NUMERICAL INVESTIGATION OF A LIQUID DROPLET IMPINGING ON A HEATED SURFACE

Andres Diaz  
Laboratory for Advanced Thermal and Fluid Systems  
Department of Mechanical Engineering  
Villanova University  
Villanova, Pennsylvania, USA

Alfonso Ortega  
Laboratory for Advanced Thermal and Fluid Systems  
Department of Mechanical Engineering  
Villanova University  
Villanova, Pennsylvania, USA

Ryan Anderson  
Laboratory for Advanced Thermal and Fluid Systems  
Department of Mechanical Engineering  
Villanova University  
Villanova, Pennsylvania, USA

ABSTRACT
Previous studies, most of them experimental, reveal that the cooling effectiveness of a water drop impinging on a heated surface depends on the wall temperature, droplet shape and velocity. All previous studies focus on the behavior of a droplet falling in a quiescent environment, such as still air. Evidence in the literature also shows that gas assisted droplet sprays, in which a gas phase propels the droplets, are more efficient in heat removal than sprays consisting of droplets alone. It is conjectured that this is due to an increase in the maximum droplet spreading diameter upon impact, a thinner film, and consequently an increase in the overall heat transfer coefficient. Recent experiments in the author’s group [1, 2] show that the carrier gas jet strongly influences droplet spreading dynamics by imposing normal and shear forces on the liquid surface. The heat transfer is greatly augmented in the process, compared to a free falling droplet. To date, there has been no fundamental investigation of the physics of gas assisted spray cooling. To begin to understand the complicated process, this paper reports on a fundamental problem of a single liquid droplet that impinges on a heated surface. This paper contributes a numerical investigation of the problem using the volume of fluid (VOF) technique to capture droplet spreading dynamics and heat transfer in a single drop event. The fluid mechanics is investigated and compared to the experimental data. The greatest uncertainty in the simulation is in the specification of the contact angle of the advancing or receding liquid front, and in capturing the onset of the three-dimensional fingering phenomena.

INTRODUCTION
A water droplet that impinges onto an isothermal or heated flat surface has been extensively studied. In investigations related to heat transfer, most of the studies have considered evaporation of liquid droplets that are gently deposited on heated horizontal surfaces [3-8]. Chandra et al. [3] studied the effect of liquid-solid contact angle on droplet evaporation. The contact angle was varied by adding a surfactant to the water droplets and then reducing the surface tension. It was shown that heat transfer can be enhanced by reducing the contact angle, which increases the contact area between the liquid and solid surface, and therefore the evaporation rate. Crafton and Black [4] studied the evaporation rate of small liquid droplets that were placed on surfaces that were maintained at temperatures below the saturation temperature. The results were compared with analytical models for evaporation and they showed that the values of heat flux decrease as the droplet size increases. Di Marzo et al. [5] studied the transient thermal behavior of a single water droplet that was deposited over the surface of a semi-infinite solid. In this study two models for the prediction of the thermal behavior of the droplet-solid interaction during evaporative cooling were proposed. The first one solved the liquid and the solid transient conduction equations simultaneously, and the second one introduced a constant and uniform heat flux boundary condition at the liquid and solid interface. Ruiz and Black [6] performed numerical studies of the evaporation process of small water droplets on hot solid surfaces taken into account the internal fluid motion that occurs as a result of the thermocapillary convection in the droplets and its effect on the heat transfer between the drop and the solid surface. Sodtke et al. [7] carried out experiments of liquid droplet evaporation on an electrically heated thin stainless foil, where the temperature distribution was measured using thermochronic liquid crystal. Others studies have investigated the evaporative cooling by the impingement of single water droplets [9-17]. Nikolopoulos et al. [9] and Strotos
et al. [10] implemented the volume of fluid method (VOF) to model the fluid mechanics and heat transfer during droplet impact. They also implemented an adaptive local grid refinement technique [18] to study numerically the flow development during normal impingement of droplets onto a hot wall. Tartarini and di Marzo [11] presented a theoretical model for predicting the thermal behavior of the droplet-solid interaction, where the solid surface was being heated by radiant panels from above. A few studies have investigated the heat transfer during droplet impact due to convection and conduction in single-phase. Healy et al. [19] numerically studied the effect of heat transfer on the spreading process to validate the assumption that many researchers have applied in previous studies where the droplet has been assumed to spread adiabatically over the surface. Pasandideh-Fard et al. [20] performed both experiments and numerical simulations using the VOF model to measure the temperature distribution within a water droplet and on a stainless steel surface. They also developed an analytical model to estimate the droplet cooling effectiveness. The present paper intends to continue the work of the author’s group [1, 2] and represents a preliminary investigation to study the correlation between the droplet dynamics and the instantaneous heat transfer, before studying the effects of a carrier gas.

NOMENCLATURE

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>$D(t)$</td>
<td>Instantaneous spreading diameter, mm</td>
</tr>
<tr>
<td>$D_{\text{max}}$</td>
<td>Maximum spreading diameter, mm</td>
</tr>
<tr>
<td>$D_i$</td>
<td>Droplet initial diameter, mm</td>
</tr>
<tr>
<td>$h$</td>
<td>Heat transfer coefficient, W/m$^2$-K</td>
</tr>
<tr>
<td>$k$</td>
<td>Thermal conductivity, W/m-K</td>
</tr>
<tr>
<td>$\hat{n}$</td>
<td>Surface normal</td>
</tr>
<tr>
<td>$\hat{n}_r$</td>
<td>Normal unit vector</td>
</tr>
<tr>
<td>$N_u$</td>
<td>Nusselt number ($N_u = hD(t)/k$)</td>
</tr>
<tr>
<td>$q''$</td>
<td>Heat flux, W/m$^2$</td>
</tr>
<tr>
<td>$Re$</td>
<td>Reynolds number ($Re = \rho v_D/\mu$)</td>
</tr>
<tr>
<td>$T_{\text{wall}}$</td>
<td>Wall temperature, °C</td>
</tr>
<tr>
<td>$T_{\text{ao}}$</td>
<td>Ambient temperature, °C</td>
</tr>
<tr>
<td>$t$</td>
<td>Time, s</td>
</tr>
<tr>
<td>$\hat{t}_w$</td>
<td>Tangential unit vector</td>
</tr>
<tr>
<td>$v_o$</td>
<td>Droplet impact velocity, m/s</td>
</tr>
<tr>
<td>$We$</td>
<td>Weber number ($We = \rho v_o^2 D_i/\sigma$)</td>
</tr>
</tbody>
</table>

Greek Symbols

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>$\alpha_n^k$</td>
<td>$k^{th}$ fluid’s volume fraction</td>
</tr>
<tr>
<td>$\mu$</td>
<td>Water Viscosity, kg/m-s</td>
</tr>
<tr>
<td>$\theta_s$</td>
<td>Contact angle, °</td>
</tr>
<tr>
<td>$\rho$</td>
<td>Water Density, kg/m$^3$</td>
</tr>
<tr>
<td>$\sigma$</td>
<td>Surface tension, N/m</td>
</tr>
</tbody>
</table>

EXPERIMENTAL AND NUMERICAL PROCEDURE

Water droplets were generated from a 0.9 mm diameter hypodermic needle, which was placed at different heights in order to obtain impact velocities of 1.7, 2.0 and 2.27 m/s. High-speed digital video images were captured with a Phantom V7.1TM color camera at a nominal framing rate of 4700 frames per second with an exposure time of 50 µs. The numerical test surface was selected in such a way that it represents a 25.4 x 25.4 mm$^2$ stainless steel (SS) with a thickness of 0.0254 mm, which is the test surface that the authors intend to employ in a future experimental investigation. The initial condition was set to be at the instant when the 3.0 mm diameter water droplet with initial temperature of 22°C first touches the surface initially at 60°C. The contact angle was assumed to have a constant value of 110° during the spreading process.

MATHEMATICAL MODEL

The computational simulations were carried out using a two-dimensional axisymmetric laminar model in FLUENT. The VOF model was implemented to track the interface between the liquid and gas, where a continuity equation for the volume fraction was solved:

$$\frac{\partial \alpha_n}{\partial t} + \vec{u} \cdot \nabla \alpha_n = 0 \quad (1)$$

The primary-phase volume fraction was computed based on Eq. (2)

$$\sum_{n=1}^{m} \alpha_n = 1 \quad (2)$$

Surface tension was included in the flow equations as a source term. Heat conduction through the stainless steel foil was solved simultaneously with fluid mechanics, where wall adhesion was incorporated as boundary condition at the solid-fluid interface by $\theta_w$, which is related to the surface normal as shown in Eq. (3)

$$\hat{n} = \hat{n}_w \cos \theta_w + \hat{t}_w \sin \theta_w \quad (3)$$

Figure 1a shows a schematic of the boundary conditions employed in the simulations. A grid refinement technique was implemented every time step to improve the accuracy of the simulations at the liquid-gas interface and measure the temperature distribution inside the water droplet, ensuring that a grid independent solution was achieved. Approximately 20000 cells at the beginning of the simulation were used for 2 levels of grid refinement (Fig. 1b). First-order implicit formulation was used for time discretization, where the time step needed for carrying out the simulation without affecting the results was $1 \times 10^{-6}$ s. The pressure and velocity were coupled with pressure implicit with splitting of operator (PISO). An implicit body force formulation was used in the multiphase model, where the face fluxes for the VOF model were calculated using the geometric reconstruction scheme. A total of three computational simulations were performed for varying Re and We.
RESULTS AND DISCUSSION

In this section, the experimental and numerical results for a 3 mm diameter water droplet impinging on a heated surface are presented. High-speed images were used to measure the parameters required for the simulations. The measured contact angle was 110 ± 10° and the droplet initial diameter was 3 ± 0.02 mm for the three cases studied (see table 1).

A side by side comparison of the images obtained for 2.27 m/s impact velocity between the experiments and the simulations is shown in Fig. 2, from the moment the water droplet first touches the surfaces until it reaches its maximum diameter. Despite the instabilities seen in the advancing ring at t = 3.4 ms, the 2D axisymmetric assumption was very accurate.

Table 1: List of parameters studied

<table>
<thead>
<tr>
<th>Case</th>
<th>$v_o$ (m/s)</th>
<th>$d$ (mm)</th>
<th>$\theta_w$</th>
<th>$Re$</th>
<th>$We$</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>1.7</td>
<td>3.0</td>
<td>110°</td>
<td>5332</td>
<td>119</td>
</tr>
<tr>
<td>2</td>
<td>2.0</td>
<td>3.0</td>
<td>110°</td>
<td>6273</td>
<td>165</td>
</tr>
<tr>
<td>3</td>
<td>2.27</td>
<td>3.0</td>
<td>110°</td>
<td>7119</td>
<td>213</td>
</tr>
</tbody>
</table>

The variation in time of the spreading factor is shown in Fig. 3. As expected, an increase in the droplet impact velocity leads to an increase in the maximum droplet spreading diameter, since inertial forces become more important than surface tension. This behavior can be better appreciated in Fig. 4, where the evolution of the maximum spreading factor as a function of $We$ is shown. Less than 0.5% percent difference between the experiments and the simulations was achieved.
Figure 4: Comparison between experimental and computational results obtained for the maximum spreading factor at different We.

Figure 5 shows the pressure field inside the droplet during the spreading process. The early stage of the impact is found to have the highest pressure level. As a result of this high pressure, the water droplet starts to deform, which makes the droplet-surface contact area larger, until the droplet reaches its maximum diameter, where the pressure is no longer significant. Due to the pressure uniformity, the surface tension begins to dominate, and the receding process begins.

Heat transfer from the heated wall, initially at 60°C, to the water droplet, initially at 22°C, was measured computationally. Figure 6 shows the results of the numerical simulation for 2.27 m/s impact velocity. It is seen in Fig. 6a that the region with the highest temperature over the droplet surface corresponds to the region where the air recirculates as a consequence of the droplet motion (see Fig. 6b). Nevertheless, this behavior is purely numerical, since the air was assumed to have zero velocity at the beginning of the simulation, which neglects the fact that the air has already been perturbed by the droplet when it was falling.

The maximum temperature within the droplet is achieved inside the advancing ring at t = 3.4 ms, at this point the droplet has reached its maximum diameter and starts to recede.

Figure 6: a) Temperature distribution and b) streamlines for Re = 7119 and We = 213.

Figure 7 shows the vector field within the droplet at four different times. It is observed that both the normal and the tangential component of the velocity have their maximum magnitude at the early stage of the impact. At the stagnation line, where the maximum pressure level is located, the normal vector has its highest magnitude. Similarly, the maximum magnitude of the tangential vector is located inside the advancing ring. However, the droplet motion is mainly driven by the normal component of the velocity. In contrast, as the droplet spreads, the tangential component becomes more significant. Thus, it is reasonable to assume that the highest heat transfer is achieved at the center of the droplet, where the normal component of the velocity is responsible for the increase in convective heat transfer. On the other hand, as the droplet spreads further, the tangential component of the velocity dominates and diffusion takes place, therefore heat transfer becomes poor closer to the edge.
Figure 8 shows the temperature distribution at four different locations for 2.27 m/s impact velocity. Confirming the previous discussion, it is seen that there is a rapid change in temperature after the droplet impacts the surface, reaching a minimum within 3.4 ms, and this response becomes more significant closer to the stagnation point. It can be observed that as the distance from the droplet center increases, the time at which the temperature drop takes place increases as well. In fact, the change in temperature at any point over the heated surface starts at the moment when the edge of the droplet reaches that specific point, and since the maximum droplet diameter achieved in the simulations was less than 6.0 mm, the temperature at that point remained constant during the spreading process. Different droplet impact velocities were tested in order to examine the influence of Re and We on the heat transfer. The numerical results for the temperature distribution at the wall center are shown in Fig. 9. For the velocity range investigated in this paper, the temperature decay seems to be independent of impact velocity, and the increase in maximum droplet spreading diameter appears to be the only important consequence of raising the impact velocity.

Figure 9: Temperature variation at the wall center.

Hence, in order to quantify the effect of droplet velocity in heat transfer, it is more suitable to express Nusselt number as function of the instantaneous wetted area

$$ Nu = \frac{h \cdot D(t)}{k} $$

where $h$ was defined as the ratio of the heat flux at the wall ($q''$) to the temperature difference ($T_{wall} - T_{∞}$). Some researchers have used the wetted area to determine the cooling effectiveness. In [20] the ratio of the total energy absorbed by the droplet to the total energy that it can absorb was measured. In [10] the cooling effectiveness was defined based on a comparison between the heat transfer achieved with and without the presence of the droplet. Figure 10 shows the transient Nu variation for the three cases studied. The heat transfer is enhanced primarily due to the increase in the wetted area. The capacity for transferring heat...
during the spreading process depends mostly on the fluid properties rather than the intensity of the impact velocity.

![Graph](image)

Figure 10: Transient Nusselt number variation at the wall center.

**CONCLUSION**

The effect of droplet impingement dynamics on the heat transfer from a heated surface was numerically investigated. High-speed photography was used to capture the droplet dynamics and validate the simulations. The VOF model with an adaptive local grid refinement technique was employed, and a conjugate solution was found for a constant heat flux.

The assumption that the water droplet spreads out with a constant advancing contact angle, was found to be very accurate, and less than 0.5% percent difference between the experiments and the simulations was attained. It was found that the change in the wall center temperature depends mainly on the fluid properties rather than droplet impact velocity. Nonetheless, water droplet impact velocity has an important effect on the augmentation of the wetted area, which increases the heat transfer. Moreover, it was found that the radial temperature was strongly influenced by the droplet dynamics. The highest heat transfer was achieved at the wall center, where the normal component of the velocity is responsible of an increase in the convective heat transfer. However, as the droplet spreads further, this velocity component decreases as well as the heat transfer.

**REFERENCES**


