Journal of Computational and Applied Mathematics 281 (2015) 277–283

Contents lists available at ScienceDirect

Journal of Computational and Applied Mathematics

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

Assessment of dynamic adaptive grids in Volume-Of-Fluid simulations of oblique drop impacts onto liquid films

P. Brambilla, A. Guardone *

Department of Aerospace Science and Technology, Politecnico di Milano, via La Masa 34, 20156 Milano, Italy

ARTICLE INFO

Article history:
Received 10 November 2013
Received in revised form 10 June 2014

Keywords:
Drop impact
Two-phase flow
Dynamic adaptive grids
Volume-Of-Fluid
OpenFOAM

ABSTRACT

Grid spacing dependence in three-dimensional numerical simulations of non-normal drop impact onto thin liquid films is assessed for different impingement angles and grid refinement levels. To describe the liquid phase dynamics, the Navier–Stokes equations are coupled to a Volume-Of-Fluid (VOF) model. Numerical simulations are performed with a modified version software OpenFOAM over a structured grid of hexaedra. Grid adaptation is carried out using an edge subdivision technique which results in non-conformal meshes. Grid convergence is assessed by monitoring integral parameters describing the dynamics of the post-impact free-surface waves. Starting from an initial grid spacing between $D/8$ and $D/5$, with D drop diameter, a refinement level of three is found to be sufficient to describe the diverse flow feature and to identify the splashing regime.

1. Introduction

The evaluation of the dynamics of a liquid drop impacting onto a liquid film of the same fluid is of paramount importance in both scientific and technical applications, including for example chemicals production, ink jet printing and fuel injection in combustion engines. Worthington was the first to investigate drop splashing into liquid film in 1908 [1]. More recently, from experimental results, Yarin and Weiss introduced a criterion to characterize normal drop impacts and an empirical relation describing the evolution of the radius of the crown in time [2]. Oblique drop impacts of interest here were investigated experimentally in [3,4].

The numerical simulations of drop impact required the development of multi-phase solvers capable of capturing the drop–film interaction, the occurrence of secondary droplets and the formation of dry regions over the solid surface. Two-dimensional or axisymmetric numerical simulations of normal drop impact were performed in [5,6]. Oblique drop impacts were simulated under the two-dimensional approximation in [7]. In 1999, Rieber and Frohn were the first to perform three-dimensional numerical computations of drop impacts whereby the drop trajectory is normal to the film surface [8]. In 2007, Nikolopoulos and collaborators simulated the same conditions studied by Rieber and Frohn using an adaptive grid technique [9]. Both Refs. [8,9] used a Volume-Of-Fluid (VOF) technique to study normal drop impacts. Finally, in 2012, Brambilla and collaborators simulated for the first time oblique drop impacts onto liquid films in three spatial dimensions using dynamic adaptive grids [10].

The evaluation of the splashing dynamics in drop impact simulations calls for accurate spatial and temporal integration schemes as well as a very fine local spatial and temporal resolution to capture the liquid–gas interface. In particular, a fine grid spacing is mandatory if secondary droplet and corona breaking are to be correctly captured. In the present paper, the

* Corresponding author.
E-mail address: alberto.guardone@polimi.it (A. Guardone).

http://dx.doi.org/10.1016/j.cam.2014.12.030
0377-0427/

grid spacing requirements for normal and oblique drop impact simulations are assessed in connection to a novel dynamic adaptive grid VOF scheme implemented in the open-source CFD solver OpenFOAM (www.openfoam.org), see Ref. [10]. Grid convergence is monitored using the integral parameters describing the dynamics of the post-impact free-surface waves introduced in [10].

This paper is structured as follows. In Section 2 the governing equations for the two-phase flow under scrutiny are introduced and the main features of the OpenFOAM solver are briefly reported. In Section 3, simulations results are discussed and a grid convergence study is presented. In Section 4, final remarks and comments are given.

2. Governing equations and multi-phase solver

In the present work, the Volume-Of-Fluid (VOF) method of Hirt and Nichols [11] – a volume tracking method in which the interface is not described as a sharp discontinuity but instead is represented by an indicator function α, namely, the volume fraction of the dispersed phase – is used. As described in [12], in the OpenFOAM implementation the VOF scheme is rigorously derived from the so-called Euler–Euler approach, in which the Navier–Stokes equations are solved for two phases, which amounts to complement the single-phase model with an additional conservation law for the local concentration of the liquid phase. Therefore, the gas–liquid interface is an output of the solution procedure and its shape can be reconstructed a posteriori by free-surface methodologies.

In each control volume, the volume fraction α varies in the interval [0, 1], where

$$\alpha = \begin{cases} 1 & \text{Liquid only} \\ 0 < \alpha < 1 & \text{Liquid–gas interface} \\ 0 & \text{Gas only.} \end{cases} \tag{1}$$

The Navier–Stokes equations for incompressible flows of two immiscible fluids in the Euler–Euler formulation read

$$\begin{cases} \dfrac{\partial(\rho \boldsymbol{v})}{\partial t} + \nabla \cdot (\rho \boldsymbol{v} \otimes \boldsymbol{v}) = -\nabla p + \nabla \cdot \Sigma + \rho \boldsymbol{f} + \displaystyle\int_{S(t)} \sigma k' \boldsymbol{n}' \delta(x - x') dS \\ \dfrac{\partial \alpha}{\partial t} + \nabla \cdot (\alpha \boldsymbol{v}) = 0 \\ \nabla \cdot \boldsymbol{v} = 0 \end{cases} \tag{2}$$

where ρ is the density, $\boldsymbol{v}$ is the velocity vector, p is the pressure, $\boldsymbol{f}$ is the acceleration due to the volume forces, Σ is the stress tensor, σ is the surface tension coefficient, k is the surface curvature and $\boldsymbol{n}$ is the local normal. In incompressible flows of Newtonian fluids, $\Sigma = \mu[\nabla \otimes \boldsymbol{v} + (\nabla \otimes \boldsymbol{v})^{\mathrm{T}}]$, where μ is the kinematic viscosity. According to the VOF approximation considered here, the local values of ρ and μ in system (2) are computed as a linear combination of the liquid and gas values as

$$\rho = \alpha \rho_{\mathrm{L}} + (1 - \alpha)\rho_{\mathrm{G}} \quad \text{and} \quad \mu = \alpha \mu_{\mathrm{L}} + (1 - \alpha)\mu_{\mathrm{G}},$$

where the subscripts L and G indicate liquid and gas values, respectively. To model the effects of surface tension (the last term of the first equation of system (2)), the Continuum Surface Force (CSF) model of Brackbill et al. [13] is used.

System (2) governing the fluid motion is integrated using the solver *interDyMFoam* included in the OpenFOAM suit, which implements the VOF method. The solver allows to use a dynamic grid refinement technique through the *dynamicMesh* utility. The refinement technique is based on Jasak's and Jasak and Gosman's h-refinement approaches [14,15]. At each refinement step the edge of the cell is split into two in every direction, so that eight new cells are inserted in place of the initial cell. The cells marked for refinement are those containing the interface, that is with a value of α between 0 and 1. The *blockMesh* dictionary is used to generate the initial grid and the *setFields* dictionary sets non-uniform initial conditions such as for the phase fraction α in this case. The initial conditions are assigned on the unrefined grid. The modified procedure presented in [10] is used to refine the grid across the interface before the computation starts. In particular, the initial grid is refined using the initial solution until the maximum level of refinement is attained at the gas–liquid interface. Then, the initial condition is re-applied in the refined mesh and the computation starts.

The artificial compression term in the interFOAM solver is activated to reduce the thickness of the gas–fluid interface. A first order implicit backward Euler scheme is used to approximate the time derivative terms, a second order Gaussian integration is performed to compute the spatial gradient, the Laplacian and the advection terms. A linear interpolation scheme is used during mesh adaptation. In all computations the Courant–Friedrichs–Lewy number is equal to 0.3. Courant–Friedrichs–Lewy number was kept ≤ 0.3, as suggested by OpenFOAM developers for 3D simulations, see [16].

3. Numerical simulations of oblique drop impacts

Numerical simulations of the oblique impact of a liquid drop onto a liquid film are presented in this section. The numerical simulations cover a range of impingement angles β from 10° to 90°. In all cases under scrutiny, the Weber number We is equal to 250, where

$$\mathrm{We} = \frac{\rho_{\mathrm{d}} D V_{\mathrm{d}}^2}{\sigma},$$