

# Coupled Level-Set Volume of Fluid Simulations of Water Flowing Over a Simplified Drainage Channel With and Without Air Coflow

Mehriar Dianat, Maciej Skarysz, Graham Hodgson, Andrew Garmory, and Martin Passmore  
 Loughborough University

## ABSTRACT

The motivation for this paper is to predict the flow of water over exterior surfaces of road vehicles. We present simulations of liquid flows on solid surfaces under the influence of gravity with and without the addition of aerodynamic forces on the liquid. This is done using an implementation of a Coupled Level Set Volume of Fluid method (CLSVOF) multiphase approach implemented in the open source OpenFOAM CFD code. This is a high fidelity interface-resolving method that solves for the velocity field in both phases without restrictions on the flow regime. In the current paper the suitability of the approach to Exterior Water Management (EWM) is demonstrated using the representative test cases of a continuous liquid rivulet flowing along an inclined surface with a channel located downstream perpendicular to the oncoming flow. Experimental work has been carried out to record the motion of the rivulet in this case and also to measure the contact angle of the liquid with the solid surface. The measurements of the liquid/solid characteristics such as equilibrium and dynamic contact angles are described along with the analytical expression for contact angle vs. capillary number used in the CFD code. The results from the simulations are compared to experimental measurements. The simulations are carried out with air co-flows of 0, 0.5 and 10 m/s. The simulations are seen to reproduce physical phenomena such as the liquid pinning at sharp corners and the longitudinal stretching of the rivulet with higher air velocity.

**CITATION:** Dianat, M., Skarysz, M., Hodgson, G., Garmory, A. et al., "Coupled Level-Set Volume of Fluid Simulations of Water Flowing Over a Simplified Drainage Channel With and Without Air Coflow," *SAE Int. J. Passeng. Cars - Mech. Syst.* 10(1):2017, doi:10.4271/2017-01-1552.

## INTRODUCTION

Predicting and controlling the motion of rainwater over the exterior of road vehicles, EWM, is an important consideration for vehicle designers. Phenomena such as A-pillar overflow can obscure a driver's vision and droplets or rivulets falling on passengers or their belongings can be a significant source of customer dissatisfaction. There are a wide range of engineering applications involving the flow of liquid films, rivulets and droplets over solid surfaces. For example, water or ice on an airplane wing can significantly interfere with its overall performance [1, 2]. Rivulets are also observed in coating flows in the chemical industry [3]. In the automotive sector examples of EWM simulations can be found in [4] and [5] that both use Lagrangian particle tracking for the airborne droplets and a simple 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 has limitations. These 2D film models, details of which can be found in [6] for example, solve transport equations for the film thickness but make no attempt to resolve the 3D shape of the surface water. Using such film methods behaviour such as film-stripping and breakup can only be modelled empirically, with the results being valid within certain flow regimes. More fundamentally the two-way interaction between the aerodynamic drag on the water and the shape of the water feature cannot be predicted using such methods.

The objective here is to develop and demonstrate a computational method that can capture the underlying physics and predict the motion of various liquid flow features on the exterior surface of vehicles. By predicting the resolved 3D shape of the droplet it is hoped that this will be more general than existing thin film models. By resolving water features fully in 3D it is possible to predict droplet breakup or coalescence without the need for empirical modelling as is the case for film models. The most common numerical method for predicting interface resolved two-phase flows is the Volume of Fluid (VOF) method where a conservative transport equation is solved for the fraction of liquid in each cell. See [7] for a review of the approach. Most commercial codes [8] have an algebraic VOF solver available (where a typical discretisation method is used to convect the VOF value), often with special numerical techniques for sharpening the interface between phases. Despite these techniques, algebraic VOF simulations suffer from numerical diffusion that causes the interface to lose definition; this reduces the accuracy of the predicted shape that in turn affects terms such as surface tension that rely on accurate predictions of surface curvature. A further problem is the choice of interface VOF value. A value  $0 < \text{VOF} < 1$  must be chosen to define the interface but this choice is essentially arbitrary and the value chosen will affect the visualisation of results. Small liquid flow features such as drops can appear to vanish as the VOF value drops below the threshold due to numerical diffusion. Various improvements to the VOF approach have been

made using more specialist codes, such as ‘geometric VOF’ (see [7] for details) where the gradient of the VOF field is used along with its local value to define the position of the interface. A VOF approach is combined with a Lattice-Boltzmann solver in [9] which is able to keep the interface (where  $0 < \text{VOF} < 1$ ) confined to a single layer of cells. However, all VOF methods suffer from the difficulty of finding a surface normal and curvature from what is ideally a binary field. This can reduce accuracy in the evaluation of the effect of surface tension. An alternative is to use Level-Set (LS) methods where a transport equation is solved for a signed distance function from the interface [10]. The zero level-set gives a well-defined and sharp interface, allowing the curvature to be found more easily, but as the transport equation is not conservative mass conservation is not guaranteed and high mesh resolution is required. The method used here is a CLSVOF [11] method where both LS and VOF fields are used together to give a method that combines good interface definition and calculation of curvature with mass conservation.

Surface material plays an important role in liquid transport phenomena and the degree to which a material is *hydrophilic* or *hydrophobic* will affect how the water behaves. Here the approach is taken of setting the contact angle, as a function of contact line velocity, as a boundary condition using experimental data. To demonstrate the use of our CLSVOF method with contact angle boundary conditions we apply it here to a laboratory test case representative of EWM applications; the flow of a rivulet towards a channel. This is done firstly with the droplet moving under gravity before the simulations are changed to include an air co-flow velocity. An air velocity of 0.5 m/s was found to have little effect but a velocity of 10 m/s shows significant differences. The method used is described in the following section. Experimental details of the laboratory case used for comparison as well as contact angle measurements are described in Section 2. Finally, Section 3 provides the results of the simulations and some comparison with the experimental data.

## COMPUTATIONAL METHOD

The method is based on a CLSVOF algorithm for interface tracking [11]. It has been implemented [12] into the general purpose open source solver OpenFOAM [13] using a formulation suitable for general unstructured grids. Details of the CLSVOF method implemented in an in-house code for structured grids with no contact models can be found in [14], and [15, 16, 17, 18]. The topology of the two-phase flow is represented by two properties; VOF represents the liquid volume fraction and LS is the shortest distance from the interface, positive within the liquid, negative outside and zero on the interface.

The solution domain is first initialised with the initial VOF and LS conditions. The interface is then reconstructed from LS and VOF fields so that the position of the interface within a computational cell and where it intersects cell faces is found. Interface is represented by Piecewise-Linear Interface Calculation (PLIC) with its normal obtained from the LS field and with its location adjusted to give the same volume fraction obtained from the VOF solution. In this way a defined interface position, consistent with both fields, is found and the fraction of each face occupied by liquid can be found from the

intersection of the face and the interface plane. This is used to define an Area of Fluid (AOF) as the area of the face occupied by liquid, rather than using traditional discretisation methods. Advection equations are then solved for both VOF and LS.

$$\frac{\partial \alpha}{\partial t} = -\frac{1}{V} \sum_i AOF_i \cdot U_i \cdot S_{f,i}$$

1

$$\frac{\partial \phi}{\partial t} + \nabla(U\phi) = 0$$

2

Here  $V$  is the volume of the cell and summation is over all faces in the cell,  $S_f$  is the face normal vector and  $U$  is the velocity vector. AOF is the area of the face occupied by liquid calculated by the intersection of the face and the interface. The VOF equation is solved using an explicit first order Euler temporal solver. The equation for the LS is solved implicitly using a limited Total Variation Diminishing (TVD) scheme for spatial discretisation. Furthermore, a re-initialisation procedure is employed at every time step to ensure that LS stays a distance function in the vicinity of the interface. This is done following the method of Sussman et al [10].

The velocity field is then found by solving the momentum and pressure equations using the OpenFOAM pressure-velocity PIMPLE correction procedure. The PIMPLE algorithm is the merged PISO-SIMPLE predictor-corrector solver for large time step transient incompressible laminar or turbulent flows. It is based on an iterative procedure for solving equations for velocity and pressure. PISO (Pressure Implicit Split Operator) is a transient solver and SIMPLE (Semi Implicit Method for Pressure Linked Equations) is a steady-state solver for incompressible flows. A single set of momentum equations are solved for both phases. The following surface tension effect in the momentum equations is based on the Continuum Surface Force (CSF) [19] with the curvature term calculated from the LS field.

$$f_\sigma = \sigma \kappa \nabla \alpha$$

3

where  $\sigma$  is the surface tension coefficient of liquid in gas and  $\kappa$  is the mean curvature of the free surface calculated from the following equation

$$\kappa = -\nabla \cdot \frac{\nabla \phi}{|\nabla \phi|}$$

4

The variable time step option available in OpenFOAM [13] was used with the maximum global and interface Courant numbers set to 0.5.

The local density and viscosity are determined using volume weighted averages found from the VOF field. To implement the contact angle model into the CLSVOF formulation the method already available for a generic contact angle model for the interFoam VOF solver in OpenFOAM is followed. This is done by setting the value of LS and VOF for the wall face of an interface cell such that

the magnitude of their gradients normal to wall are equal to the cosine of the desired contact angle. A key feature of an interface resolving method such as this is that a single velocity and pressure field is solved in space with no restrictions on the topology of the flow (e.g. film, droplets, rivulets etc.).

### Computational Domain and Boundary Conditions

The test case involves the flow of a continuous liquid rivulet along an inclined solid surface of slope  $7^\circ$  with a channel of width 12.5 mm and height 2.5 mm located downstream perpendicular to the oncoming flow. The computational domain is shown in Figure 1. It consists of liquid/air inlets on the left, outflow on the right, solid walls on the floor, side walls or symmetry in the spanwise z-direction and pressure on the top boundary. The floor upstream of the channel has an axial length of 50 mm. Downstream of the channel it is 22.5 mm. The width in the spanwise direction is 30 mm and the vertical height is 12.5 mm. A total hexa mesh of just over a million cells has been used. The grid spacing in the proximity of the walls is around 0.1 mm. The mesh is generally non-uniform being refined near the surface, in the regions close to the channel and in the central region in the spanwise direction where liquid rivulet flows. A typical mesh is shown in Figure 1.

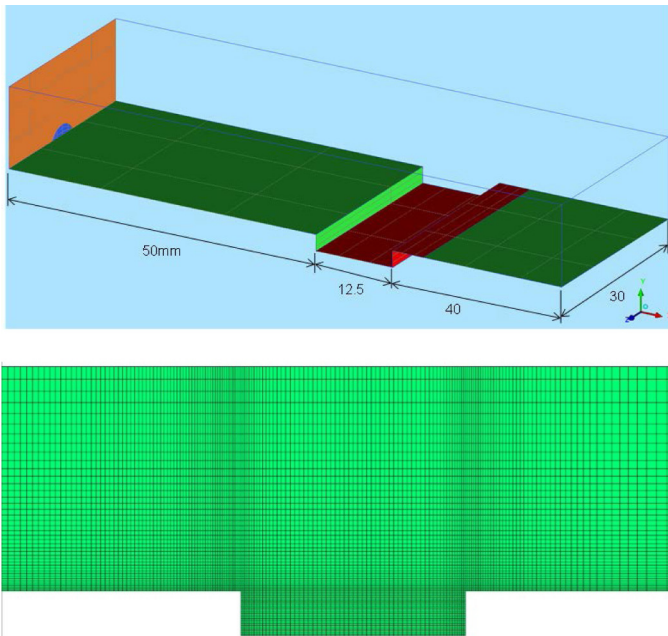


Figure 1. Computational domain and mesh

The liquid is injected through an inlet at the experimentally measured volumetric flow rate of 1 ml/s. At the rivulet front, the measured velocity, once it attains a constant speed, is approximately  $U_L = 16$  cm/s. Assuming the same velocity at the liquid inlet, the inlet area is then  $6.25 \text{ mm}^2$ . The measured equilibrium contact angle of the upstream surface is  $58^\circ$ . To make the shape of the inlet area consistent with the contact angle, a circular cap is used with the same area and the same measured contact angle at liquid/solid interface. This will lead to a rivulet thickness of around 1.6 mm and width of around 5.7 mm. The liquid Reynolds number is  $Re_L \sim 909$  and the liquid Weber number is  $We_L \sim 2$  where

$$Re_L = \frac{U_L W}{\nu_L}$$

5

and

$$We_L = \frac{\rho_L U_L^2 W}{\sigma}$$

6

Here  $\rho_L$  and  $\nu_L$  are respectively the liquid density and kinematic viscosity,  $W$  is the rivulet width at the inlet and  $\sigma = 0.072 \text{ N/m}$  is the surface tension coefficient at water/air interface. The Bond number is around  $Bo = 4.4$  where

$$Bo = \frac{\rho_L g W^2}{\sigma}$$

7

with  $g$  being the gravitational acceleration. Three different conditions were investigated for the air inlet: no air flow, laminar low speed air flow and turbulent higher speed air flow. The former is the test case for which measured data are available while the latter cases were simulated to study the effects of air flow on the rivulet behaviour. With the laminar case, a boundary layer thickness at the inlet was assumed using the Blasius profile

$$\delta \approx 5 \sqrt{\frac{\nu x_{off}}{U_0}}$$

8

where  $x_{off} = 50 \text{ mm}$  is the offset distance from the start of the boundary layer and  $U_0 = 0.5 \text{ m/s}$  is the free stream velocity. This yields a thickness of around 6 mm compared to the rivulet thickness of around 1.6 mm. The velocity profile was approximated by a quadratic fit within the boundary layer thickness. With turbulent air flow, using the same offset, a boundary layer thickness was assumed using

$$\delta \approx \frac{0.37 x_{off}}{Re_x^{0.2}}$$

9

Where

$$Re_x = \frac{U_0 x_{off}}{\nu}$$

10

is the Reynolds number. For this case, the free stream velocity is now 10 m/s. This would give a thinner boundary layer thickness of around 2.3 mm. For the mean velocity, the 1/7th power-law profile was fitted. This can be thought of as representative of the early part of a boundary layer on a vehicle moving at moderate road speed. In addition, random perturbations with Gaussian distribution were added to all three components of the velocity with the standard deviation of 10 % of the free stream velocity. Future work will investigate the use of fully correlated synthetic inlet turbulence but this is not available