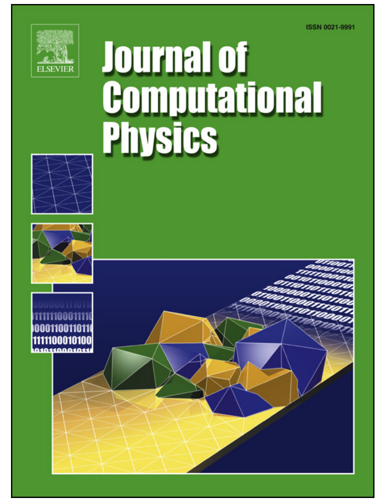
Accepted Manuscript



A robust implicit-explicit acoustic-transport splitting scheme for two-phase flows

S. Peluchon, G. Gallice, L. Mieussens

PII: S0021-9991(17)30206-1
DOI: http://dx.doi.org/10.1016/j.jcp.2017.03.019
Reference: YJCPH 7223

To appear in: *Journal of Computational Physics*

Received date: 3 June 2016
Revised date: 7 March 2017
Accepted date: 9 March 2017

Please cite this article in press as: S. Peluchon et al., A robust implicit-explicit acoustic-transport splitting scheme for two-phase flows, *J. Comput. Phys.* (2017), http://dx.doi.org/10.1016/j.jcp.2017.03.019

A robust implicit-explicit acoustic-transport splitting scheme for two-phase flows

S. Peluchon[a,c], G. Gallice[a], L. Mieussens[b,c]

[a] CEA-CESTA, 15 avenue des sablières - CS 60001, 33116 Le Barp Cedex, France,
[b] Univ. Bordeaux, CNRS, Bordeaux INP, IMB, UMR 5251, F-33400 Talence,
[c] INRIA, F-33400 Talence, France.

Abstract

In this paper, a splitting strategy to simulate compressible two-phase flows using the five-equation model is presented. The main idea of the splitting is to separate the acoustic and transport phenomena. The acoustic step is solved in a non-conservative form using a scheme based on an approximate Riemann solver. Since the acoustic time step induced by the fast sound velocity is very restrictive, an implicit treatment of this step is performed. For the transport step driven by the slow material waves, an explicit scheme is used. Although non-conservative forms are used to derive numerical schemes for the two steps, the overall scheme resulting from this splitting operator strategy is conservative. It preserves contact discontinuities and reveals to be very robust compared to a standard unsplit scheme.
Numerical simulations of compressible two-phase flows are presented on two-dimensional structured grids. The implicit-explicit strategy allows large time steps, which do not depend on the fast acoustic waves.

Keywords: two-phase flows, positive Riemann solver, implicit scheme.

Introduction

The present work takes place in the context of the atmospheric re-entry problem. This study can concern re-entry vehicles globally or partially made of metallic components, space debris for instance. During the re-entry phase, a solid undergoes a heating due to the friction of atmospheric gases. Conversion of kinetic energy to thermal energy leads to a sudden increase of the temperature of the object. This rise drives to a physical-chemical degradation of the thermal protective system, and to a boundary recession. Sublimation (injection of gas into the atmosphere) and the fusion of the metallic part (creation of a liquid phase into the gas) are the main causes of the solid ablation during the re-entry phase. A zonal approach is considered (see Fig 1) to cope with this very complex problem. In the gas flow region, far from the object or near a wall made of carbon where the ablation process is driven by the sublimation, classical schemes can be used. Numerical simulations of the sublimation process have already been studied in [MB14, MC13, BNM10, Mul10, Lat13]. In the region near a metallic wall, the velocity of the gas is small and with the appearance of the liquid phase, the dynamics of the flow is very different. From our own experience, usual numerical schemes are not very robust to compute such two-phase flows when large time steps are considered. In the present paper, we focus on the multiphase flow region and we propose a numerical method to simulate two-phase flows. The work presented here is really the first step of a global project since viscosity effects and heat transfers are not taken into account.

The modelling and computation of multiphase flows have been widely studied for the past decades. There are two main approaches to compute compressible flows with interfaces: sharp interface methods, and diffuse interface methods. In the first approach, the interface between the two media, considered as a sharp discontinuity, is followed explicitly and each phase can be computed with different models. In Lagrangian or Arbitrary Lagrangian-Eulerian methods, the mesh moves during the computation like the interface. Large distortions and interface topological changes can hardly be taken into account. Front capturing methods are Eulerian methods where the interface is reconstructed. In Level Set methods [OS88] the interface is located as the zero of an implicit function. In the Volume Of Fluid method [HN81], the interface is reconstructed from the volume fraction of each fluid. In the second approach, diffuse interface methods [BN86, KMB+01, ACK02, MSNA02, FBC+11, KL10, SA99], based on an Eulerian mesh, allow numerical diffusion of the interface. The same equations are solved in the entire domain. In addition, these models allow the creation of new interfaces and topological changes during the simulation. In the present context, the fusion of the metallic part leads to a significant topological change. Consequently, we choose to use a diffuse interface approach in order to simulate the two-phase flow. Those methods