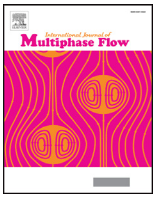
International Journal of Multiphase Flow 91 (2017) 19–38

Contents lists available at ScienceDirect

International Journal of Multiphase Flow

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



A Coupled Level Set and Volume of Fluid method for automotive exterior water management applications

M. Dianat, M. Skarysz, A. Garmory*

Department of Aeronautical and Automotive Engineering, Loughborough University, Loughborough, LE11 3TU, UK

ARTICLE INFO

Article history:
Received 7 June 2016
Revised 11 January 2017
Accepted 23 January 2017
Available online 23 January 2017

Keywords:
Surface flows
Level set
Volume of Fluid
CLSVOF
Contact angle

ABSTRACT

Motivated by the need for practical, high fidelity, simulation of water over surface features of road vehicles a Coupled Level Set Volume of Fluid (CLSVOF) method has been implemented into a general purpose CFD code. It has been implemented such that it can be used with unstructured and non-orthogonal meshes. The interface reconstruction step needed for CLSVOF has been implemented using an iterative ‘clipping and capping’ algorithm for arbitrary cell shapes and a re-initialisation algorithm suitable for unstructured meshes is also presented. Successful verification tests of interface capturing on orthogonal and tetrahedral meshes are presented. Two macroscopic contact angle models have been implemented and the method is seen to give very good agreement with experimental data for a droplet impinging on a flat plate for both orthogonal and non-orthogonal meshes. Finally the flow of a droplet over a round edged channel is simulated in order to demonstrate the ability of the method developed to simulate surface flows over the sort of curved geometry that makes the use of a non-orthogonal grid desirable.

1. Introduction

There are several engineering applications which involve the flow of liquid droplets or rivulets over solid surfaces. One such application is ‘Exterior Water Management’ (EWM) on road vehicles. EWM is important when driving, for example managing the water flowing from the windscreen onto the side glass, or stripping off the wing-mirror housing and impacting the side glass and thereby obscuring vision. It is also important in static situations where water run-off from the roof can enter the vehicle, making seats or the luggage space wet. Hence the motion of individual drops under gravity is of interest when designing features such as drainage channels which prevent this. Hagemeier et al. (2011) provides a thorough review of the issue of vehicle EWM and the state of the art of numerical simulation. His review indicates that there are a number of significant gaps in the simulation capability and because the water management features, such as channels, must be fixed at an early stage in the vehicle design it is clear that an accurate method to simulate EWM and contamination would be highly advantageous. Examples of EWM simulations can be found in Gaylard et al. (2012) and Jilesen et al. (2015) that both use Lagrangian particle tracking for the airborne droplets and a 2D film model for the surface flow. While the approach to the dispersed phase (airborne) may be satisfactory, the assumption that the surface flow can be modelled using a 2D film assumption has limitations.

Two dimensional film models such as that used in Gaylard et al. (2012) and Jilesen et al. (2015) or that implemented in OpenFOAM following Meredith et al. (2013) solve transport equations for the film thickness but do not resolve the 3D shape of the surface water. In doing so these models make the assumptions that there is no velocity in the liquid normal to the surface and that the three dimensional shape of the film is not important. While it is possible to use this type of film model to predict the motion of droplets and rivulets there will be situations where these assumptions will not hold. For example droplets filling or crossing a drainage channel will have significant velocities normal to the surface and an example of this is included in Section 6. For the cases where aerodynamic drag on the drop or rivulet is important then the two-way coupling between the forces on the liquid and its shape will be important. A thin film approximation cannot simulate this as it does not change the shape of the boundary seen by the flow solver unless complex mesh morphing techniques are also used.

Fluid film models also make use of empirical sub models to account for phenomena such as droplet impingement or film stripping. For these to give accurate predictions it is necessary to use them for the circumstances they were derived for. For example the film model used in the OpenFOAM fluid film model uses a film stripping model (Owen and Ryley, 1985) which assumes that if the

---

* Corresponding author.
E-mail address: A.Garmory@lboro.ac.uk (A. Garmory).

http://dx.doi.org/10.1016/j.ijmultiphaseflow.2017.01.008

film is stripped off the surface it breaks down immediately into a spray of droplets small in comparison to the computational mesh. Hence this model would not give correct results in cases where the liquid leaves the surface in a coherent mass.

The focus of this paper is therefore on developing methods that overcome this limitation. There are a number of requirements for a method suitable for the practical simulation of 3D droplets rivulets and films on a vehicle surface:

- It would require both high resolution of the water surface in 3D to capture droplet shapes, particularly at the surface contact line, and mass conservation to simulate droplet motion over large distances.
- It is essential for the method to work with realistic geometry, including highly curved surfaces, such as those found on vehicles.
- Implementation in a general purpose CFD code will make the method a more practical tool for real applications.
- The method must include the different behaviour of water on different surfaces such as paintwork, glass or treated hydrophobic surfaces.

1.1. Interface capturing methods

Several numerical methods have been developed for 3D interface capturing in multiphase Computational Fluid Dynamics (CFD) which may be relevant for EWM simulations. The most common ones are Front Tracking, Volume of Fluid (VOF), Level Set (LS) and Coupled Level Set and VOF (CLSVOF). Front tracking methods, see for example Tryggvason et al. (2001), represent the interface between the phases using a series of points, joined by triangular elements, located on the interface. The location of a CFD cell relative to this interface determines the fluid properties (i.e. those of liquid or gas) which are used to calculate the velocity field. The points defining the interface are then moved in a Lagrangian fashion using velocities interpolated from the CFD velocity field. The method is able to give precise definition for the interface location but does not strictly conserve mass. Marić et al. (2015) have recently implemented a hybrid Level Set/Front Tracking method for unstructured grids and have, so far, presented results for some test cases but not ones including surface flows.

With the VOF method, the volume fraction, α, is defined as the fraction of volume occupied by the liquid in each cell. VOF is thus bounded between 0 and 1 but changes discontinuously across the interface, see Scardovelli and Zaleski (1999) for a review of the approach. The evolution of α is governed by a simple advection equation using the resolved velocity field. The advantage of VOF is that mass is correctly conserved and it can be applied on any mesh. The simplest, and easiest to implement, VOF method is ‘algebraic VOF’ where the VOF field is transported by a convection term using standard discretisation methods. However, numerical diffusion in the transport scheme causes non-physical smearing of the interface leading to a loss of accuracy in the definition of the interface location. A method of defining or ‘reconstructing’ the interface location within a cell using the value of α and the normal to the interface given by $\nabla\alpha$ can be used to overcome this. Such methods are classed as ‘geometric VOF,’ see Scardovelli and Zaleski (1999) or for a recent example Marić et al. (2013). But as the magnitude of $\nabla\alpha$ should ideally be infinite at the interface, this can lead to numerical problems in the evaluation of this and the interface reconstruction.

An alternative choice for interface capturing, proposed by Sussman et al. (1994), is the Level Set (LS) method. Unlike α, LS function, ϕ, is a continuous variable. It is defined as the signed distance from the interface being positive in the liquid and negative in the gas and zero at the interface itself. LS function is also evolved by another simple advection equation using the resolved velocity field. The advantage of LS methods is that they give a sharp definition to the interface but the disadvantage is that they are not mass conservative and therefore require high-order numerical schemes. For example the Level Set approach was applied by Griebel and Klitz (2013) who used a Cartesian mesh with a 5th order WENO scheme to simulate the motion of a droplet impinging on a plate. More recently a conservative form of the Level Set approach has been developed by Pringuey and Cant (2014) for use with unstructured meshes. Early results with this method are encouraging but the method still relies on high order spatial schemes which are complex to implement particularly in general purpose unstructured CFD codes.

Previous researchers have combined the advantages of LS and VOF methods. Albadawi et al. (2013) proposed a simple coupled Level Set Volume of Fluid (S-CLSVOF) which was also later used by Yamamoto et al. (2016). In this method a transport equation is solved for VOF but not the Level Set. Instead a Level Set is constructed from the interface (defined as the VOF=0.5 isosurface). This allows more accurate calculation of the surface curvature using the level set.

A fully Coupled LS/VOF (CLSVOF) method was proposed by Sussman and Puckett (2000) and has been implemented by a number of researchers, e.g. Menard et al (2007) and Wang et al. (2009). In this fully coupled method transport equations are solved for both a level-set field and a VOF field. These are used together to reconstruct the interface within a cell. The level set provides a defined contour for the interface and a smoothly differentiable field while the VOF ensures mass conservation even on coarse meshes. Details of the CLSVOF method implemented in an in-house code for structured grids with no contact models can be found in Xiao (2012), and Xiao et al. (2013, 2014a,b). Yokoi (2013) applied the method using a Cartesian structured grid to the problem of droplet splashing, showing the method’s suitability for EWM type applications. Previously the CLSVOF method has been applied using orthogonal meshes. An interesting recent development was published by Arienti and Sussman (2014) in which they use a Cartesian adaptive grid with the CLSVOF method but include complex surface geometry by defining it as a second level-set function on the Cartesian grid. In this paper we present a method based on the Coupled Level-Set Volume of Fluid (CLSVOF) implemented such that it can be used in non-orthogonal or unstructured meshes. However in order for it to be used for EWM applications it will also need to include some method of modelling surface contact properties.

1.2. Surface contact modelling

With the surface contamination application, interaction between the liquid, gas and solid surface introduces additional complexity. The surface water flow will be affected by the different surfaces it flows over, for example automotive paintwork, glass, seals and possibly specially treated hydrophobic surfaces. Sui et al. (2014) provides a thorough review of the topic of the moving contact line problem. The motion of the contact line across the surface implies a contradiction with the no-slip boundary condition used in viscous flow CFD. This apparent contradiction must be resolved by the use of some physical modelling to include the effect of this singularity. A widely used method is to allow for a ‘slip length’ at the contact point, see for example Dussan (1979). As discussed by Sui et al. (2014) to fully capture all the physics involved requires resolving a very wide range of scales. To do this in a CFD calculation would require a very high mesh resolution, much higher even than that typically required for a DNS calculation of turbulent flow. For practical situations this will be prohibitively expensive.