Commun. Comput. Phys.
doi: 10.4208/cicp.140213.210214a

Vol. **16**, No. 2, pp. 403-439
August 2014

Robust Conservative Level Set Method for 3D Mixed-Element Meshes — Application to LES of Primary Liquid-Sheet Breakup

Thibault Pringuey* and R. Stewart Cant

CFD Laboratory, Department of Engineering, University of Cambridge, UK.

Received 14 February 2013; Accepted (in revised version) 21 February 2014

Available online 21 May 2014

Abstract. In this article we detail the methodology developed to construct an efficient interface description technique — the robust conservative level set (RCLS) — to simulate multiphase flows on mixed-element unstructured meshes while conserving mass to machine accuracy. The approach is tailored specifically for industry as the three-dimensional unstructured approach allows for the treatment of very complex geometries. In addition, special care has been taken to optimise the trade-off between accuracy and computational cost while maintaining the robustness of the numerical method. This was achieved by solving the transport equations for the liquid volume fraction using a WENO scheme for polyhedral meshes and by adding a flux-limiter algorithm. The performance of the resulting method has been compared against established multiphase numerical methods and its ability to capture the physics of multiphase flows is demonstrated on a range of relevant test cases. Finally, the RCLS method has been applied to the simulation of the primary breakup of a flat liquid sheet of kerosene in co-flowing high-pressure gas. This quasi-DNS/LES computation was performed at relevant aero-engine conditions on a three-dimensional mixed-element unstructured mesh. The numerical results have been validated qualitatively against theoretical predictions and experimental data. In particular, the expected breakup regime was observed in the simulation results. Finally, the computation reproduced faithfully the breakup length predicted by a correlation based on experimental data. This constitutes a first step towards a quantitative validation.

AMS subject classifications: 65M08, 76-04, 76N99

Key words: Multiphase flow, level set method, flux limiter, MULES, WENO scheme, three-dimensional, unstructured mesh, mixed element, conservative method, atomisation, primary breakup, flat sheet breakup.

*Corresponding author. *Email address:* tp299@cam.ac.uk (T. Pringuey)

1 Introduction

Multiphase flows are encountered in a very broad variety of fields ranging from fundamental physics to geophysics and engineering. As a result, the numerical simulation of multiphase flows and in particular, the modelling of atomisation, has applications in many industries such as: aeronautics, automotive engineering, pharmaceutical, power generation, petro-chemical, manufacturing and agriculture.

In particular, aero-engines rely on air-blast atomisers to inject the kerosene in combustion chambers. The kerosene is generally injected as an annular liquid sheet sheared on either side by a faster co-flowing gas stream. This sheet undergoes a series of instabilities (longitudinal and transverse) which lead to the fragmentation of the liquid bulk into liquid structures that further disintegrate into droplets. This initial process of the atomisation is called the primary breakup and occurs in the vicinity of the injection point. As the prediction of fuel sprays in gas turbines is of critical importance to maximise the combustion efficiency and reduce the pollutant emissions from aviation, aero-engine manufacturers are investing in the development of numerical methods to model the injection process.

Various numerical methods have been developed for the simulation of multiphase flows and the most popular ones can be categorised into two groups known as moving-grid methods and fixed-grid methods. The moving-grid methods [22, 34, 49] treat the interface between the two phases as a boundary between two sub-domains of the mesh. In the case of large deformations of the interface this approach generally requires remeshing and becomes quite cumbersome when topological changes occur. The fixed-grid methods can either describe the interface explicitly or implicitly. The explicit description of the interface typically involves the solution of the flow properties on a fixed grid, together with the Lagrangian transport of a web of massless particles representing the interface. This approach using marker particles — also called the front tracking method [20, 43, 59, 61] — generally provides a precise location for the interface and also offers better control of the interfacial topology changes. However, its extension to 3D is notoriously difficult.

As implicit interface description methods are able to handle changes of interface topology automatically, they offer great potential for the simulation of atomisation. One of the most popular approaches in this category is the Volume Of Fluid (VOF) method [11,23,36] which captures the interface through the transport of the volume fraction. The volume fraction represents the volume occupied by the liquid within a computational cell. Another widely used approach is the Level Set (LS) method [40, 41, 51] which is based on the transport of a continuous function $\phi(\mathbf{x},t)$ by the underlying velocity field. In this framework, the level set (usually $\phi_\Gamma = 0$) of the function ϕ is taken to represent the interface and ϕ takes values below ϕ_Γ in one fluid and above ϕ_Γ in the other. From a numerical point of view, a smooth function is desirable and ϕ is generally taken to be equal to the signed distance from the interface. The main challenge in developing an interface description method is to produce an implicit technique that conserves mass (like Volume