to [12,13] for a complete description of the algorithm in two dimensions or [15] for the 3D extension. However, its main features have to be described here.

The FVCF-ENIP algorithm requires the use of a Cartesian grid. Therefore, system (5) can be solved using a directional splitting and it then reduces to mono-dimensional subsystems that are solved by alternating $x$–$y$ and $y$–$x$ time step after time step for the sake of symmetry and time integration stability.

To each cell in the domain, we associate a data structure containing the amounts of material present in this cell and the volume fractions of the materials. For each of them, we have the corresponding values of the conserved variables $v$. Since we deal with multi-material flows, the common terminology is used. A cell is said to be "pure" if it contains only one material, or "mixed" if there are at least two materials present.

When applying the space-splitting, treating the 2D domain line by line (or column by column), we generically obtain a 1D object (that we will call the generic $x$-line), consisting in a juxtaposition of pure and mixed cells.

The FVCF-ENIP method mainly relies on the use of a data structure called the "condensate" [12,13] to deal with mixed cells. In the condensates, layers of the same fluids in neighboring mixed cells are agglomerated (namely condensed), or not, according to their order of appearance in the cell. The order in question is prescribed by the direction of the unit vector normal to the interface.

Once the condensates are constructed, the generic $x$-line consists in a juxtaposition of single material layers with different volumes. At this stage, the edges separating the layers are of two types: Eulerian (and therefore fixed), when the edge lies between two cells of the same fluid, or Lagrangian, when the interface separates two different fluids. The latter corresponds to material interfaces and can be found only in the inner parts of the constructed condensates.

The construction of a condensate is illustrated in Fig. 1.

Integration of (5) over a time-dependent control volume $K_{i,j,k}(t) = [x_i(t), x_{i+1}(t)] \times [y_j, y_{j+1}] \times [z_k, z_{k+1}]$, keeping only the terms corresponding to the derivation in the generic $x$-direction, leads to a system of ordinary differential equations:

$$\frac{d|K_{i,j,k}|V_{K_{i,j,k}}}{dt} + \big(|\Gamma_{i+1/2,j,k}|\phi(v_{i+1,j,k},\, v_{i,j,k}) - |\Gamma_{i-1/2,j,k}|\phi(v_{i,j,k},\, v_{i-1,j,k})\big) = S(V_{K_{i,j,k}}). \tag{6}$$

As usual in finite volume methods, $V_{K_{i,j,k}(t)}$ denotes the mean value of $v$ on the control volume: $V_{K_{i,j,k}(t)} = \frac{1}{|K_{i,j,k}|}\int_{K_{i,j,k}(t)} v(\mathbf{v}, t)d\mathbf{x}$, and $\phi(v_{i+1,j,k},\, v_{i,j,k})$ denotes the numerical flux at the interface between volumes $K_{i,j,k}$ and $K_{i+1,j,k}$. Here, $|\Gamma_{i+1/2,j,k}|$ is the measure of the area of the edge located at $x_{i+1/2} \equiv \frac{x_i + x_{i+1}}{2}$. Eventually, $|K_{i+1,j,k}|$ is the measure of the control volume $K_{i+1,j,k}$.