eprints@whiterose.ac.uk
https://eprints.whiterose.ac.uk/

1 CFD BASED OPTIMISATION OF THE SUPERHYDROPHOBIC FUNCTIONAL SURFACE IN DROPWISE CONDENSATION

Z. Khatir[@a]**, K.J. Kubiak**[*b]

[a]**Curtin University, School of Engineering and Science, Department of Mechanical Engineering**
Sarawak Campus CDT 250, 98009 Miri, Sarawak, Malaysia
[b]**University of Liverpool, School of Engineering, Liverpool L69 3GH, United Kingdom**
[@]*zkhatir@hotmail.com* [*]*k.kubiak@liverpool.ac.uk/kris@kubiak.co.uk*

Abstract: Dropwise condensation processes, where condensation occurs through small droplets on a solid surface, has been demonstrated to significantly improve heat transfer rates in comparison to film-wise condensation (where a whole surface is covered by a thin film of liquid). Dropwise condensation usually takes place on superhydrophobic surfaces, and one of the main engineering and scientific challenges is to develop such a surface to allow condensation, and removal of the droplets, to occur in a continuous manner. Experimentally validated CFD analysis of droplets on superhydrophobic surfaces is undertaken to describe the mechanism of self propelled droplets 'jumping' from the surface after coalescence, the process by which two droplets merge during contact to form a single droplet. Owing to the larger diameter of the newly created droplet, it starts to push against the surface and spread beyond static contact angle usually observed between the fluid and the solid surface in equilibrium. Spreading beyond the equilibrium state creates a reaction at solid-liquid interface and shifts the momentum upwards, generating unbalanced kinetic energy and propelling the droplet. This allows the liquid to overcome the adhesion forces and dewet the surface separating droplet from surface. Optimisation of droplets radius and contact angle to obtain high 'jumping' droplet velocity, hence improving droplets evacuation, was performed using design of experiment and surface response techniques. CFD-based optimisation is proven to be a reliable and fast predictive modelling tool.

Key words: dropwise condensation; heat transfer; superhydrophobic surfaces; self propelled drops; CFD optimisation; surface response.

1 INTRODUCTION

The so called dropwise condensation when liquid condense on a solid surface in a form of droplets has been studied for several decades and there has been a huge effort to enhance the heat transfer from vapour to liquid state using dropwise condensation which shows much higher heat transfer rate than filmwise condensation (Lee2013) when surface is covered by a liquid film. In dropwise condensation crucial task is to efficiently remove condensate from the surface and create place for new droplets nucleation. This is where the droplets coalescence and self propelled ('jumping') phenomenon can enhance this process.

Droplets coalescence is a complex physical phenomenon which despite decades of research is still not fully understood. Recent developments in experimental and numerical techniques (Sellier2009; Blanchette2010; Fezzaa08) allows to investigate very early stage of coalescence below $100\mu s$ and electrical method adopted by Case and Nagel allows even shorter times in bridge creation regime below $10ns$ (Case2008). Recent interest in hydrophobic and superhydrophobic surfaces which are hardly wetted by the water, increased the research in this area and especially research related to droplets on functional surfaces. It has been presented that energy released upon coalescence is sufficient to overcome the barrier between the Wenzel state (when liquid penetrates

the surface asperities) and Cassie-Baxter state (when liquid stays on top of the asperities and air is trapped between liquid and solid) (Wang2011) and in some cases to fully dewet the surface allowing the droplet to 'jump'.

Process of droplet coalescence can be described by three flowing stages:

1. Nucleation of bridge, stage when the bridge is created and gap between the droplets rapidly start to close, while contact line start to move creating the neck,
2. Neck expansion stage when contact line move and composite droplet with 'peanut' like shape is created,
3. Composite droplet relaxation stage where free surface oscillate and droplet slowly relax to the equilibrium shape and finally to spherical cap shape.

The dynamics of coalescence in stage I and II is a few order of magnitude higher that in stage III. All of those processes involved in droplet coalescence has been modelled using different numerical tools and techniques. In our analysis, we use two different techniques. A Lattice Boltzmann Method (LBM) code with a newly implemented model for contact angle hysteresis and a Volume of Fluid (VoF) flow solvers are effectively applied.

2   EXPERIMENTAL VALIDATED CFD ANALYSIS

The dynamics of sessile droplet coalescence process was analysed numerically using LBM (Succi01) and VoF (Hirt-Nichols1981) as implemented in the OpenFoam CFD toolbox (OpenFOAM$^{©}$).

The LBM mesoscopic approach has the benefit that liquid free surfaces do not require special tracking or reconstruction at each time step, they arise naturally as part of the (liquid-gas) phase separation model, which in this case is the popular Shan-Chen single component, multiphase model (Shan93). More details about the LBM used in this study can be found in (Castrejon-Pita2011). On the other hand, the two-phase finite volume *interFoam* solver of the OpenFOAM open source CFD toolbox (OpenFOAM$^{©}$) was chosen because of its characteristics for automatic interface tracking and mass conservation). *interFoam* uses a VOF (Hirt-Nichols1981) approach modified with the introduction of an additional term in the volume fraction equation, to obtain interface compression by means of a tunable parameter as detailed in (Guilizzoni2012).

An experiment of the coalescence of two water droplets with radius $R = 405\mu m$ and static contact angle $\theta_S = 157^o$ is undertaken to validate both CFD methods. Fig. 2.1 displays the results for both CFD techniques together with experimental data. It is shown that both CFD methods show very good agreement with the experiment. Droplets coalesce to form a single droplet which then 'jumps' upwards due to the particular conditions.
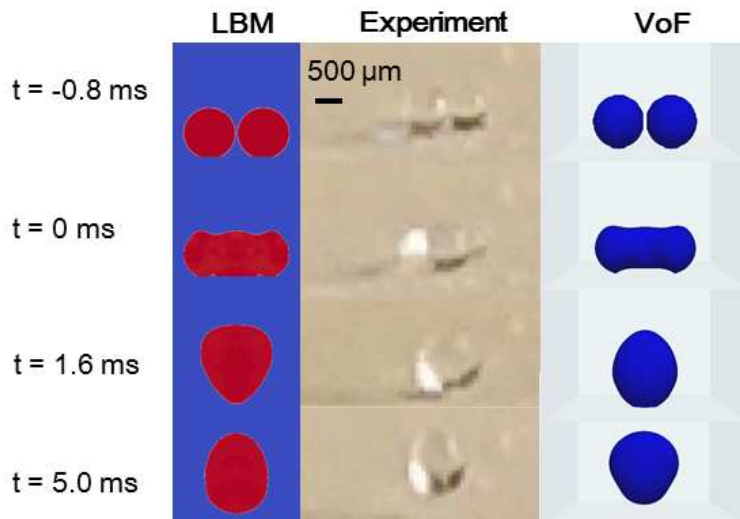

Figure 2.1   Coalescence of two water droplets with radius $R = 405\mu m$ and contact angle $\theta_S = 157^o$ at various time: comparison between LBM and VoF flow simulations with experimentation.

In the following, we will investigate the conditions by which such droplets jump after coalescence.

3   OPTIMISATION ANALYSIS AND RESULTS

We perform two simulation of the coalescence of droplets with radius R=100μm and static angles $\theta_S = 160^o$ and $\theta_S = 170^o$. Numerical results are presented in Fig. 3.1. They show that due to the larger diameter of the bigger combined droplet, it starts to press against the solid surface resulting in unstable conditions.

Surface tension is tending to minimize free energy of the system and change droplet back to its spherical shape. Release of the energy will follow Laplace ($\Delta p = \frac{\gamma}{R}$) resulting in lower pressure inside combined drop. However, gained during coalescence kinetic momentum will create droplet osculations and deformation beyond
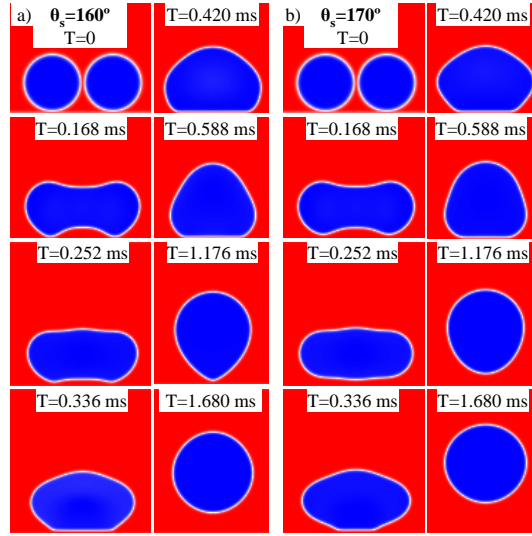
Figure 3.1   Influence of static contact angle on coalescing droplet shape and self propelling behaviour, LBM simulation (initial droplets radius R=100μm).
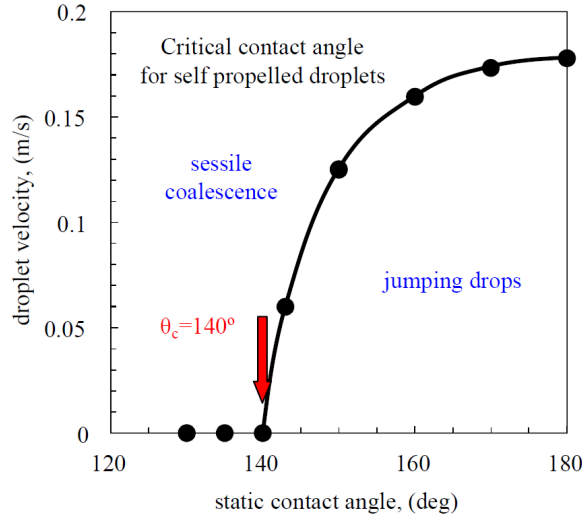

Figure 3.2   Critical contact angle ($\theta_C = 140°$) for self propelled behaviour of drops on superhydrophobic surfaces, numerical analysis for water droplets of initial radius $r = 100\mu m$.

spherical shape. When the droplet deforms and pushes against the solid surface, it can only escape upwards, shifting kinetic momentum in direction normal to the surface. When the static contact angle is high enough ($\theta_c > 140°$) the drop will 'jump' from the surface.

Droplet jumping phenomenon can be observed mainly on superhydrophobic surfaces, our numerical analysis on the contact angle influence on droplet velocity reveals the existence of a critical static contact angle around $\theta_c = 140°$ required to observe this phenomenon. In the results presented in Fig. 3.2, the velocity profile of jumping drop indicates that for lower contact angle an energy released upon coalescence is dissipated mainly by surface dewetting process, and for $\theta_S < \theta_C$ there is not enough energy left to separate droplet from the surface, and hence the combined droplet will stay attached to the surface after coalescence.

Optimisation analysis is undertaken to establish an optimal surface and droplets conditions to obtain maximum velocity of jumping droplets. The surface response method was used to analyse the influence of contact angle of solid surface and the droplet diameter. Design of experiment were used to define experimental conditions and several models were build using CFD solver, based on lattice Boltzmann method.

Optimisation consists of building a meta-model in function of the two design variables namely: the contact angle θ_S and initial droplet radius R. A second order non-linear polynomial approximation is employed involving the terms $\theta_S^2; R^2; \theta_S; R$ and $\theta_S \cdot R$, and the resulting meta-model for the jumping droplets velocity U_{drop} is thus obtained:

$$U_{drop} = -5.2 \times 10^{-5}\theta_S{}^2 + 0.09 \times 10^{-5}R^2 - 0.83 \times 10^{-5}(\theta_S\,R) + 0.021\,\theta_S + 0.0006\,R - 1.88 \qquad (3.1)$$

Results of this analysis and the estimated response surface are presented in Fig. 3.3.

It can be noted that optimum conditions concentrate in superhydrophobic region where contact angle is above $\theta_S > 160°$. Droplet radius shows optimum kinematic conditions for small droplets however this region
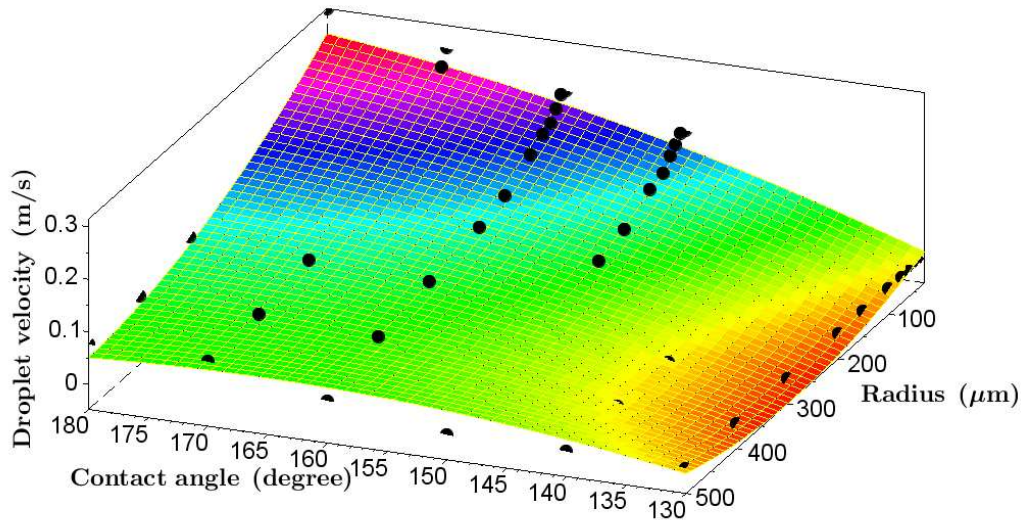
Figure 3.3 Results of surface response analysis based on LBM CFD-based model of droplets jumping upon coalescence. (Droplet velocity - m/s; Droplets Radius - μm; Contact angle - degree))

is rather technologically not yet accessible. Smaller droplet can jump from surface with higher initial velocity and bigger droplets have too large contact area to dewet the surface and they will remain attached.

4 CONCLUSION

A CFD based optimisation procedure has been applied for heat transfer enhancement in dropwise condensation process. The following conclusions can be formulated:

- Higher static contact angle (θ_S) results in higher velocity of droplets jumping after coalescence,
- Critical contact angle for droplets to 'jump' upon coalescence was estimated to be $\theta_C = 140°$,
- Optimisation of droplets size indicate that the maximum speed of jumping droplets can be achieved for smaller droplets $D < 130\mu m$,
- Superhydrophobic functional surface allowing condensation, coalescence and droplets 'jumping' can be a good strategy to enhance heat transfer rate in dropwise condensation process.

References

S. Lee, K. Cheng, V. Palmre, M.H. Bhuiya, K.J. Kim , B.J. Zhang, H. Yoon, (2013) *International Journal of Heat and Mass Transfer* Vol. 65, pp. 619-626.

M. Sellier and E. Trelluyer, "Modeling the coalescence of sessile droplets," *Biomicrofluidics*, vol. 3, no. 2, p. 022412, 2009.

F. M. C. Blanchette, "Simulation of mixing within drops due to surface tension variations," *Phys. Rev. Lett.*, vol. 105, p. 074501, Aug 2010.

K. Fezzaa and Y. Wang, "Ultrafast x-ray phase-contrast imaging of the initial coalescence phase of two water droplets," *Phys. Rev. Lett.*, vol. 100, p. 104501, Mar 2008.

S. C. Case and S. R. Nagel, "Coalescence in low-viscosity liquids," *Phys. Rev. Lett.*, vol. 100, p. 084503, Feb 2008.

F.-C. Wang, F. Yang, and Y.-P. Zhao, "Size effect on the coalescence-induced self-propelled droplet," vol. 98, no. 5, p. 053112, 2011.

S. Succi, *The Lattice Boltzmann Equation for Fluid Dynamics and Beyond*. Oxford University Press, 2001.

X. Shan and H. Chen, "Lattice boltzmann model for simulating flows with multiple phases and components," *Phys. Rev. E*, vol. 47, pp. 1815–1819, Mar 1993.

J. R. Castrejon-Pita, E. S. Betton, K. J. Kubiak, M. C. T. Wilson, and I. M. Hutchings, "The dynamics of the impact and coalescence of droplets on a solid surface," *Biomicrofluidics*, vol. 5, no. 1, p. 014112, 2011.

OpenFOAM©, The open source CFD toolbox, website: http://www.openfoam.com/.

M. Guilizzoni, "Drop deposition and low-speed impact on flat, curved and microfinned solid surface: comparison between simulations, models and experiments", DIPSI Workshop 2012 on Droplet Impact Phenomena & Spray Investigation, May 18, 2012, Bergamo, Italy.

C. W. Hirt, B. D. Nichols, "Volume of fluid (VOF) method for the dynamics of free boundaries", *Journal of Computational Physics*, 1981, 39, 201-225.