DNS of droplet impact onto heated surfaces

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

Introduction

Droplet impact onto heated surfaces is a widespread process, which is encountered in many existing cooling technologies [1] used in industrial applications. These cooling technologies often involve multiple droplets, such as in sprays, but understanding their behaviour is complex. Therefore, we first need to study the impact dynamics and heat transfer mechanism of a single droplet impact onto a homogeneously heated substrate. This single droplet impact process can be influenced by numerous factors, including droplet parameters, liquid properties, surrounding gas parameters, and surface characteristics [2]. In this study, the impact dynamics of a water droplet impacting onto a highly conducting material is simulated. This work focuses on the temperature changes of the droplet's liquid during its spreading process within the film evaporation regime.

When a droplet at room temperature comes into contact with a heated surface, the liquid absorbs heat from the surface because of the temperature difference. This heat absorption results in a cooling effect on the heated surface, which is an advantageous strategy widely used in industrial applications [1]. Depending on the impact velocity, a droplet typically splashes, bounces or spreads onto the surface. Maximum heat transfer to the liquid is achieved when a droplet sticks to the surface for a longer period of time. However, the solid-liquid contact time and area is relatively higher for spreading droplets than droplet splashing and bouncing [3]. Therefore, investigating heat transfer during the spreading process is essential. Here, a water droplet impact onto an iso-thermally heated surface with a static contact angle of 90 deg. is simulated, from just before droplet-wall contact until the recoiling stage.

Methods

Direct Numerical Simulation (DNS) is conducted using the in-house multiphase flow solver code Free Surface 3D [4] (FS3D) running on a supercomputer of the High-Performance Computing Center Stuttgart (HLRS), HPE Apollo Hawk platform. The study is conducted both from the hydrodynamic and heat transfer perspective in a Computational Fluid Dynamic (CFD) framework. The incompressible Navier-Stokes equations are solved using the Finite-Volume method, where the interface is captured using the Volume of fluid [5] (VOF) method in a Cartesian grid. Separate temperature fields are solved for both phases [6]. The continuum surface force [7] (CSF) model is used, while the interface reconstruction is performed using the piecewise linear interface calculation (PLIC) method.

A quarter of a droplet, with an initial diameter of D_0 = 2 mm and an initial impact velocity of V_0 = 3 m/s, is initialised in a cubic domain of size $2D_0$. The surrounding ambient of the droplet is at 1 atmospheric pressure, and the initial temperature is 298 K. The gravitational force g acts in the downward direction. Continuous boundary conditions were applied to the far faces of the domain except for symmetric planes and the bottom surface. The bottom surface is assumed to be a highly conductive substrate, therefore, an isothermal boundary condition is enforced at a temperature of 363 K, below the boiling point of water. Evaporation is ignored as the time scale of evaporation is large compared to the spreading time scale. In figure 1, a schematic 2D diagram of the computational domain and a 3D visualization of the computational domain is shown.

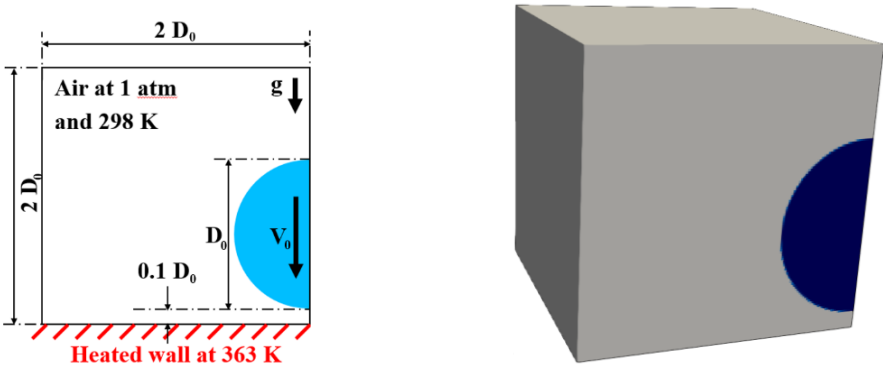

Figure 1. (Left) Schematic 2D diagram of the computational domain with domain configuration at the initial time. (Right) 3D visualization of the computational domain showing a quarter of the droplet.

Results and Discussion

The DNS were conducted for a simulation time of 3 ms, which corresponds to a dimensionless time of $\tau = 4.5$, where

$$\tau = (t\, V_0)/\, D_0 \qquad (1)$$

A line probe *L* is placed at the middle of the domain to record the temperature distribution along the z-axis at different time instances, for the gas liquid system, shown in figure 2. Grid sizes were increased systematically by a factor of two, from 256 to 1024 in each direction of the cubic domain, which resulted in a systematic increase in the total number of cells from two million to one billion. Hence, the number of cells per initial drop diameter was varied from 128 cells to 512 cells. To record the liquid temperature profile at the probe just after 1 dimensionless time, solution at $\tau = 1.25$ is used. Solution with grids 512^3 show convergences in the liquid temperature, shown in figure 2. It is observed that an increase of a maximum of 5 K is recorded in the liquid temperature until a maximum height of 40 µm above the surface. This change can be considered negligible for influencing the temperature-dependent material properties of water, such as viscosity, density, and surface tension.

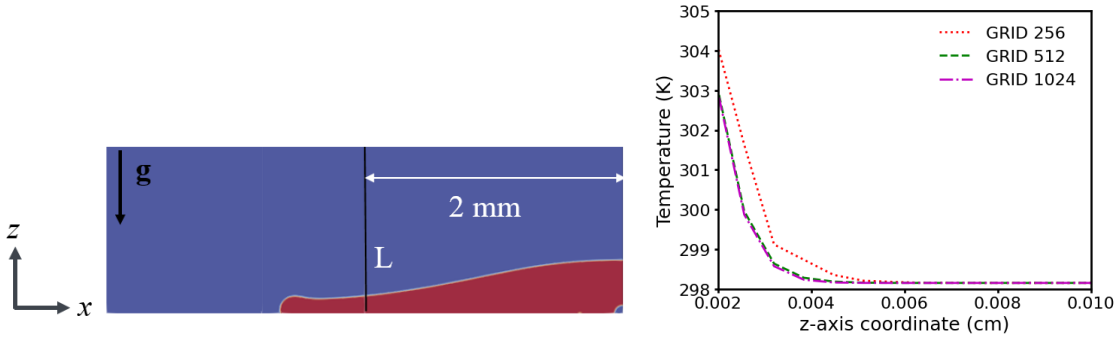

Figure 2. (Left) Location of the line probe *L* is shown at $\tau = 1.25$. (Right) Temperature profile for solution with grids 256^3, 512^3 and 1024^3, along the line probe *L* at $\tau = 1.25$.

Conclusion

A DNS study of the spreading process of a small water droplet impacting onto a heated surface is performed. A grid resolution of 512^3 was found to be sufficient enough to adequately resolve the impact dynamics and heat transfer in the liquid for the droplet considered in the present study. A line probe is utilized to record the increase in liquid temperature during spreading. As the droplet spreads, it forms a rim at the edge and attains maximum diameter, which is followed by a recoiling stage. It is observed that during spreading, only a very thin portion of the liquid film is heated, indicating that the heat conduction timescale is less than the spreading timescale for the water. The temperature independent material property assumption still holds, as the thermodynamic and fluid properties of water has a negligible change with a temperature difference of 5 K.

Future works

In this work, simplest configurations are used for the study of droplet impact onto a heated surface. However, the heat transfer mechanism for the process is very complex. To understand this, we need to consider different liquids such as, Heptane, Ethanol, Acetone and other engineering fluids, which has a lower specific heat compared to water, and volatile in nature. So that, the conduction timescale and the spreading timescale overlap and both the effects can be observed in the simulation, to have a greater influence in spreading and heat transfer.

Acknowledgments

We appreciate the financial support of this work by the Deutsche Forschungsgemeinschaft (DFG) in the framework of the International Research Training Group “Droplet Interaction Technologies” (GRK 2160/2: DROPIT) under project number 270852890. Furthermore, we would like to thank the *High-Performance Computing Center Stuttgart* (HLRS) for providing computation time on the HPE Apollo Hawk platform under Grant Number FS3D/11143.

References

[1] Jia, W., Qiu, H.H.: Experimental investigation of droplet dynamics and heat transfer in spray cooling. Exp. Therm. Fluid Sci. 27(7), 829–838 (2003)
[2] Liang, G., Mudawar, I.: Review of drop impact on heated walls. Int. J. Heat Mass Transfer. 106, 103–126 (2017)

[3] Josserand, C., Thoroddsen, S.T.: Drop impact on a solid surface. Annu. Rev. Fluid Mech. 48, 365–391 (2016)
[4] Eisenschmidt, K., Ertl, M., Gomaa, H., Kieffer-Roth, C., Meister, C., Rauschenberger, P., Reitzle, M., Schlottke, K., Weigand, B.: Direct numerical simulations for multiphase flows: An overview of the multiphase code FS3D. Appl. Math. Comput. 272, 508–517 (2016)
[5] Hirt, C.W., Nichols, B.D.: Volume of fluid (vof) method for the dynamics of free boundaries. J. Comput. Phys. 39(1), 201–225 (1981)
[6] Schlottke, J., Dulger, E., Weigand, B.: A vof-based 3d numerical investigation of evaporating, deformed droplets. Prog. Comput. Fluid Dyn. 9(6-7), 426–435 (2009)
[7] Brackbill, J.U., Kothe, D.B., Zemach, C.: A continuum method for modeling surface tension. J. Comput. Phys. 100(2), 335–354 (1992