Open Access Library Journal
2020, Volume 7, e6789
ISSN Online: 2333-9721
ISSN Print: 2333-9705

Research on Nozzle Spray Cooling Flow Field

Hao Jiang, Chengwen Yang, Baolin Liu

College of Nuclear Technology and Automation Engineering, Chengdu University of Technology, Chengdu, China
Email: mxy_1995@sina.com

How to cite this paper: Jiang, H., Yang, C.W. and Liu, B.L. (2020) Research on Nozzle Spray Cooling Flow Field. *Open Access Library Journal*, **7**: e6789.
https://doi.org/10.4236/oalib.1106789

Received: September 4, 2020
Accepted: September 26, 2020
Published: September 29, 2020

Abstract

In this paper, spray cooling is the research object, and the component transportation model and DPM model are used in the simulation to simulate the spray and its cooling process to the high temperature heat source. By observing the diameter of droplet and velocity distribution at different heights, the effect of spray on the surface of heat source has been explored. Under the same atomization parameters, the droplet velocity and diameter under different spray distances are quite different. The thickness and velocity distribution of the liquid film formed on the surface have an important impact on heat transfer.

Subject Areas

Mechanical Engineering

Keywords

Nozzle, Spray Cooling, Heat Transfer Coefficient, Cooling Experiment

1. Introduction

Convection cold air, water cooling, heat pipes and cold plates are more heat dissipation methods used in electronic components. However, at present, the degree of integration of electronic devices is getting higher and higher, and the heat generated per unit time is also increasing. The heat flux density of some high-end components can even reach 250 W/cm [1]. Such high heat will damage precision components and cause performance instability. Therefore, it is necessary to improve the heat dissipation performance of the components to ensure the normal use of the components. With its high heat exchange efficiency and controllable cooling performance, spray cooling is widely used in industrial cooling at high temperatures [2]. However, the mechanism of its action is very complicated, and it is the product of the close combination of spray and heat transfer disciplines. The effect of spray characteristics on heat transfer performance is very important. When studying spray cooling, many scholars focus on the critical heat flux value of the technology in different environments, hoping to find the best heat transfer performance under different conditions. Mudawar *et al.* [3] conducted experiments to measure the critical heat flux density of fluorocarbon liquids through their own device. The results showed that the critical heat flux density can be reached when the atomization edge and the hot plate boundary reach a tangent. Lin *et al.* established a set of experimental systems using methanol and water as cooling media, and the measured results were that the critical heat flux density was 490 W/cm and 500 W/cm, respectively [4]. So far, most researchers have obtained the critical value under specific conditions through experiments, and have not been able to guide the optimization of the performance of the device through theory.

The characteristics of the spray mainly include the speed, number, diameter, and distribution of the droplets formed by the spray. The characteristics of the spray are not only related to the nozzle, air pressure and liquid flow rate, but also affected by the nozzle distance and spray angle. Regarding the influence of spray characteristics on heat transfer characteristics, there are differences between the results of theoretical models established by different scholars. Chen *et al.* used the spray characteristics as the influence factor, and observed the influence of the speed, diameter and distribution of the droplets on the heat transfer performance through experiments. The result is that the average speed is the largest influence factor [5]. Mei Guohui *et al.* studied the effect of different jet directions on heat transfer, and the results showed that under low water flow density, the heat transfer effect at 90˚ is the best [6]. After the spray is ejected from the nozzle, it presents a fan-shaped distribution along the jet direction, and the droplet velocity and diameter distribution at different distances are different. Therefore, the main purpose of this article is to explore the influence of nozzle distance on heat transfer performance.

2. Model Theory

2.1. Establishment of Spray Model

This time, an air-blast-atomizer is used. The spraying process is that the liquid is broken into droplets under the action of compressed air. The gas adopts a continuous phase and the liquid droplets adopt a discrete phase. The continuous phase model includes three conservation laws of mass, momentum and energy Such as Formula (1)-(4). Establish a species transport model as Equation (5).

$$\frac{\partial y}{\partial x} + \nabla\left(\rho \boldsymbol{v}\right) = S_m \tag{1}$$

$$\frac{\partial \rho}{\partial t}\left(\rho \boldsymbol{v}\right) + \nabla \cdot \left(\rho \boldsymbol{v}\boldsymbol{v}\right) = -\nabla \rho + \nabla \cdot \bar{\bar{\tau}} + \rho \boldsymbol{g} + \boldsymbol{F} \tag{2}$$

$$\bar{\bar{\tau}} = \mu\left[\left(\nabla \boldsymbol{u} + \nabla \boldsymbol{u}\right) - \frac{2}{3}\nabla \cdot \boldsymbol{u}I\right] \tag{3}$$

$$\begin{aligned}&\frac{\partial}{\partial t}(\rho E)+\nabla\cdot\left(\boldsymbol{u}(\rho E+\rho)\right)\\&=\nabla\cdot\left(k_{eff}\nabla T-\sum_j h_j\boldsymbol{J}_j+\left(\overline{\overline{\tau}}_{eff}\cdot\boldsymbol{u}\right)\right)+S_h\end{aligned}\tag{4}$$

where S_m is the mass of the discrete phase entering the continuous phase, $\overline{\overline{\tau}}$ is the continuous corresponding force tensor, ρ is density, E refers to energy, $\rho\boldsymbol{g}$ and $\boldsymbol{F}$ are the momentum changes caused by gravity and volume force, respectively. k_{eff} corresponds to the realizable k-ε turbulence model

$$\frac{\partial}{\partial_t}(\rho Y_i)+\nabla\cdot(\rho\boldsymbol{u}Y_i)=-\nabla\cdot\boldsymbol{J}_i+S_i\tag{5}$$

where S_i is the source term produced by the droplet phase change.

The choice of turbulence model has a great influence on the effect of spray. The k-ε model is commonly used in current turbulence models. The standard k-ε turbulence model is used in this model.

Particles are affected by air in the process of motion, and the dynamic drag force model is used to consider the effect of droplet shape changes on the drag coefficient. Introducing the deformation value y, the drag coefficient expression is [7]:

$$C_D=C_{D,sphere}(1+2.632y)\tag{6}$$

2.2. Spray Cooling Heat Transfer Coefficient Model

The droplet directly collides with the high-temperature wall, and the heat exchange of a single droplet as Formula (7):

$$q_d=\frac{\pi dv^2}{6}\rho_1 h_{fg}\left\{0.027\exp\left[\frac{0.08\sqrt{\ln\left(\frac{we}{35}+1\right)}}{B^{1.5}}\right]+0.21k_d B\exp\left(\frac{-90}{We+1}\right)\right\}\tag{7}$$

The heat transfer per unit area per unit time caused by collision as Formula (8):

$$q_1=q_d\cdot n\tag{8}$$

where q_d refers to the energy absorbed by a single drop, n refers to the number of droplets, and q_1 refers to the total heat exchange.

2.3. Wall Liquid Film Model

The atomized droplets will form a liquid film when they touch the high temperature wall. There are two liquid film models to choose from in Fluent: Eulerian wall film model and Lagrange wall-film model .The maximum temperature of the liquid film will not exceed the boiling point of the liquid [8]. Yin bao zhen compared these two liquid film models and found that the latter is closer to the experimental results and is more suitable for liquid film simulation in spray cooling [9]. Therefore, the Lagrange liquid membrane model is used this time.

3. Simulation

3.1. Simulation of Steady-State Continuous-Phase Flow Field

Establish a heat source plane of 100 mm long, 100 mm wide and 5 mm high as shown in Figure 1. Figure 2 shows a cube area with a calculation area of 100 × 100 × 100.

The nozzle inlet is at the top center, and the jet direction is vertically downward along the y-axis. In order to make the result more in line with the actual situation, first use a steady state solution to obtain a stable flow field. Among them, the inlet pressure is 0.32 mpa, the four sides are pressure, the outlet gauge pressure is 0 mpa, and the environment is standard atmospheric pressure and temperature is 300 k. After the simulation, the speed cloud diagram is shown in Figure 3 and the speed change on the axis is shown in Figure 4. Take the cross section at a distance of 50 mm from the nozzle to get the speed change curve as shown in Figure 5.

The air is sprayed into the atmosphere through the nozzle, and the spray area presents a fan-shaped distribution, and the speed decreases continuously along the direction of the jet, and the speed changes at different distances. The velocity on the section parallel to the nozzle outlet presents a symmetrical distribution. In addition, it can also be seen from Figure 5 that the speed on the axis is the largest. In order to achieve the best cooling effect, the nozzle center should be at

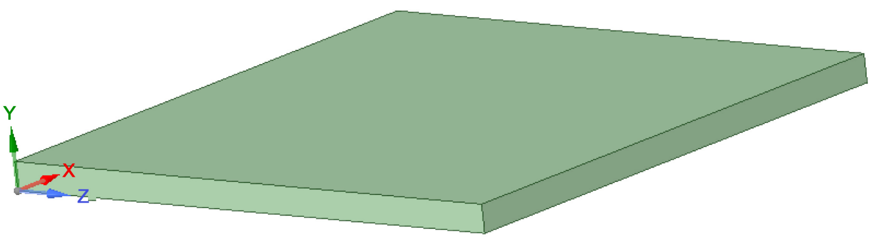

Figure 1. Heat source.

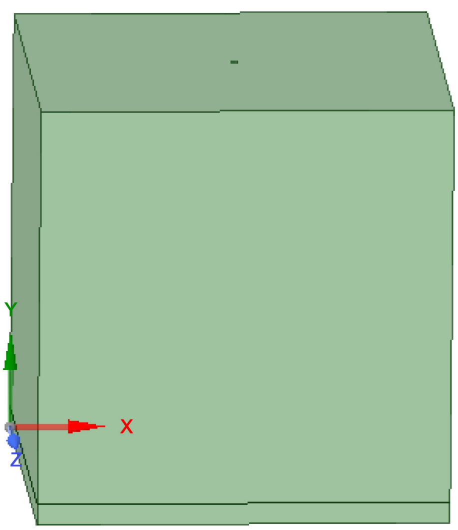

Figure 2. Physical model of computing area.

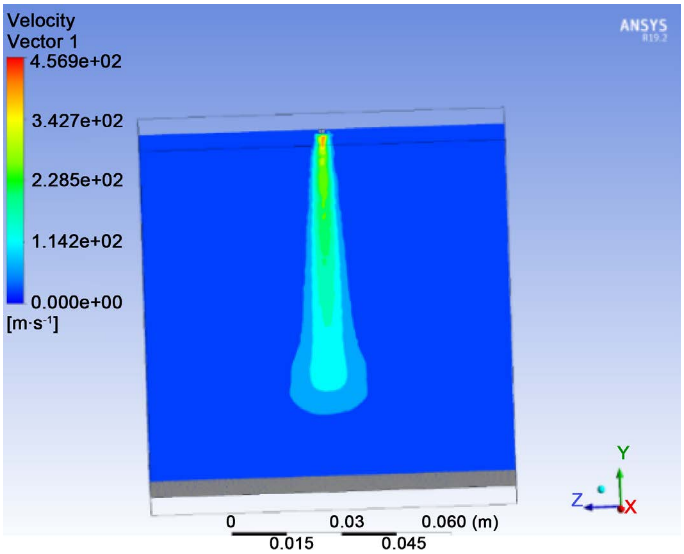

Figure 3. Cloud diagram of velocity distribution.

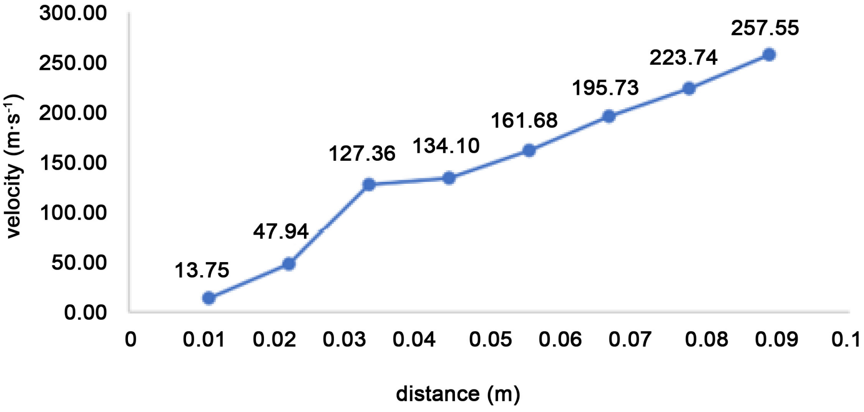

Figure 4. Speed change graph.

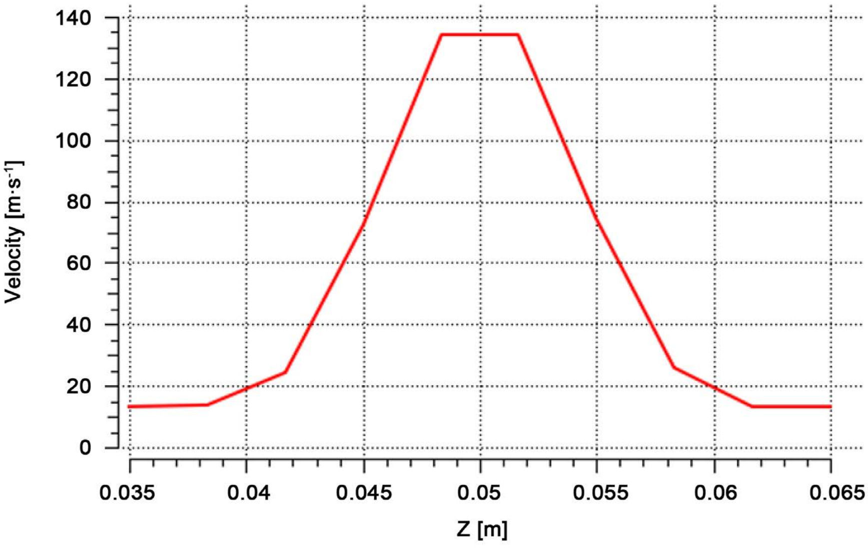

Figure 5. Velocity distribution on the section.

the highest temperature part to enhance the convection heat dissipation capability.

3.2. Cooling Simulation

After a stable continuous phase flow field is obtained, the particle phase is added through the DPM model to simulate the movement of the broken liquid particles in the flow field. The air-assisted nozzle model is selected, and the parameter settings of the model are shown in Table 1. The liquid is formed into a sheet by a nozzle, and air is then directed against the sheet to promote atomization. The addition of the external air stream past the sheet produces smaller droplets than without the air. In addition, air can help disperse the droplets and prevent collisions between droplets. The droplet breaking model adopts the TAB model, which is widely used in a variety of engineering jet calculations, and is also suitable for the conditions of this study. The temperature of the wall to be cooled is initialized with patch as shown in Figure 6(a), and the component transport model (species) and wall-film model are opened to simulate the phase change of the liquid at high temperature and the liquid film form on the high temperature wall.

After the simulation results are processed, the water mass fraction distribution cloud map is shown in Figure 7, and the heat source surface temperature distribution cloud map is shown in Figure 6(b). It can be seen from Figure 7 that the distribution of water is roughly fan-shaped around the nozzle, and a liquid film is formed after impacting the wall and spreading around.

The liquid film formed on the hot surface is divided into liquid film protrusion, impact liquid film, extrapolated liquid film and free liquid film from the inside to the outside. These areas can be found in the liquid film distribution map of the hot surface. The flow rate of the pushing liquid film is very high, which is beneficial to enhance the forced convection heat transfer of the liquid film. Judging from the cloud picture, the diffusion of water on the wall is not even to the surroundings. The reason may be that the air disturbance is increased after the liquid is added. The collision and aggregation of the droplets before reaching the wall affects the distribution of the droplets in space, so that the distribution of the droplets when they reach the wall is not symmetrically

Table 1. Main parameters of nozzle.

variable	value
Temperature (c)	26
Flow Rate (kg/s)	0.0035
Injector Inner Diameter (mm)	0.5
Injector Inner Diameter (mm)	2
Spray Half Angle (deg)	30
Relative Velocity (m/s)	80

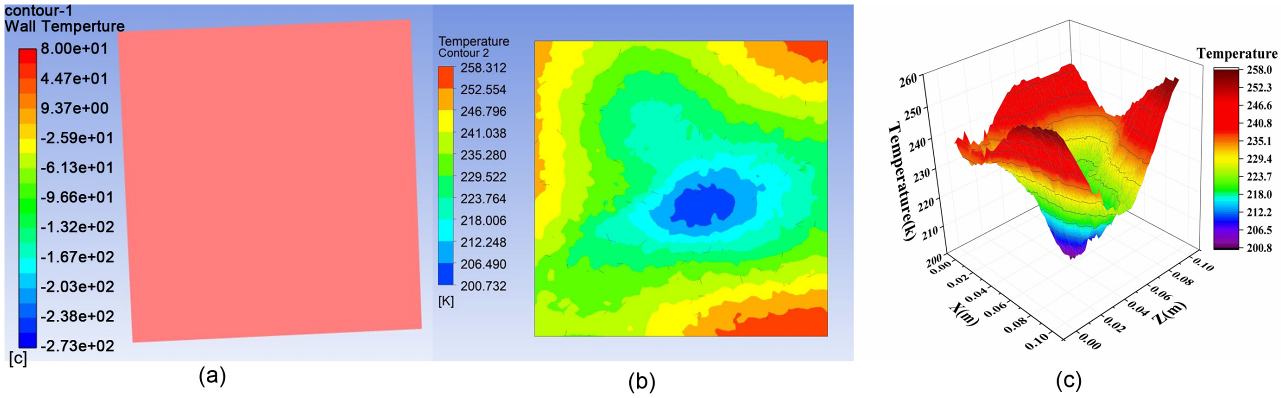

Figure 6. (a) Heat source temperature, (b) Wall temperature distribution, (c) Three-dimensional cloud map of wall temperature.

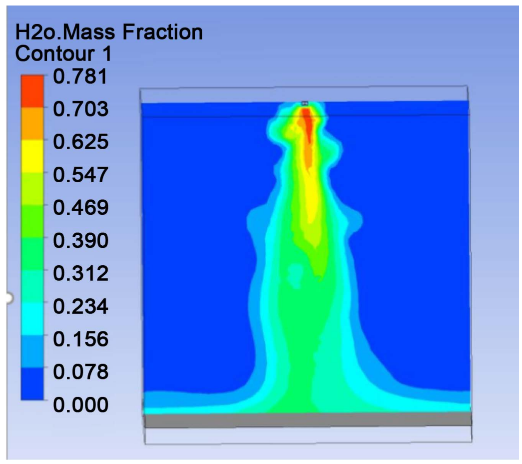

Figure 7. Distribution of water mass fraction.

distributed around the center of the nozzle. The asymmetric distribution of droplets will inevitably affect the heat dissipation to the wall of the heat source. This effect is well shown in Figure 6(b). In order to observe the temperature distribution of the wall more intuitively, export the temperature data of the wall to generate a three-dimensional cloud diagram as shown in Figure 6(c). The lowest temperature is not directly below the nozzle, and there is still a small area on the edge of the hot plate that has not been cooled, which indirectly indicates that the height of the nozzle has not reached the optimal height under the current working conditions.

4. Conclusion

Through the establishment of a model and simulation analysis, it can be found that when the vertical atomizing jet is cooled, there is a stagnant point on the surface of the heat source, and the liquid film undulates and diffuses around the stagnant point. The action mechanism of spray cooling is more complicated, and the various parameters of the droplets affect each other and jointly determine the heat exchange efficiency. When the droplet velocity is too high, the liquid film flows fast, but the vapor layer formed will prevent the subsequent droplets from contacting the surface of the heat source. On the other hand, when the spray distance is small, the spray range cannot cover the heat source boundary. If the spray distance is too large, the speed of the droplets has been excessively attenuated, and the heat dissipation capacity is greatly reduced. Therefore, it is necessary to determine the appropriate spray height according to the actual heat source size, air pressure and liquid flow rate.

Acknowledgements

Thanks for the help of colleagues.

Conflicts of Interest

The authors declare no conflicts of interest regarding the publication of this paper.

References

[1] Li, Q.Y., Wang, W. and Zhou, G.M. (2005) Research on Heat Dissipation Methods of Electronic Components. *Electronic Devices*, No. 4, 937-941.

[2] Geng, X.C. (2010) Simulation Analysis of Spray Cooling Liquid Film and Research on Its Heat Transfer Performance. Xidian University, Xi'an.

[3] Mudawar, I. and Estes, K.A. (1996) Optimizing and Predicting CHF in Spray Cooling of a Square Surface. *Journal of Heat Transfer*: *Transactions of the ASME*, **118**, 672-679. https://doi.org/10.1115/1.2822685

[4] Lin, L. and Ponnappan, R. (2003) Heat Transfer Characteristics of Spray Cooling in a Closed Loop. *International Journal of Heat and Mass Transfer*, **46**, 3737-3746. https://doi.org/10.1016/S0017-9310(03)00217-5

[5] Chen, R., Chow, L.C. and Navedo, J.E. (2002) Effects of Spray Characteristics on Critical Heat Flux in Subcooled Water Spray Cooling. *International Journal of Heat and Mass Transfer*, **45**, 4033-4043. https://doi.org/10.1016/S0017-9310(02)00113-8

[6] Mei, G.H., Meng H.G. and Xie, Z. (2004) The Effect of Spray Direction on Spray Cooling Heat Transfer. *Journal of Northeastern University*, No. 4, 374-377.

[7] Wang, Y.Q., Liu, M.H., Liu, D., *et al*. (2010) The Effect of Heat Dissipation Surface Temperature on the Heat Transfer Characteristics of No Boiling Zone in Spray Cooling. *Chinese Laser*, **37**, 115-120.

[8] Yin, B.Z. (2017) Numerical Simulation of Spray Cooling in Single-Phase Zone and Analysis of Influencing Factors. Dalian University of Technology, Dalian.

[9] Kate, R.P., Das, P.K. and Chakraborty, S. (2007) Hydraulic Jumps Due to Oblique Impingement of Circular Liquid Jets on a Flat Horizontal Surface. *Journal of Fluid Mechanics*, **573**, 247-263. https://doi.org/10.1017/S0022112006003818