Numerical simulation on multiphase spray cooling

Peiliang Yan, Hong Liu*, Chang Cai, Jiuliang Gao, Hongchao Yin

Key Laboratory of Ocean Energy Utilization and Energy Conservation of Ministry of Education, School of Energy and Power Engineering, Dalian University of Technology, Dalian, 116024, P.R. China
yanpeiliang@mail.dlut.edu.cn

Abstract. The purpose of this work is using distilled water as working fluid to study the spray cooling heat transfer characteristics from non-boiling zone to boiling zone by CFD method. Simulation is performed using a Euler-Lagrangian method based on the air and liquid droplet two phase flow dynamics. The results of this simulation are in accordance with the experimental results of the laboratory. The simulation results show that the spray height is an important factor influencing the cooling characteristics. With the decrease of spray height, the heat transfer effect is enhanced.

1. Introduction
Since the invention of the computer, with the development of the electronic device packaging technology, the number of electronic devices that a unit area can accommodate increases as well, which has been developed to integrate millions or even hundreds of millions of semiconductor components in a large scale integration circuit. However, the rise in the power of electronic devices and the increase in the calorific value per unit area must be taken into account as well. Each electronic component has its rated operating temperature range, if the electronic device operating temperature exceeds its operating temperature range, the component would generally have some failure. Therefore, the heat problem of the electronic components has become a very important technical issue.

The parameters of the spray cooling phenomenon for study of it are significant [1]. After a numerical simulation on spray cooling have been done by Charles [2] and others, many researchers have also done a lot of numerical work on this phenomenon. It has been found by Issa [3] that the characteristic parameters have a great impact on the final result. A numerical simulation of the spray cooling of the multi-nozzle via Hou [4] was carried out and it was found that the nozzle-to-surface distance was a very sensitive parameter for the droplet velocity distribution. The studies by Alkhedhair [5] showed that velocity has a significant effect on the spray effect. However, most simulation researches only describe the flow behaviour and rarely discuss the its impact on heat transfer.

Based on the Ansys Fluent 18.0 commercial software [6] and the fundamentals of air and liquid droplet two phase flow, the Euler-Lagrangian method was applied to simulate the spray cooling process including the single-phase zone and the boiling two-phase zone.

2. Experiment measurements
The spray test was carried out in a closed spray platform. The platform mainly consists of four parts: liquid supply system, spray system, heating system and data acquisition system. The system diagram shown in Figure 1. The nozzle is a solid cone nozzle, the spray height of it is adjustable. A heated surface with the diameter of 20 mm is horizontally oriented and centered directly below the nozzle. There
were six heating rods to heat the copper pillar, while three thermocouples were used to measure the temperature of heater. In the side and the bottom surface of the copper pillar, insulation material were wrapped to reduce heat loss as well as to improve the experimental accuracy. In the experiment, the distilled water was atomized from the nozzle and sprayed onto the upper surface of the copper pillar, which then flowed into the collector and finally into the incubator via the pump. During this period, the data logger records the thermocouple to measure the temperature profile.

3. Numerical model
A Euler-Lagrangian method is performed to describe the spray cooling process in this paper. The continuous phase model is used for describing the gas flow, and the atomization liquid droplets are tracked using the discrete phase model. The k-ε turbulence model is applied to describe the turbulence characteristics of the spray flow field and the liquid wall film. The Lagrangian liquid film model is implemented to describe the interaction between the particle and the heater surface and the motion of the liquid film on the wall.

3.1. Continuous phase model
In the spray model, the continuous phase (distilled water) is considered incompressible, and all the continuum fluid flows are described by the equations including mass conservation and momentum conservation. The mass conservation equation:

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \mathbf{u}) = 0$$

(1)

The momentum conservation equations are as follows:

$$\frac{\partial (\rho \mathbf{u})}{\partial t} + \nabla \cdot (\rho \mathbf{u} \mathbf{u}) = \nabla \cdot (\eta \nabla \mathbf{u}) - \frac{\partial p}{\partial x} + S_x$$

$$\frac{\partial (\rho \mathbf{u})}{\partial t} + \nabla \cdot (\rho \mathbf{u} \mathbf{u}) = \nabla \cdot (\eta \nabla \mathbf{u}) - \frac{\partial p}{\partial y} + S_y$$

$$\frac{\partial (\rho \mathbf{u})}{\partial t} + \nabla \cdot (\rho \mathbf{u} \mathbf{u}) = \nabla \cdot (\eta \nabla \mathbf{u}) - \frac{\partial p}{\partial z} + S_z$$

(2)

Among the equations, S represents the source term. In the finite volume method, the source term is an unsteady term, which could not been converted from the Gaussian theorem like the integral of the convective term and the diffusion term, and could only be solved by the volume integral.

3.2. Discrete phase model
In a multiphase flow system consisting of discrete phases (droplets) and fluid phases (liquids or gases), the discrete phase is tracked. The motion trajectory curve is calculated by integrating a large number of droplets or particle motion equations. In the spray cooling process, the water is dispersed into a large number of small droplets, which are treated as discrete phases and solved by the Lagrangian method. Trajectories are traced using a Stochastic Tracking model. The trajectories of the particles are affected by the external forces such as Brownian force and heat swirl. The equations are:

$$\frac{d\mathbf{u}_p}{dt} = F_v (\bar{u} - \mathbf{u}_p) + \frac{\mathbf{g} (\rho_e - \rho)}{\rho_p} + \mathbf{F}$$

(3)

$$F_v = \frac{18 \mu \cdot C_f \cdot Re}{\rho_p d_p^2}$$

(4)

$$Re = \frac{\rho d_p |\bar{u} - \mathbf{u}|}{\mu}$$

(5)

Where $F_v (\bar{u} - \mathbf{u}_p)$ is the traction of the mass of the unit particles, and $F_v$ is the relaxation time of the droplets or particles.
The droplet diameter distribution is R-R (Rosin-Rammler) distribution[7]. The R-R distribution is a relatively simple and representative distribution of droplet size. The relationship between the droplet diameter \( d \) and the mass fraction \( Y_d \) of the droplets larger than this diameter is:

\[
Y_d = e^{-\left(\frac{d}{d_f}\right)^n}
\]  

(6)

3.3. Turbulence model

The \( k-\epsilon \) two-equation model is commonly used in simulating spray cooling, and the realizable \( k-\epsilon \) model makes up for the shortcomings of the standard \( k-\epsilon \) model for axisymmetric jet prediction.

In view of the existence of viscous forces in the near wall area, it cannot be simply assumed that it is the complete flow of turbulence. Enhance wall treatment can compensate for this defect effectively, so the enhance wall treatment function is utilized.

![Figure 1. Spray cooling system structure diagram](image)

1- Reservoir 2,8,9,15,18- thermometer 3- Centrifugal pump 4- buffer 5- filter 6- Flowmeter 7- Pressure gauge 10- nozzle 11- Condensation heat exchange brass 12- Heating copper blocks 13- Low temperature thermostat bath 14- Cold light source 16- Acquisition card 17- computer 19- DC power supply 20-24V power supply 21- Liquid collection box 22- Heat exchanger 23- Diaphragm pump

**Figure 1.** Spray cooling system structure diagram

![Figure 2. Geometry model](image)

**Figure 2.** Geometry model

3.4. Boundary conditions

The study case is a cylinder whose upper and lateral sides are defined as pressure outlets and the bottom is set to the wall. Except for the cooling wall, the other wall are adiabatic walls. The solid cone nozzle is placed above the cooled wall in a certain distance. The cooled surface is set to a fixed heat flow boundary condition, and the discrete droplets would form a Lagrangian wall film when they are in contact with the wall. The Lagrangian wall film model uses the wall temperature and Weber number-based criteria to define a fixed value threshold between impact conditions (stick, rebound, spread, and
4. Results and discussion

4.1. Computational geometry
In order to investigate the spray cooling characteristics of heater surface, the simulation domain is a cylinder based on the test bench, the size is 40mm in diameter and the height is 42mm. The droplets injected from the solid cone nozzle were sprayed onto the upper surface of the cylinder with a diameter of 20 mm and a height of 2 mm. Figure 2 shows the 3D calculation domain. A grid of simulation domain is 600,000, and the grid near the surface of the heater surface is locally encrypted.

Grid-independent verification is required before performing 3D simulations based on the data from the experiment. The temperature distributions on the heater surface were examined using three different grids, i.e. 300,000, 600,000, 800,000. The simulation results show that the influence of different grid numbers on the temperature distribution is very low, and the calculated error is about 1%, that is, the temperature distribution varies little with the number of grids. Considering the accuracy of calculation and the workload of the study, the number of grid computing area of 600,000 are used in the latter part of the simulation.

4.2. Comparison of simulation results with experimental data
Using the exact same conditions as the experiment, the results obtained from the verification experiment are compared with the simulated data. The measurement point is the centre point of the cylindrical surface. The calculated results of the temperature near the centre point are compared with the experimental data. The maximum error within 3K, indicating that the mathematical model established in this paper can be a good spray simulation.

![Figure 3. Comparison of simulation results with experimental data](image)

4.3. Temperature and velocity distribution characteristics
Figure 3 shows the temperature distribution in the spray field and the temperature distribution on the wall. It can be seen from the temperature distribution cloud that the temperature field is symmetrically distributed as a whole. It is worth noting that a low temperature region appears at the near wall region and the initial temperature of 297K is reduced to about 290K. The physically reason is, water spray evaporation of droplet is coupled with the movement of water droplets in the air. As the droplets approaches to the heater surface, the evaporation rate is also accelerated. Consequently, the temperature of the droplets and air decreases due to the evaporative heat consumption. When droplets impact on the
wall to form wall film, the heat conducted from the high temperature of the wall gradually to the wall film so that the cooling medium temperature rise again.

![Figure 4](image)

**Figure 4.** Spray temperature distribution cloud and wall temperature distribution

As shown in Figure 4 (b), the heating surface temperature is symmetrically distributed as a whole and has some degree of temperature nonuniformity. Specifically, the temperature in the centre of the heating surface is the lowest, and with the radial distance increases, the wall temperature gradually increased. The wall temperature in the heating surface around the outer edge is highest. The temperature difference between the outer edge and the centre of the maximum is about 20K. The nonuniformity of the surface temperature distribution is determined by the spray characteristics. The largest particle mass flux appears in the centre of the heating surface, which has the strongest impact on the wall, leading to the best heat transfer effect. With the increase of the radial distance, the mass flux of the spray particles decreases, and the heat transfer effect is slightly lower than that of the heating surface. The mass flux of spray particles at the outer surface of the heated surface is minimized, and because of the role of surface tension, where the liquid film aggregation will appear, decreasing the heat exchange, the highest temperature appears in the heating surface of the outer edge.

It can also be seen from Figure 4 that within the circumference other than the outer edge of the heating surface, there is a substantially linear area with a temperature lower than the temperature of the heating surface and higher than the temperature of the outer wall. This indicates that the liquid film that has been heated on the heated surface is splashed, and the temperature of the splashing membrane is higher than the ambient temperature, which is lower than the lowest wall temperature.

Also, there is a circular area near the heating surface whose temperature is lower than the ambient temperature, the reason of which is that it is in the coverage of the solid cone, a liquid film was formed on its surface, which causes the reduction of the wall temperature. And the temperature of the heated wall surface exhibits a tendency to rise along the radius based on the centre of the circle.

4.4. **Impact of spray height**

Changing the spray height can control the droplets sprayed to the wall, which can change the spray efficiency significantly.

Figure 5 shows the relationship between the temperature and the spray height under different heat flux. It can be seen that the wall temperature increases with the increase of the spray height, but the increase of the surface temperature corresponding to the different spray height is different. With the increase of the spray height, the wall temperature rise speed reduces, and the main reason for this phenomenon is with the decrease of the spray height, the velocity of the droplet particle to the wall increases, leading to the acceleration of the wall liquid film movement which would improve heat transfer efficiency.
Table 1. List of key operating parameters

| Parameter | Value                  |
|-----------|------------------------|
| $P_0$     | 3.5 (bar)              |
| $h_0$     | 10, 15, 20, 30, 40 (mm) |
| $u_0$     | 19.98 (m/s)            |
| $T_0$     | 297 (K)                |
| $\alpha_0$| 60 (deg)               |
| $m_0$     | 0.00584 (kg/s)         |
| $t_0$     | 1 (s)                  |
| $d_n$     | 0.61 (mm)              |
| $d_o$     | 0.08 (mm)              |

Figure 5. the effect of spray height on the wall temperature

5. Conclusion
In this paper, the spray characteristics were investigated by the CFD simulations. A two-phase flow mathematical model was presented based on the Eulerian-Lagrangian approach. The effects of spray height were examined. The conclusions are summarized as follows:

1) The model was tested using available experimental data.
2) The wall temperature increases while the spray height increases, as the heat flux increases, the effect of spray height on wall temperature increases first and then decreases.
3) While the spray height increases, the heat transfer coefficient increases and finally almost linearly, as the heat flux increases, the effect of spray height on wall temperature decreases.

References
[1] R H Chen and L C Chow, J E Navedo 2004 Int J Heat Mass Tran vol47 p5095-99.
[2] Westbrook and Charles K 1977 Symp Combust vol16 p1517-26.
[3] R J Issa and S C Yao, A S Mujumdar 2016 Appl Therm Eng vol107 p453-62.
[4] Yan Hou and Yujia Tao, Xiulan Huai, Zhixiong Guo 2012 Appl Therm Eng vol39 p163-70.
[5] Abdullah Alkhedhair and Hal Gurgenci et al.2013 Appl Therm Eng vol61 p416-24.
[6] ANSYS Inc. 2017 ANSYS Fluent 18.0 Theory Guide (USA).
[7] P Rosin and E Rammler 1933 J Inst Fuel vol31 p29-36.
[8] D W Stanton and C J Rutland 1996 SAE Paper 960628.
[9] D W Stanton and C J Rutland 1998 Int J Heat Mass Tran vol41 p3037-54.
[10] P J O’Rourke and A A Amsden 1996 SAE Paper 961961.