Numerical simulation of droplet impact onto structured surfaces

Rishav Saha* , Bernhard Weigand
Institute of Aerospace Thermodynamics, University of Stuttgart, Germany
*Corresponding author: rishav.saha@itlr.uni-stuttgart.de

Introduction

The interaction of droplets with a solid surface has been a topic of great interest in the diverse area of science and technology such as aerospace, electronics, energy and materials. In recent years, a tremendous increased interest has developed on micro-structured surfaces. Most of these studies are carried out as experimental investigations. In contrast, there is still only a limited number of numerical works, which has been done to understand the fundamental physical mechanisms taking place for a drop impact on such surfaces. However, continuously increasing computer capabilities make it now possible to analyse drop impacts on textured surfaces in great detail by using Direct Numerical Simulations (DNS) for multi-phase flows. Thus, this project aims to numerically study in detail drop impact and wetting behaviour on textured surfaces and its effects when the surface is heated.

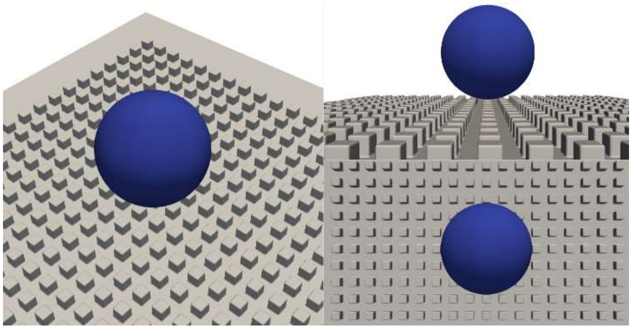


Figure 1. Numerical simulation of droplet impact on cubical micro-structured surfaces

Methods

In this study, a computational fluid dynamics framework is employed to simulate the impact of an Iso-Propanol droplet onto a superhydrophilic cubical micro-structured surface under ambient conditions. The simulation tool used is the in-house DNS code Free Surface 3D (FS3D) [1]. The gas and the liquid phase are defined by Volume-of-Fluid (VOF) method, whereas the solid phase (cubical microstructures) is defined by Cut-cell method [2], which was implemented in FS3D. Reconstruction of the interface is done using PLIC [3]. Incompressible Navier-Stokes equation is solved using Finite Volume method.

Simulation setup and assumptions

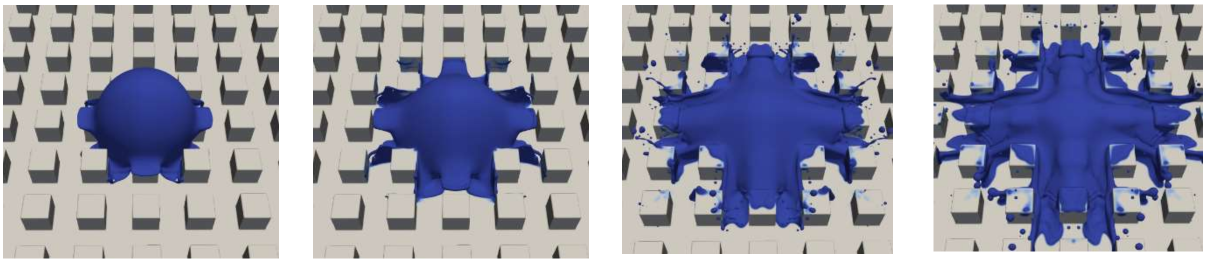
A spherical Iso-propanol droplet of 2mm impacts onto a cubical equally spaced non-heated micro-structured surface of sizes 500 µm (Case I) and 200 µm (Case II), at a velocity of 3.2 m/s and 1.7 m/s, respectively. The simulation is conducted to capture maximum spreading and splashing phenomenon. The droplet impacts exactly at the middle of a cubical pillars, therefore the spreading and splashing on the surface is considered to be symmetrical about the droplet impact direction. Hence, a quarter of the problem (one fourth of the droplet) is simulated to reduce the computational cost. The simulation domain size is 3.2Dx3.2Dx1.6D, where D represents the droplet diameter, with a resolution of 120 cartesian cells per droplet diameter, which is approximately 3.8 million cells in total. The simulations are conducted on AMD Ryzen 7 3700X 8-Core Processor.

Table 1. Simulation parameters for the two case with different pillar size, impact velocity and total simulation time.

	Pillar size [µm]	Impact velocity [m/s]	Total simulation time [ms]
Case I	500	3.2	1
Case II	200	1.7	4

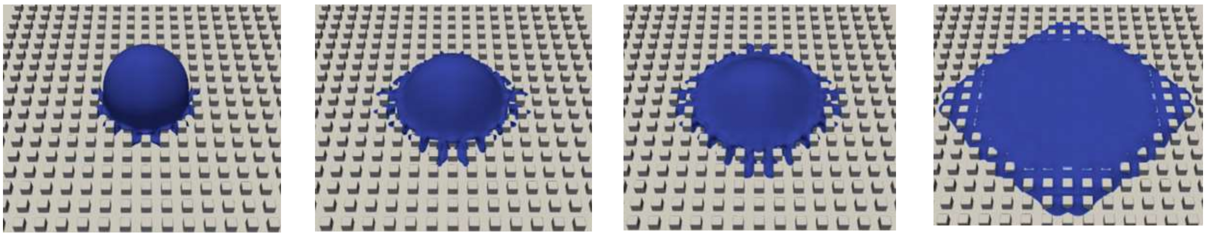
Results

The simulation result for the two mentioned cases show two different events, splashing and spreading. In figure 2 and figure 3, the splashing and spreading of a droplet impact at different time instances are shown, respectively.



(a) t = 0.3 ms (b) t = 0.4 ms (c) t = 0.6 ms (d) t = 0.8 ms

Figure 2. Case I: Numerical simulation of 2 mm droplet impacts at a velocity of 3.2 m/s, onto a cubical micro-structured surface of size 500 µm, at different time instances t.



(a) t = 0.4 ms (b) t = 0.6 ms (c) t = 0.8 ms (d) t = 3.2 ms

Figure 3. Case I: Numerical simulation of 2 mm droplet impacts at a velocity of 1.7 m/s, onto a cubical micro-structured surface of size 500 µm, at different time instances t.

In figure 2, secondary droplet formation is seen, which is formed because of the high impact velocity. Here, the inertial forces are dominant, whereas in figure 3, the droplet hits at a lower weber number on smaller micro-structures, which creates a quadrilateral shape while spreading, an interesting area of the on-going research.

References

[1] K.Eisenschmidt, M.Ertl, H.Gomaa, C.Kieffer−Roth, C.Meister, P.Rauschenberger, M.Reitzle, K.Schlottke and B.Weigand. Direct numerical simulations for multiphase flows: An overview of the multiphase code FS3D. Appl.Math. Comput.,272(2):508−517,2015.

[2] Baggio, Martina. Volume of Fluid Numerical Simulation of Drop Impact on Superhydrophobic Complex Solid Surfaces Embedded in a Staggered Cartesian Grid. Publisher Dr. hat, 2020.

[3] Rider, William J., and Douglas B. Kothe. "Reconstructing volume tracking." Journal of computational physics 141.2 (1998): 112-152.