International Journal of Heat and Fluid Flow 000 (2016) 1–11

Contents lists available at ScienceDirect

International Journal of Heat and Fluid Flow

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

A numerical study on droplet-particle collision dynamics

Ilias Malgarinos[a,*], Nikolaos Nikolopoulos[a,b], Manolis Gavaises[a]

[a] School of Mathematics, Computer Science & Engineering, City University London, Northampton Square, EC1V 0HB London, UK
[b] Centre for Research and Technology Hellas, Chemical Process and Energy Resources Institute, Egialeias 52, Marousi, Athens, Gr-15125, Greece

ARTICLE INFO

Article history:
Received 13 January 2016
Revised 10 May 2016
Accepted 22 June 2016
Available online xxx

Keywords:
VOF
Impingement
Droplet
Particle
Collisions
Local refinement

ABSTRACT

The impact of liquid droplets onto spherical stationary solid particles under isothermal conditions is simulated. The CFD model solves the Navier-Stokes equations in three dimensions and employs the Volume of Fluid Method (VOF) coupled with an adaptive local grid refinement technique able to track the liquid-gas interface. A fast-marching algorithm suitable for the quick computation of distance functions required during the grid refinement in large 3-D computational domains is proposed. The numerical model is validated against experimental data for the case of a water droplet impact onto a spherical particle at low We number and room temperature conditions. Following that, a parametric study is undertaken examining (a) the effect of Weber number ($= \rho u^2 D_o/\sigma$) in the range of 8 to 80 and (b) the droplet to particle size ratio ranging in-between 0.31 and 1.24, on the impact outcome. This has resulted to the identification of two distinct regimes that form during droplet-particle collisions: the partial/full rebound and the coating regimes; the latter results to the disintegration of secondary satellite droplets from elongated expanding liquid ligaments forming behind the particle. Additionally, the temporal evolution of variables of interest, such as the maximum dimensionless liquid film thickness and the average wetting coverage of the solid particle by the liquid, have been quantified. The present study assists the understanding of the physical processes governing the impact of liquids onto solid spherical surfaces occurring in industrial applications, including fluid catalytic cracking (FCC) reactors.

1. Introduction

The dynamics of droplet impingement onto solid surfaces is realised in many engineering applications, as for example, spray cooling, spray coating, fuel injection in internal combustion engines, fire suppression, inkjet printing, metallurgy and electronic circuits cooling among other. Comprehensive reviews and important numerical and experimental works on droplet impingement onto solid flat surfaces for isothermal and non-isothermal conditions have been presented (selectively) by (Antonini et al., 2012; Rein, 1993; Rioboo et al., 2002; Yarin, 2006). The main parameters affecting similar phenomena are clearly identified as those of a) the nature of the impinging surface, (solid surface roughness, wettability), b) the liquid and gas properties, c) the temperature of the surface, d) the impact velocity, and e) the droplet diameter. Selective experimental studies, where these parameters are investigated are presented in (Antonini et al., 2012; Bayer and Megaridis, 2006; Chandra and Avedisian, 1991). The outcomes range from spreading and relaxing, to rebounding and splashing, while it becomes more complex for impingement on hot and/or rough surfaces (Pasandideh-Fard et al., 2002a).

However, unlike droplet impingement onto solid flat surfaces, very little concrete experimental and verified numerical information is available for droplet impact onto non-flat spherical surfaces and especially for droplet-particle collisions. A review of the relevant works (experimental and numerical) on this subject starts from 1971, when Levin and Hobbs, (1971) published the first (to the best of the authors knowledge) experimental work on droplet impingement onto a spherical surface, under the effect of gravity, where the regime of the impingement included droplet splashing. In 1999, Hardalupas et al. (1999) also studied droplet splashing onto spherical targets.

More recently, in 2003, Gunjal et al. (2003) used the VOF methodology in 2D computational domains, in order to simulate the liquid shape formations after the off-center impingement of a liquid droplet onto a spherical surface, using experimental data from own experimental campaigns. However, the simplification of the phenomenon in two dimensions (planar) was not enough for the accurate representation of the physics that lie behind this asymmetric impingement.

In 2007, Bakshi et al. (2007) performed experiments for the coating of a spherical solid particle by a droplet of similar size.

---

[*] Corresponding author.
E-mail address: Ilias.Malgarinos.1@city.ac.uk (I. Malgarinos).

http://dx.doi.org/10.1016/j.ijheatfluidflow.2016.06.010
0142-727X/© 2016 Elsevier Inc. All rights reserved.

Please cite this article as: I. Malgarinos et al., A numerical study on droplet-particle collision dynamics, International Journal of Heat and Fluid Flow (2016), http://dx.doi.org/10.1016/j.ijheatfluidflow.2016.06.010

Nomenclature

Acronyms

CFD	Computational Fluid Dynamics
cpR	Cells per Radius (cells that cover the drop radius)
DTP	Droplet To Particle size ratio
FCC	Fluid Catalytic Cracking
UDF	User defined Function
VOF	Volume of Fluid Method

Symbols

A	Area (m^2)
$Bo=\Delta\rho g Do^2/\sigma$	Bond number (-)
C_p	Pressure coeff. (-)
D	Diameter (m)
d	Distance (m)
$Fr=U_o/\sqrt{gD_o}$	Froude number (-)
H	Liquid Film Thickness (m)
n	Kolmogorov length scale (m)
$\hat{n}(\hat{n}_x, \hat{n}_y, \hat{n}_z)$	Free-surface unit normal (-)
P	Pressure (Pa)
R	Radius (m)
$Re=\rho_{liq}u_0D_0/\mu_{liq}$	Reynolds number (-)
$St=\rho_{liq}R_oU_o/\mu_{gas}$	Stokes number (-)
t	time (s)
$\tilde{u}(u, v, w)$	Velocity (its components) (m/s)
V	Volume (m^3)
$We=\rho_{liq}u_0{}^2D_0/\sigma$	Weber number (-)
$\vec{x}(x, y, z)$	Position vector [x,y,z-axis distance (m)]

Greek letters

α	Liquid volume fraction (-)
ε	Dissipation rate of turb. kin. energy per unit mass (m^2/s^3)
θ	contact angle (^{o})
μ	dynamic viscosity (kg/ms)
ν	kinematic viscosity (m^2/s)
ρ	Density (kg/m^3)
σ	surface tension coefficient (N/m)
τ ($=tu_o/D_o$)	Dimensionless time (-)
τ_n	Kolmogorov time scale (s)
Ω	Computational domain

Subscripts

C	cell
gas	gas
liq	liquid
m	mean
max	maximum
mom	refers to the momentum equation
o	initial condition
P	point
p	particle

This was one of the first studies, where the droplet to particle size ratio was close to unity and therefore, the “coating” outcome could be observed. The authors also presented an analytical model for the prediction of droplet spreading and presented results for the thickness of the liquid film at the impact side. During the same year, Ge and Fan (2007) presented Level-Set simulations and experiments on droplet impact on a spherical particle of similar size, in the Leidenfrost regime. In 2009, Bangonde et al. (2009) presented VOF simulations and experiments at low Weber number droplet impingement on a cylindrical pipe, followed also by the case of impact on a spherical surface. In 2012, Gac and Gradon (2014) used the Lattice-Boltzmann method to simulate the phenomenon and categorized the collision outcomes in three regimes, i,e, coalescence, ripping and coating (1–5 satellite droplets) and skirt-scattering, based on the initial droplet Weber number. The authors observed that the outcome scenarios don’t change significantly with other solid target shapes (cubical, ellipsoidal). Their work was purely numerical, i.e. no validation with experiments was presented. Moreover, the effect of droplet to particle size ratio was not explicitly investigated, as the authors differentiated collision outcomes based only on the impact Weber number. In 2013, Mitra et al. (2013) presented simulations and experiments on droplet impingement onto a spherical particle, under isothermal and non-isothermal conditions. In their work, they focus more on the effect of particle temperature on the solid-liquid contact and not on quantifying the collision outcomes. Finally, in 2014, Zhang et al. (2014) presented Lattice-Boltzmann Method (LBM) based simulations on droplet impact onto a spherical particle for low Reynolds and Weber numbers and moderate liquid to air density ratio. The authors investigated the effect of We and droplet-particle size ratio parameters on the outcome; however they only studied low Weber numbers (up to 26.14) which are below the threshold for different impact regimes to be realised. The authors also compared the non-dimensional film thickness at the impact side predicted from CFD against the correlations given in (Bakshi et al., 2007).

As it seems, studies concerning droplet-particle collisions are limited. A parametric study on the effect of droplet-particle size ratio for a wider range of Weber number of impact has not been presented, although it is needed for establishing collision outcome maps. The present study aims to fill this gap; it examines different types of collisions using CFD analysis that distinguish the impact outcome of influential parameters, such as the We number and the droplet-particle size ratio. The effect of wettability is not included in the current study as there is no experimental information regarding the contact angle values experienced during these impacts; moreover, performing further parametric studies that would examine the effect of wettability would significantly increase the computational effort. Despite that, the investigation of such phenomena may contribute to the understanding of similar processes happening in technological applications, such as for example FCC reactors used in petroleum industry in order to convert heavy hydrocarbon droplets into lighter gas products. However in the present work, the assumptions of stationary particles and the inclusion of gravity, which may not be directly relevant to FCC conditions, were mainly adopted for the validation of the proposed methodology against experimental measurements and relevant numerical works.

2. Numerical method

The numerical model used in the present paper is based on a previous work of the authors (Malgarinos et al., 2014), where it was used to simulate the droplet impact onto solid flat surfaces under isothermal conditions for a wide range of operating conditions. For completeness, in the following section, a short introduction on the basic principles of the CFD model utilized is presented, followed by the description of the advancements made here in the dynamic local grid refinement model.

2.1. Fluid Flow and volume of fluid method (VOF)

The CFD model utilized in this work includes the solution of the Navier-Stokes equations for the prediction of fluid flow, while the liquid-gas interface is tracked using the Volume of Fluid Method (VOF) (Hirt and Nichols, 1981). The equations are solved in the commercial software ANSYS FLUENT (FLUENT 2011). In (Malgarinos

Please cite this article as: I. Malgarinos et al., A numerical study on droplet-particle collision dynamics, International Journal of Heat and Fluid Flow (2016), http://dx.doi.org/10.1016/j.ijheatfluidflow.2016.06.010