Simulation of Single Row of Droplets Impact on a Flat Surface Under High-overload Condition

Qi Guo, Liping Pang and Haijun Chang

ABSTRACT

A tow-dimensional model has been developed of a row of droplets impact on a flat surface heated with constant power. The simulation based on VOF (volume of fluid) method is carried out to simulate the liquid film flow with no-phase-change heat transfer during droplets hitting the flat surface. A series of case with different droplets velocities and overload acceleration are calculated. The results obtained under high-overload and normal gravity conditions are compared to show the effect of acceleration on the heat transfer performance.

KEYWORD: high-overload environment; overload acceleration; VOF method; no-phase-change heat transfer

INTRODUCTION

Advanced thermal management technology is becoming a major factor restricting the development of electronics components nowadays. Spray cooling has received much attention in recent years because it can remove high heat fluxes.

The major heat transfer mechanism of spray cooling are heat conduction when droplets hitting the surface, force convection between liquid film and the heater surface, evaporation on the liquid film surface, nucleate boiling, secondary nucleate boiling and so on. These all contribute to the high heat removal rate of spray cooling. It is reported that its Critical Heat Flux (CHF) can be up to 1000 W/cm² when water is the coolant [1].

Experiment is the major method to carry out the research [2]. However, due to the difference in experimental conditions, many conclusions could not reach an agreement. Spray cooling involves many processes such as droplets ejection, droplets
interaction and impact into liquid film, so it is very hard to model the whole process. Many researchers attempted to set up some models to describe this process. Some researchers established VOF method to simulate the progress of a single droplet impacting on a constant temperature flat surface [3]. Some researchers established a 3D multiphase flow model to simulate the heat transfer in spray cooling [4].

In addition, other advantages including compact structure, reduced fluid inventory, low heater surface temperature and temperature gradient, make spray cooling one of the most promising heat transfer techniques used for the cooling of high-power electronics in aviation and aerospace field. However, aircraft suffers a high overload in maneuvering flight. Airborne thermal control equipment will be affected by the overload inertial forces from different directions and the heat transfer performance is likely to be changed badly [5].

In order to understand the influence of overload in maneuvering flight on the heat transfer performance of spray cooling, a two-dimensional model will be developed to study the impact of a row of droplets on a flat surface heated with constant power. In this paper, a model based on VOF method is set up to simulate the liquid film flow as well as no-phase-change heat transfer during droplets hitting on the wall surface. Numerous cases with different droplet velocities and overload acceleration are studied. The results obtained under high-overload and normal gravity conditions are compared to show the effect of acceleration. This work tries to provide a reference for the application of spray cooling under high-overload environment.

**COMPUTATIONAL MODEL**

The objective of this simulation is to understand the heat transfer performance of spray cooling in single phase region under high-overload environment. This simulation can analyze the effect of acceleration on the liquid film flow and the heat transfer performance.

Due to the complexity of the studied problem, the following assumptions will be made: (1) In single-phase region, heat convection between liquid film and heater surface is the major heat transfer mechanism. (2) It is assumed that the heater surface is stationary and the whole solution domain is in horizontal or vertical inertia force field. (3) For a small heater surface unit, there is a single row of droplets impact on it vertically and continuously.

The width of the solution domain is 25mm and the height is 20mm in the computational model. The ordinate origin is on the bottom of the left. In order to simulate the atmospheric environment, the two vertical sides are assigned the pressure outlet boundary and the top side is assigned a pressure inlet boundary. The bottom side is assigned a no-slip boundary condition. The heater is marked with red line and it has a width of 3 mm and heat power of 15 W/cm².

| Fluid | Density (kg/m³) | Specific heat (J/kg·K) | Thermal Conductivity (W/m·K) | Viscosity (kg/m·s) |
|-------|----------------|------------------------|----------------------------|-------------------|
| Air   | 1.225          | 1006.43                | 0.0242                     | 0.0000178 94      |
| Water | 998.2          | 4182                   | 0.6                        | 0.001003         |

Table 1. Physical parameters of fluid.
Air is the primary phase in the solution domain and liquid is the secondary phase. Pressure solver based implicit method applicable to incompressible flow is chosen for solver model and unsteady laminar model is used for viscous model. The governing equations in this model include momentum equation, energy equation and volume fraction equation. Pressure Implicit with Splitting of Operator (PISO) is used for pressure-velocity coupling and PRESTO! discretization scheme is used in pressure calculation. Momentum and energy equations are solved using a first-order upwind scheme and volume fraction equation is solved using Geo-Reconstruct discretization scheme. First-order implicit method is used for time discretization. In addition, the physical parameters of the gas and liquid phases are all constant. The surface tension coefficient of the liquid water is 0.073 N/m and the static state contact angle of water and wall surface is 90°. The initial temperature of air and droplets is 27 °C and other parameters are shown in Table 1.

DESIGN OF OPERATING CONDITIONS

Normal gravity environment is represented by “0g”, and “10g” and “15g” are used to represent the overload acceleration magnitude suffered by current and future high maneuver aircraft respectively.

Chen et al. [6] researched that the droplet velocity had the most dominant effect on CHF and the heat transfer coefficient. Given this conclusion, droplet velocity is the only parameter studied in this research. The droplets are spaced at 1.6 mm centers and have a diameter of 1 mm. The droplet velocities tested are 2.5m/s, 4m/s and 7m/s, respectively.

In the simulation, droplets with same initial parameters impact the heater surface continuously and the calculation time is 19.7 ms. We can judge whether the simulation reaches steady state or not according to the changes of liquid film flow and heater surface temperature. In our study, the surface temperature in steady state is obtained by the average heater surface temperature in 17.2~19.7ms. The simulation conditions are shown in Table 2.

| Case | Acceleration ( m/s² ) | Droplet Velocity ( m/s ) | Acceleration Direction |
|------|------------------------|--------------------------|-----------------------|
| 1    | 0g                     | 2.5                      | ←                     |
| 2    | 0g                     | 4                        | ←                     |
| 3    | 0g                     | 7                        | ←                     |
| 4    | 10g                    | 2.5                      | ←                     |
| 5    | 10g                    | 4                        | ←                     |
| 6    | 10g                    | 7                        | ←                     |
| 7    | 15g                    | 2.5                      | ←                     |
| 8    | 15g                    | 4                        | ←↑↓                   |
| 9    | 15g                    | 7                        | ←↑↓                   |
RESULTS AND DISCUSSION

Relationship of Surface Temperature and Droplet Velocity

![Figure 1. Heater surface temperature vs. droplet velocity under horizontal high-overload conditions.](image1)

![Figure 2. Heater surface temperature vs. droplet velocity under overweight and weightlessness condition.](image2)

The simulations are established under normal gravity and high-overload conditions, respectively. Average surface temperature is used as the criterion of heat transfer performance. From Fig. 1 and Fig. 2, it can be seen that the relationship between average surface temperature and droplet velocity is different when the value and direction of overload acceleration are different. The details are discussed below.

**Heat transfer Performance Under Normal Gravity Condition**

Fig. 1 shows that average heater surface temperature decreases and heat transfer performance improves with the increase of the droplet velocity under normal gravity condition. The decrease of curve slope indicates that average surface temperature becomes steady with the further increase of droplet velocity. The heater is heated with 15 W/cm² and the average surface temperatures are 52.84°C, 51.26°C and 50.96°C at respective droplet velocities of 2.5 m/s, 4 m/s and 7 m/s. When a droplet impact on the flat surface, it starts spreading into thin film against viscous force and surface tension. After the impact, the inertia force of droplets provides the momentum and the kinetic energy is dissipated during the spreading. Fig.3 shows that when the droplet velocity is 2.5 m/s and the initial momentum of droplet is low, the liquid film is easy to break and its distribution on the heater surface isn’t uniform. When the droplets velocity is 4 m/s, the liquid film is not break obviously and its distribution is relatively uniform. When the droplet velocity is increased to 7 m/s, the liquid film surface is serrated without breaking up and its distribution is uniform.

![Figure 3. Liquid film flow behavior at different droplet velocities.](image3)

312
The heat transfer results obtained at different droplet velocities are analyzed, and
the effect of velocity on heat transfer performance can be summarized as follows.

(1) With the increase of droplet velocity, the impact force of droplets on liquid
film is enhanced. As a result, the liquid film flow and convective heat transfer are also
increase.

(2) Droplets would bring air into the liquid film during the impact. Low thermal
conductivity of the air restricts the heat transfer when air bubbles are adhered to the
heater surface. Once the liquid film flow is enhanced, the attachment time of air
bubbles on the heater surface is decreased and then heat transfer is improved.

(3) A larger droplet velocity leads to a larger fluid flux. Increasing droplet
velocity is also expected to make more working fluid to transfer heat with the heater
surface in unit time.

(4) With the increase of droplet velocity, more droplets would impact on the
liquid film in unit time and the air carried into liquid film is increased as a result.

(5) The thickness of liquid film doesn’t change monotonously with the increase
of droplet velocity. The thicker the film is, the harder the adhering bubbles get away
from the heater surface and also the harder the cold working fluid contact with the
heater surface and transfer heat.

From the above, the average heater surface temperature is decreased and the heat
transfer is enhanced with the droplet velocity increased in the simulation of normal
gravity. With further increase of the droplet velocity, the surface temperature becomes
constant.

Heat Transfer Performance Under Horizontal High-overload Condition

It can be found in Fig.1 that heater surface temperature is lower and heat transfer
performance is better in horizontal high-overload environment than those obtained in
normal gravity environment. It is also indicated that surface temperature first
decreases and then increases with the increase of droplet velocity and the lowest
surface temperature is acquired at the droplet velocity of 4 m/s. When the droplet
velocity is 4 m/s, surface temperature obtained under 15g condition is lower than that
obtained under 10g condition. At other droplet velocities, 10g condition can get lower
surface temperature. Overload acceleration change the liquid film flow behavior on
the heater surface including liquid film velocity and thickness distribution. Those will
affect the heat transfer performance.

Fig.4 shows that at the droplet velocity of 2.5 m/s, the distribution of liquid film
on flat surface is very non uniform, and higher overload acceleration leads to more
non uniform distribution. Moreover, the liquid film on the left side breaks badly.
While that on the right side piles up and the film thickness is increased obviously.
When the droplet velocity is 4 m/s, the distribution of liquid film is relatively uniform
on the center of the heater surface and the liquid film still breaks or accumulates on the
edge of the surface. When the droplet velocity is 7 m/s, the distribution of liquid film
is relatively uniform without breaking or accumulating. Thus it can be seen that
overload acceleration changes the flow and distribution of liquid film on the flat
surface and the uniformity of liquid film distribution becomes worse with droplet
velocity decreasing.
Heat Transfer Performance Under Vertical High-overload Conditions

Under horizontal high-overload condition, the average heater surface temperature is the lowest at the droplet velocity of 4 m/s. However, when it is under vertical high-overload condition, heat transfer performance will be quite different because of acceleration direction.

Fig. 2 shows that when the droplet velocity is 4 m/s, the average heater surface temperatures are 51.26 °C, 45.98 °C and 54.10 °C under normal gravity, overweight and weightlessness conditions, respectively. The temperature decreases in overweight environment and increases in weightlessness environment. It is also obtained from Fig. 4 that when the droplet velocity is 7 m/s, the acceleration almost shows no effect on heat transfer.

Fig. 5 shows that when the droplet velocity is 4 m/s, surface temperature decreases, but liquid film breaks obviously under overweight condition. While under weightlessness condition, liquid film continuity is relatively good, but the surface temperature is increased due to the increase of liquid thickness. Fig. 6 shows that when the droplet velocity is 7 m/s, liquid film continuity is relatively good under both overweight and weightlessness conditions. Moreover, the surface temperature is almost constant. Higher droplet velocity means larger initial momentum after droplets impact the heater surface. And in this case, overload acceleration shows little influence on liquid film flow and then on heat transfer performance as well.

Dangers of Liquid Film Breaking and Accumulating

Computational model is modified slightly in this section to study the danger of liquid film distribution non-uniformity under high-overload. The length of the heater surface is extended to 23 mm in the new model. Acceleration is 15g to the left in horizontal direction and droplets with velocity of 2.5 m/s are spaced at 1.2 mm centers.

In this case we can find that that average surface temperature has a cyclical fluctuation over time. One cycle is 2 ms and the range is 60 ~ 80 °C. The heater surface temperature at the point of (2, 0) has a rapid increase. This point is in the area of liquid film breaking and when there is no liquid film on the flat surface. Heater surface is in direct contact with the air so that heat transfer coefficient is very small at this time. As a result, the local temperature increases sharply in a short time. When the surface temperature is more than 100 °C, the phase change processes are likely to play
an important role in the total heat transfer and the computational model is not applicable anymore. Even so, the simulation still shows the danger of film breaking on heat transfer. On the other hand, the temperature period is also 2 ms and it indicates that film breaking has a direct effect on average heater surface temperature. The heater surface temperature and liquid film thickness at the point of (23, 0) increases continuously with the growth of the liquid film accumulating at the point.

The results above illustrate that film breaking and accumulating will lead to high local temperature, which will become serious threat to the component working performance.

ACKNOWLEDGMENT

The research is sponsored by the Aviation Science Foundation of China (2014ZC09002).

REFERENCES

1. M.H. Sadafi, S. González Ruiz, M.R. Vetrano, I. Jahn, J. van Beeck, J.M. Buchlin, K. Hooman. An investigation on spray cooling using saline water with experimental verification. Energy Conversion and Management, Volume 108, 15 January 2016, Pages 336-347.

2. Cheng W. L., Xie B., Han F.Y., et al. An experimental investigation of heat transfer enhancement by addition of high-alcohol surfactant (HAS) and dissolving salt additive (DSA) in spray cooling [J]. Experimental Thermal and Fluid Science, 2013, 45: 198-202.

3. Sikalo S., Wihelm H.D., Roisman I.V., et al. Dynamic contact angle of spreading droplets: Experiments and simulations [J]. Physics of Fluids, 2005, 17(6): 62103-62103

4. Sarkar S., Selvam R. P. Direct numerical simulation of heat transfer in spray cooling through 3D multiphase flow modeling using parallel computing [J]. Journal of Heat Transfer, 2009, 131(12): 121007-1-121007-8

5. Ray S.S., Yarin A.L., Weickgenannt C.M., et al. Drop impact cooling enhancement on nano-textured surfaces. Part II: Results of parabolic flight experiments [zero gravity (0g) and supergravity (1.8g)] [J]. International Journal of Heat and Mass Transfer, 2014, 70: 1107-1114

6. R.H. Chen, L.C. Chow, J.E. Navedo, Effects of spray characteristics on critical heat flux in subcooled water spray cooling, International Journal of Heat and Mass Transfer 45 (19) (2002) 4033-4043.