Numerical Simulation of the Heat Transfer Characteristics of Spray Cooling With Different Spray Conditions

Liang Zhao*
Southwest China Institute of Electronic Technology, Chengdu 610036, China

*Corresponding author e-mail: zhaoliang_0302@163.com

Abstract. As an efficient cooling method, spray cooling has been studied by many former. While the heat transfer characteristics is quite complicated, many influence factors are not clear. In the present paper, three main influence factors, the heat temperature, the spray angle and the nozzle position are numerically studied by founding a two-phase flow model based on the Euler-Lagrangian method. The results shows that when the flow rate of the coolant large enough, the surface temperature and the total heat transfer flux would increase almost linearly proportional to the heat temperature. The spray angle increase could improve the heat transfer efficiency. The larger nozzle position will lead to higher heat transfer efficiency with a certain scope optionally. Due to atomizing construction, the temperature in the central of the heat would be higher, which should be pay more attention in the actual application.

1. Introduction
As the heat flux of electronic devices rising, many traditional thermal management methods, such as natural air cooling, forced air cooling, micro-channel cooling, heat pipe cooling, pool cooling, seem to be ineffective to meet the requirements of electronic devices cooling. In recent years, the spray cooling is being more attractive due to its high thermal load capability. It is reported that the critical heat flux of spray cooling could reach 1000W/cm² [1]. It means that the spray cooling could be used for microwave directed energy weapons [2], high performance micro-chip [3], and photovoltaic panel [4], which always generate large amount of heat when they performed.

In spray cooling, liquid is forced through a nozzle and atomized into small droplets with high speed by the pressure. The liquid droplets impact on the heated surface and remove heat. The process is considerably complex, which always constrains heat conduction, force convection, drop evaporation, nucleate boiling, secondary nucleate boiling, droplet breakup, droplet collision and so on. And it could be affected by the gravity, the distance between the nozzle and the heated surface, the pressure, the properties of liquid, the flow rate and others [5]. Agnieszka Cebo-Rudnicka and Zbigniew Malinowski [6] numerically and experimentally evaluated the heat transfer coefficient and heat flux at the copper plate during water-air assisted spray cooling from 700 °C to the ambient temperature. Based on the Euler-Lagrangian model, Hong Liu et al. [7] numerically studied the heat transfer characteristics, metal cooling characteristics during spray cooling and the heat transfer mechanisms during film cooling, transition boiling, nucleate boiling and forced convection regimes. A comprehensive set of numerical simulations performed to examine the current Computational Fluid Dynamics capabilities in the prediction of the interaction of a water mist spray with a vertical upward jet of hot air within an...
Euler-Lagrangian framework was conducted by Tarek Beji et al. [8]. A two-dimensional numerical model was developed by Pang Liping et al. [9] to investigate a heat and mass transfer process of liquid film formed by a row of droplets under high-overload conditions. Yan Hou et al. [10] used CFD method to study the spray cooling heat transfer characteristics, they found that the heat flux and its distribution on the heated surface were affected by the heated surface temperature, mass flux, nozzle position and the number of nozzles.

The former studies were mainly focused on the heat and mass transfer mechanism of the spray cooling and the critical heat flux, that the heater has a large surface for heat transfer. While in actual application, the heat flux would much smaller than the critical heat flux, and the heater’s surface is small. And the mass flux, the nozzle position, the heated surface and the number of nozzles could be modified easily to ensure the heater in the safe temperature range. In present paper, a numerical model of spray cooling was found to investigate the effect of the spray conditions, which are the different temperature of the heater, different spray half angles and different nozzle positions.

2. Mathematical model

2.1. Computational model

Based on the Euler-Lagrangian method, a two-phase flow model is established to investigate the thermal management performance of the spray cooling. The droplets is simulated as the discrete phase, the gas is treated as the continuous phase. The Eulerian wall film model is adopted. As shown in Figure 1(a), a single spray nozzle which was the pressure swirl atomizer was adopted, the temperature of heater’s down surface was fixed at 380K, 390K, 400k, 410K and 420K, respectively, which was used to mimic the chip that generate heat. The dimension of the heater had the height of 2mm, and the radius of 5mm. And the material of the heater was copper. Water was applied as coolant. The thermal physics properties of the coolant and the air is presented in Table 1. The coolant would be atomized into droplets as shown in Figure 1(b). The thermal physics

\[
\rho \frac{\partial h}{\partial t} + \nabla \cdot (h \mathbf{u}) = \frac{m}{\rho},
\]

2.2. Governing equations and boundary conditions

Table 1. The thermal physics properties of the coolant and the air

| Fluid | Density(kg/m³) | Specific heat(J/kgK) | Thermal conductivity (W/mK) | Viscosity (kg/ms) |
|-------|----------------|----------------------|----------------------------|------------------|
| Air   | 1.225          | 1006.43              | 0.0242                     | 0.000017894      |
| Water | 998.2          | 4182                 | 0.6                        | 0.001003         |
Where $\rho_c$ is the density of the constant phase, $\bar{u}_c$ is the velocity of the constant phase, $h$ is the height of the wall film, $m^s$ is the mass rate which is generated by the particle collection and edge separation.

$$
\frac{\partial h\bar{u}_c}{\partial t} + \nabla \cdot \left[ h\bar{u}_c\bar{u}_c \right] = -\frac{h\nabla \cdot P_c}{\rho_c} + \left( g_r \right) h + \frac{2}{3\rho_f} \frac{3v_f}{h} \bar{u}_c + \frac{\dot{q}}{\rho_c} \tag{2}
$$

$$
\frac{3}{2\rho_f} \frac{\dot{r}_f}{h} - \frac{3v_f}{h} \bar{u}_c + \frac{\dot{q}}{\rho_c} \tag{3}
$$

$$
P_L = P_{gas} + P_h + P_o \tag{4}
$$

$$
P_n = -\rho h (n \cdot g) \tag{5}
$$

$$
P_p = -\sigma \nabla \cdot (\nabla h) \tag{6}
$$

Where $P_{gas}$ is the pressure of the gas, $P_h$ is the pressure generated by gravity, $P_o$ is the pressure generated by surface tension, $g_r$ is the gravity which is perpendicular to the wall film, $\dot{r}_f$ is the shear stress, $v_f$ is the viscosity of the liquid, and $\dot{q}$ is the momentum source term which is induced by the droplet collection and edge separation.

$$
\frac{\partial \left( \rho cT \right)}{\partial t} + \nabla \cdot (\rho c u T) = \nabla \cdot \left( \frac{k}{c_p} \nabla T \right) + S_T \tag{7}
$$

Where $S_T$ is the heat source of the liquid.

$$
\frac{\partial \rho_m}{\partial t} + \frac{\partial \rho_m u_i}{\partial x_i} = \nabla \cdot \left( \rho_m D \frac{\partial Y_m}{\partial x_i} \right) + \dot{\rho}_m \tag{8}
$$

Where $\rho_m$ and $\rho_c$ is the density of m component and mixed gas, respectively. $\dot{\rho}_m$ is the source term which is induced by the evaporation of droplet. $Y_m$ is the mass fraction of the m component $D$ is the diffusion coefficient of the m component.

3. Results and discussion

In this paper, the pressure-based solver with the pressure-velocity coupling algorithms of pressure-implicit with splitting of operators is applied for convergence of the unsteady state. The spatial discretization of pressure, momentum, turbulence and energy equations is the second order upwind scheme. The temporal discretization is the second-order implicit scheme.

3.1. Effect of the surface temperature of the heater

The nozzle position, the upstream pressure, the flow rate of the coolant and the temperature of the coolant were set to be 10mm, 1.6MPa, 0.0015kg/s and 300K, respectively. Figure 2(a) shows the heat flux and the surface temperature change with different heat temperature. It is found that the surface temperature and the heat flux increase almost linearly proportional to the heat temperature. Although the surface temperature is higher than boiling temperature of the coolant, the two curves in Figure 2(a)
coincide in general, which means that the phase change between the film material and the gas species may rarely works. This is attributed to the large flow rate and the lower temperature of the coolant, which could remove the heat from the heater efficiently. It also indicates that as the temperature rises or the flow rate of the coolant decrease, the phase change of the coolant on the heat surface would occur that would further improve the heat transfer efficiency. While in actual application, the boiling phenomenon is difficult to control. The temperature of the heat surface could increase rapidly. Thus, in the certain range when the flow rate of the coolant is large enough, the surface temperature and the total heat flux would increase linearly proportional to the heat temperature.

Figure 2(b) presents the film thickness variations versus the surface radius, which reflect that the five curves almost coincide in general. The main reason is that the atomization parameters for this five case are completely same. To further explain it, the velocity distributions for the case with heat temperature of 390K and the case with heat temperature of 420K are shown in Figure 3, which indicates that the velocity distributions are almost the same. It also manifests that the droplets may have the same velocity and distribution. The collision between the droplets and the heat surface may be similar for this four cases. And the phase change phenomenon has seldom effect. Then the wall film formed on the heater surface is almost alike. And the curves also show that the film thickness rises along the radius of the heater. Due to the atomization form, the wall film thickness in the center of the heater is very small, which has little effect on the temperature of the heater, because the radius of the center part is less than 1mm.

![Figure 2](image_url)

**Figure 2.** (a) The surface temperature and the total heat flux vs the heat temperature; (b) the wall film thickness distribution.

![Figure 3](image_url)

**Figure 3.** The velocity distribution for different heat temperature.
3.2. Effect of the spray angle of the atomization

The spray angle of the atomization is an important parameter which could seriously affect the atomization form and the heat transfer of spray cooling that has been experimentally presented by W.L. Cheng et al. [11]. In order to further characterize the effect of the spray angle of the atomization, it was set as 36°, 42°, 48° and 54°, respectively, the other parameters keep same as the former simulation. The Figure 6(a) shows the surface temperature and the total heat flux of heater change with the spray angle of the atomization increasing. It manifests that as the spray angle rises, the surface temperature of the heater decreases, and the total heat flux rises, which means that the heat transfer efficiency will rise with the spray angle increasing. When the spray angle of the atomization is 54°, the total heat flux would be about 5.3e9 W/m², which is approximately 1.26 times than that the spray angle of atomization is 36°. While the temperature only changes from 379.28 K to 381.5 K.

To further investigate the changes, the temperature distribution is presented in Figure 4. By comparing the temperature distributions, the differences are not very obvious as well. It can be seen that the spray cooling forms a ring with lower temperature in the central part of the heat surface. And as the spray angle increase, the radius of the ring becomes larger, the domain with lower temperature becomes smaller. While the central part of the heater’s surface has higher temperature. It is attributed to that the spray with larger spray angle would form thinner wall film, the radial velocity may be much larger, and then the heat transfer efficiency would be increase as shown in Figure 5. In the Figure 5, it indicates that the difference of the radial velocity distribution in the four cases are similar that the radial velocity in the central is small. By comparing the four cases, it could be seen that the radial velocity in the case with the spray angle of 54° is more uniform, the radial velocity in the case with spray angle of 36° has the most uneven radial velocity distribution. For the case with the spray angle of 54°, besides the central part, the radial velocity in the other part is a little larger. Thus the heat transfer efficiency could be bigger than the other three.

![Figure 4](image4.png)

**Figure 4.** The temperature distribution for different spray angles.

![Figure 5](image5.png)

**Figure 5.** The radial velocity distribution for different spray angles.
The wall film thickness along the radius of the heater is presented in Figure 6(b). It is significant that the wall film thickness would be thinner as the spray angle rises. The flow rate of the coolant is same for this four cases. It means that when the spray angle increases, the coolant on the heat surface would be less. It is attributed to that when the spray angle rises, the radial velocity of coolant of the heater surface is much larger as explained in the former and depicted in Figure 5. The heat transfer efficiency and the radial velocity have the positive correlation, thus the heat transfer efficiency would be much larger and the surface temperature would be lower. Therefore, it can be concluded that the spray angle of the atomization is an important factor influencing the spray velocity distribution and the heat transfer efficiency.

Figure 6. (a) The surface temperature and the total heat flux vs the heat temperature; (b) the wall film thickness distribution.

3.3. Effect of the nozzle position
In order to ascertain the effect of the nozzle position on spray, the surface temperature of the heater, the total heat flux and the wall film thickness were all simulated. The upstream pressure, the flow rate of the coolant and the temperature of the coolant were set to be 1.6MPa, 0.0015kg/s and 300K, respectively. The distance was 6, 8, 10 and 12mm, respectively. The computation results are shown in Figure 7.

It can be seen from Figure 7(a) that the surface temperature rises with the nozzle position increasing in the range from 6mm to 12mm. While the trends of the curves show that the temperature would be change little when the nozzle position continues to increase. By comparing the four cases, the total heat flux for the nozzle position of 12mm is 1.28 times of the total heat flux for the nozzle position of 6mm. The conclusion could be made that the larger nozzle position would improve the heat transfer efficiency with a certain scope optionally.

To further explain the phenomenon, the wall film thickness along the radius of the heater is shown in Figure 7(b). It can be seen that the wall film thickness for the case with the nozzle position of 12 mm is smaller than the other three. In the central part of the heater, the wall film thickness is much smaller especially for the case with the nozzle position of 12mm, whose radius is about 1mm. Which means that the heat transfer efficiency for the case with the nozzle position of 12 mm is much larger, it may be cause by the fact that the radial velocity of the wall film of coolant is much larger.
Figure 7. (a) The surface temperature and the total heat flux vs the heat temperature; (b) the wall film thickness distribution.

To further explain the heat transfer efficiency of the four cases, the Figure 8 depicts the temperature distribution of the heater’s surface. It is clear that as the nozzle position increases, the temperature of the surface seems to be more uniform, but the temperature in the central part of the heater rises especially for the case with the nozzle position of 12mm. This is attributed to the reason that the droplet of nozzle would form a circle ring, which is not uniformly distributed, that the droplets in the circle ring is very thin. Accordingly, the central of the atomization has fewer droplets, the wall film in the central part of the heater is thinner. Due to the high thermal conductivity of the heater, the temperature seems to be still uniform, which means that in the actual application, the nonuniform distribution of droplets should be pay more attention.

Figure 8. The temperature distribution for different nozzle positions.

The radial velocity for the four cases is displayed in the Figure 9. In the central part of the heat surface, the velocity is small. As the nozzle position increase, there is no evident phenomenon that the radial velocity becomes uniformly or increases. On the contrary, the domain with lower radial velocity becomes lager as the nozzle position increases. Thus, the improvement of heat transfer efficiency by increasing the nozzle position could be realize in a contain range.

Figure 9. The radial velocity distribution for different nozzle positions.
4. Conclusion

Spray cooling is a promising cooling method which would be used in electronics equipment with high heat flux. In this paper, a numerical investigation of spray cooling is conducted. The heat transfer characteristics of spray cooling with different spray conditions are analyzed. The conclusion could be made as followings:

1. Due to the large flow rate and low temperature of the coolant, the surface temperature and the heat flux increase almost linearly proportional to the heat temperature, which is attributed to that the four cases may have the same velocity distribution and wall film distribution;
2. As the spray angle rises, the surface temperature of the heater decreases, and the total heat flux rises, which means that the heat transfer efficiency will rise with the spray angle increasing;
3. The larger nozzle position would improve the heat transfer efficiency with a certain scope optionally.

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 Convers. Manag. 108 (2016) 336-347.
[2] G. Liang, I. Mudawar, Review of spray cooling – Part 1: Single-phase and nucleate boiling regimes, and critical heat flux, Int. J. Heat Mass Transf. 115 (2017) 1174-1205.
[3] M. Visaria, I. Mudawar, Theoretical and experimental study of the effects of spray inclination on two-phase spray cooling and critical heat flux, Int. J. Heat Mass Transf. 51 (2008) 2398-2410.
[4] S. Nižetić, D. Ćoko, A. Yadav, F. Grubišić-Ćabo, Water spray cooling technique applied on a photovoltaic panel: The performance response, Energy Convers. Manag. 108 (2016) 287-296.
[5] W.M. Grissom, F.A. Wierum, Liquid spray cooling of a heated surface, Int. J. Heat Mass Transf. 24 (1981) 261-271.
[6] A. Cebo-Rudnicka, Z. Malinowski, Identification of heat flux and heat transfer coefficient during water spray cooling of horizontal copper plate, Int. J. Therm. Sci. 145 (2019) 106038.
[7] H. Liu, H. Chen, C. Cai, M. Jia, H. Yin, Numerical investigation on the unsteady spray cooling of high temperature steel plate, Int. J. Heat Mass Transf. 143 (2019) 118342.
[8] T. Beji, S. Ebrahimzadeh, G. Maragkos, B. Merci, Numerical modelling of the interaction between water sprays and hot air jets - Part II : Two-phase flow simulations, Fire Saf. J. 96 (2018) 143–152.
[9] P. Liping, L. Kun, G. Qi, L. Shuxin, Y. Chao, Numerical study of high-overload effect on liquid film of spray cooling, Appl. Therm. Eng. 127 (2017) 1015-1024.
[10] Y. Hou, Y. Tao, X. Huai, International Journal of Thermal Sciences Numerical simulation of multi-nozzle spray cooling heat transfer, Int. J. Therm. Sci. 125 (2018) 81–88.
[11] W.L. Cheng, Q.N. Liu, R. Zhao, H.L. Fan, Experimental investigation of parameters effect on heat transfer of spray cooling, Heat Mass Transf. Und Stoffuebertragung. 46 (2010) 911–921.