Numerical characterization of multi-nozzle spray cooling

Yan Hou a,b, Yujia Tao a, Xiulan Hua a,b,*, Zhixiong Guo c

a Institute of Engineering Thermophysics, Chinese Academy of Sciences, P.O. Box 2706, Beijing 100190, China
b Graduate School of the Chinese Academy of Sciences, Beijing 100049, China
c Department of Mechanical and Aerospace Engineering, Rutgers, The State University of New Jersey, 98 Brett Road, Piscataway, NJ 08854, USA

A R T I C L E   I N F O

Article history:
Received 9 October 2011
Accepted 11 January 2012
Available online 20 January 2012

Keywords:
Multi-nozzle
Spray
Two-phase flow
Droplet
Simulation

A B S T R A C T

This work aims to study the characteristics of multi-nozzle spray cooling using CFD method based on the fundamentals of air flow and liquid droplet collision dynamics. A mathematical model for the two-phase flow was presented. The simulations were performed using a Eulerian–Lagrangian approach. Focus was placed on revealing the flow behavior with multiple nozzles, the droplet trajectory, and the influencing factors. The predictions by the present simulations matched well with the experimental results available in the literature, with a comparison showing deviation below 10%. It is concluded that the multi-nozzle spray characteristics including the Sauter Mean Diameter (SMD) of droplets and the mass weighted average droplet velocity are influenced by the droplet number and pressure, the droplet diameter and surface area. With increase of the number of nozzles and droplet diameter, the droplet SMD decreases and the mass weighted average droplet velocity increases. With increase of the mass flow, both the droplet SMD and the mass weighted average droplet velocity increase. The droplet number and pressure are very sensitive parameter to the droplet velocity distribution. Nevertheless, the droplet velocity distribution is not a monotonic function of the nozzle-to-surface distance. With increasing nozzle number, the change in droplet size is not appreciable; whereas the mass weighted average droplet velocity decreases and the distribution of the droplet size is improved significantly.

© 2012 Elsevier Ltd. All rights reserved.

1. Introduction

Spray cooling is important in a wide spectrum of industrial applications. It has great potentials in the cooling of high power systems. Traditionally, spray cooling was utilized to cool highly heated surfaces for equipments and processes in metallurgy, chemical and nuclear industry. Recently, it has received increasing attention in the development of modern technologies, such as the cooling of electronic devices and high power solid-state lasers. In the process of liquid spray, the liquid is usually injected into a chamber through an atomizing nozzle, resulting in the production of a spray comprising a large number of liquid droplets. The research of spray cooling involves three major subtopics: liquid atomization, interaction between droplets and surface, and the associated mass and heat transfer. A well-controlled particle size distribution is desirable in liquid atomization because the huge interfacial contact area between liquid and vapor can enhance heat and mass transfer rate and directly determines the cooling performance [1]. Hence, understanding the details of liquid atomization, such as the droplet diameter and velocity profiles, is essential in the design and performance optimization in spray cooling applications.

A number of review papers in spray cooling have addressed important issues focusing on the critical heat flux (CHF) and parametric studies [2–5]. However, little work has been done toward the disclosure of the spray characteristics. Experiments providing comprehensive information with regard to the droplet characteristics close to the injection point are very scarce since it is difficult to perform accurate measurements at locations very close to the atomizing nozzle. The experimental studies of Kurt and Mudawar [6] and Cheng et al. [7] both used the Phase Doppler Anemometry method to test a single nozzle and obtained the Sauter Mean Diameter (SMD) of droplets at different nozzle-to-surface distances. Such experiments are very costly, not to mention that it is not so easy to realize non-intrusive measurement. To this end, numerical simulation is a good means which can predict the spray characteristics and provide a theoretical guidance for experiments. However, numerical simulation of droplet dynamics in a two-phase flow is a particularly challenging problem because of the complicated process of film formation, droplet breakup, collision, coalescence and evaporation.

* Corresponding author. Tel./Fax: +86 10 8254 3108.
E-mail address: hx@mail.etp.ac.cn (X. Hua).

1359-4311/$ – see front matter © 2012 Elsevier Ltd. All rights reserved.
doi:10.1016/j.applthermaleng.2012.01.030
The current theoretical investigations about droplet size distribution were focused on the industry of spray combustion. Two theoretical approaches were available to solve the two-phase flow problem, namely the Eulerian and the Lagrangian methods [1]. A review by Gouesbet and Berlemont [8] revealed the advantages of each approach. The Eulerian code is usually fast running, but the dispersion tensor introduced in a transport equation for mean number-densities lacks generality. The Lagrangian approach is well suited to the simulation of turbulent flows, which followed the dispersed particles trajectories [1, 8]. Dhuchakallaya and Watkins [9] presented the development and implementation of spray combustion modeling based on the spray size distribution moments. They solved the governing equations for gas phase and liquid phase employing the finite volume method based on a Eulerian framework. Guo et al. [1] explored the possibility of simulating the agglomeration process in a spray using the Lagrangian particle tracking method. Their model system consisted of a spray nozzle within a uniform air flow in a square-section chamber. Kolaitis and Fonti [10] simulated a confined, atmospheric pressure, turbulent and evaporating spray test case using an in-house Eulerian—Lagrangian CFD code. Most of the previously-mentioned works were aimed to solve the single nozzle spray problem. In some cases, cooling of the surfaces with high heat flux requires the use of multiple nozzles. Ma et al. [11] and Amon et al. [12] presented that using multiple nozzles could effectively control the spray distribution on the heated surface and subsequently improve the cooling uniformity. Nguyen et al. [13] found that in the case with same mass flow rate, increasing the number of nozzles could enhance the critical heat flux. Unfortunately, few simulations that can well predict multi-nozzle spray characteristics have been reported in the literature, to the authors’ knowledge. The demand becomes pressing for in-hand accurate and flexible models that can reliably describe the two-phase flow behavior in multi-nozzle spray cooling.

The objective of the present work is to reveal the multi-nozzle spray characteristics using the computational fluid dynamics (CFD) simulations. A two-phase flow model was developed based on the mixed Eulerian—Lagrangian approach for a three-dimensional spray system including at least two nozzles in a cuboid chamber. The effects of nozzle inlet pressure, mass flux, nozzle-to-surface distance and nozzle number on the droplet SMD and the mass weighted average velocity of droplets were examined. The present model was validated through comparison with available experimental data in the literature [6, 7] under the same conditions.

2. Mathematical model

The two-phase flow of multi-nozzle spray in a confined chamber was simulated using the Lagrangian discrete phase model of the Fluent software which follows the Euler—Lagrange computational approach. The model consists of continuous fluid and dispersed phases. The fluid phase (air) is modeled by solving the Navier—Stokes equations using the Eulerian approach, and the dispersed phase is solved by tracking a large number of representative liquid droplets using the Lagrangian approach. The dispersed phase can exchange momentum and mass with the fluid phase, with conventional gas-particle coupling. The model contains the assumptions that the particle—particle interactions and the effect of the particle volume fraction on the fluid phase are negligible.

2.1. Continuous phase

The continuous phase is treated as an unsteady, incompressible and turbulent air flow. It is described by the time-averaged Navier—Stokes equations based on the standard k-ω turbulence model and solved using the SIMPLE algorithm. The mass and momentum conservation equations are set up as follows:

The mass conservation equation is

\[
\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \mathbf{U}) = 0
\]  

(1)
The momentum conservation equations are

\[
\frac{\partial (\rho u_x)}{\partial t} + \nabla \cdot (\rho u_x U) = \nabla \cdot (\eta \nabla u_x) - \frac{\partial p}{\partial x} + S_x
\]

\[
\frac{\partial (\rho u_y)}{\partial t} + \nabla \cdot (\rho u_y U) = \nabla \cdot (\eta \nabla u_y) - \frac{\partial p}{\partial y} + S_y
\]

\[
\frac{\partial (\rho u_z)}{\partial t} + \nabla \cdot (\rho u_z U) = \nabla \cdot (\eta \nabla u_z) - \frac{\partial p}{\partial z} + S_z
\]

(2)

### 2.2. Dispersed phase

The Lagrangian approach is utilized for the calculation of the dispersed phase. A multitude of computational parcels, each one representing a group of real droplets that have the same properties, is traced through the flow field [10]. Each parcel’s trajectory, as well as mass transfer to and from it, is calculated by solving the instantaneous droplet motion equations below. In the model, the drag force and the gravitational force are taken into account. The particle force balance equations are

\[
du_x = F_D(u_x - u_{p,x}) + \frac{g_x}{\rho_p} (\rho_p - \rho)
\]

\[
du_y = F_D(u_y - u_{p,y}) + \frac{g_y}{\rho_p} (\rho_p - \rho)
\]

\[
du_z = F_D(u_z - u_{p,z}) + \frac{g_z}{\rho_p} (\rho_p - \rho)
\]

(3)

where \( F_D = \frac{18 \mu C_D \text{Re}}{\rho_p d_p^2} \frac{24}{24} \)

The particle trajectory equations are

\[
dx = u_{p,x}
\]

\[
dy = u_{p,y}
\]

\[
dz = u_{p,z}
\]

(4)

### 2.3. The coupling between continuous and dispersed phases

The two-way coupling between the two-phases and its impact on both the dispersed phase trajectories and the continuous phase flow are examined. The continuous phase flow pattern is impacted by the discrete phase and vice versa. The coupled approach is described as follows. First the continuous flow is computed, and then creates the droplet particles as the discrete phase. When the trajectory of a particle is computed, the mass and momentum gained or lost by the particle stream that follows the trajectory can be tracked and these quantities can be incorporated as a source term in the subsequent continuous phase calculations. Because the continuous phase and the discrete phase impact each other, the effect of the discrete phase trajectories on the continuum is also incorporated. This two-way coupling is accomplished by alternately solving the equations of the discrete and continuous phases until a converged coupled solution is achieved. In the coupled approach, the mass and momentum transfer from the continuous phase to the discrete phase are computed by examining the change in mass and momentum of a particle as it passes through each control volume in the model. The mass and momentum transfer equations between the two-phases are shown in Eqs. (5) and (6).

\[
M = \frac{\Delta m_p}{m_{p,0}}
\]

\[
F_x = \sum \left( \frac{18 \mu C_D \text{Re}}{\rho_p d_p^2} (u_{p,x} - u_{i,x}) \right) m_p \Delta t
\]

\[
F_y = \sum \left( \frac{18 \mu C_D \text{Re}}{\rho_p d_p^2} (u_{p,y} - u_{i,y}) \right) m_p \Delta t
\]

\[
F_z = \sum \left( \frac{18 \mu C_D \text{Re}}{\rho_p d_p^2} (u_{p,z} - u_{i,z}) \right) m_p \Delta t
\]

(5)

### 2.4. Boundary conditions

The case studied is a confined cuboid chamber with multiple nozzles. Two round outlets, defined as pressure outlets, are set at the bottom of the chamber. The pressure-swirl-atomizers are placed above the bottom surface with a specified nozzle-to-surface distance. The wall-film boundary condition is applied for the near-wall boundaries. Film particles are assumed to be in direct contact with the wall surface. The particles in the wall-film also satisfy the conservation equations of momentum and mass. The momentum equation is

\[
p e \frac{d\bar{u}_p}{dt} + e \left( \nabla p \right)_p = \tau_f \bar{u}_f + \tau_w \bar{u}_w + \bar{P}_{\text{imp},a} - M_{\text{imp},a} \bar{u}_p + \bar{F}_{\text{n,a}} + \rho e \left( g - \bar{a}_w \right)
\]

(7)

The mass transfer from the film is

\[
N_i = B_f (C_{1,a} - C_{1,w})
\]

(8)

### 3. Simulation results

The three-dimensional computational domain, measuring 0.04 m (length) \times 0.02 m (width) \times 0.02 m (height), was discretized using the structured hexahedral meshes as shown in Fig. 1. The number of the nozzles can be 1, 2 or 3. Considering the flow symmetry, half of the computational domain was calculated. Water is used as the working liquid and sprays in just from the top of the chamber. The first test case refers to a single hollow nozzle in turbulent flow condition. The droplet SMD was examined using the two-phase CFD code with three different grid arrangements. The

![Fig. 1. The sketch for simulation domain.](image)
number of the nodes was \(2.42 \times 10^6\), \(1.53 \times 10^6\) and \(5.31 \times 10^5\), respectively. The time step used is \(10^{-5}\) s. The predictions for the temporal evolution of the droplets’ SMD by the three different grids were illustrated in Fig. 2. It is seen that the calculation tends to be stable after 0.04 s. The computation results utilizing the grid arrangement of \(1.53 \times 10^6\) do not deviate more than 1.6% as compared to the finer grid size of \(2.42 \times 10^6\). As compared to the coarser grid size of \(5.31 \times 10^5\), the max deviation is about 2.1%. Thus, considering the increased computational time, the grid size of \(1.53 \times 10^6\) was selected for all the simulations thereafter.

The second test case was performed to verify the validity of the CFD code. The computational results were compared with the experimental results obtained by Estes and Mudawar [6], in which the droplet SMD was correlated with the pressure drop across the spray nozzles as shown in Fig. 3. The experimental correlation was fitted using FC-72 and water as the working fluid, respectively, with a mean relative error of 12.4%. Fig. 3 shows a good agreement between the present model predictions and the experimental results. The averaged deviation between these two results is below 10%.

3.1. Effect of inlet pressure on multi-nozzle spray

The inlet pressure of each nozzle is a dominant parameter influencing the spray characteristics. In order to study the effect of the inlet pressure on the performance of the multi-nozzle spray, the SMD of the droplets in the spray cone and the mass weighted average droplet velocity at the red section as indicated in Fig. 1 under different inlet pressures using two and three nozzles were numerically simulated. Figs. 4 and 5 show their variations with the inlet pressure, respectively. It is found from Fig. 4 that the droplet SMD decreases with the increase of the inlet pressure and the variations are similar for both two and three nozzle sprays. As shown in Fig. 5, however, the mass weighted average droplet velocity increases with the increase of the inlet pressure. It is mainly because that the increase of the inlet pressure increases the atomization energy, leading to a high energy for droplet breakup. As a result, the mass weighted average droplet velocity increases and the droplet SMD decreases. When the inlet pressure is over \(5 \times 10^5\) Pa, the droplet velocity in the case of two nozzles is greater than that in the case of three nozzles.

3.2. Effect of mass flux on multi-nozzle spray

The effect of mass flux on multi-nozzle spray was disclosed in Figs. 6 and 7, in which the droplet SMD and the mass weighted average droplet velocity with an inlet pressure of \(6 \times 10^5\) Pa are shown, respectively. It is found that when the mass flux of each nozzle increases from 0.015 kg/s to 0.040 kg/s, the droplet SMD
increases almost linearly proportional to the mass flux for the two and three nozzle sprays. The two curves in Fig. 6 coincide in general. The mass weighted average droplet velocity also increases with the increase of the mass flux. But the mass weighted average droplet velocity with two nozzles is greater than that with three nozzles. It is mainly because the interference among three nozzles is much more intense than that of two nozzles. The droplets impact each other and then splash to all directions, which will counteract the droplet velocities.

3.3. Effect of nozzle-to-surface distance on multi-nozzle spray

In order to characterize the effect of the nozzle-to-surface distance on the spray with multiple nozzles, the droplet SMD and the droplet velocity distributions from center to the side along the horizontal axis of the bottom were numerical simulated for five different cases, with the nozzle-to-surface distance being 0.007 m, 0.009 m, 0.011 m, 0.013 m and 0.014 m, respectively. The computation results are shown in Figs. 8—10.

It is seen from Fig. 8 that the droplet SMD is not so sensitive to the nozzle-to-surface distance. The droplet SMD varies in a range of $(4.2 \times 10^{-4})$ to $(4.4 \times 10^{-4})$ m with the increase of the nozzle-to-surface distance for both the cases of two and three nozzles. However, the droplet velocity profile is very sensitive to the nozzle-to-surface distance as shown in Fig. 9. A small change of the nozzle-to-surface distance substantially alters the distribution of the droplet velocities. Moreover, the droplet velocity distribution is not a monotonic function of the nozzle-to-surface distance. It is found from Fig. 9 that when using two nozzles, there exist two peaks from the center to the side along the horizontal axis of the bottom at different nozzle-to-surface distances. When the distance is 0.014 m, the velocity variation in the curve is most significant. At the smallest distance (0.007 m), such a variation is also large. The variation in the velocity tends to be smallest when the distance is 0.011 m. The conclusion is also the same for the three nozzle spray. From Fig. 10, it is observed that the mass weighted average droplet velocity varies with the distance, with distance at 0.011 m being the lowest, although the effect of the distance on the mass weighted average droplet velocity is not as significant as on the velocity distribution. Therefore, it can be concluded that the nozzle-to-surface distance is an important factor influencing the spray velocity distribution.

3.4. Effect of nozzle numbers on spray characteristics

In order to ascertain the effect of the nozzle number on spray, the droplet SMD and droplet velocity distribution for three cases with different nozzle numbers were simulated. Fig. 11(a)–(c) show the droplet SMD distribution at the cross-section 0.002 m away from the nozzle when the nozzle number is 1, 2 and 3, respectively. It is found that in the case of single nozzle, the droplet size distributed in the range of $(3.2 \times 10^{-4})$ m accounted for approximately 42%. The percentage increases to approximately 44% when adding a more nozzle. If there are three nozzles, the droplet size distributed in the same range accounted for approximately 49%. Hence, the increase of the nozzle number can improve the uniformity of droplet size distribution.

Fig. 12(a)–(c) show the droplets velocity distributions from the center to the side along the horizontal axis of the bottom for three cases with different nozzle numbers, respectively. It is seen that in the case of a single nozzle, the velocities of about 20 percent of the droplets are in the rage of 0.4–0.66 m/s. The droplet velocity increases first, and then decreases rapidly from the center to the side along the horizontal axis of the bottom, resulting in the overall droplet velocity distribution uneven. When adding one nozzle, the velocities of approximate 50 percent of the droplets is in the rage of approximately 49%. Hence, the increase of the nozzle number can improve the uniformity of droplet size distribution.

Fig. 13(a)–(c) show the mass weighted average droplet velocity distribution for three cases with different nozzle numbers, respectively. It is seen that in the case of a single nozzle, the velocities of about 20 percent of the droplets are in the rage of 0.4–0.66 m/s. The droplet velocity increases first, and then decreases rapidly from the center to the side along the horizontal axis of the bottom, resulting in the overall droplet velocity distribution uneven. When adding one nozzle, the velocities of approximate 50 percent of the droplets are in the rage of 0.25–0.78 m/s. There exist two velocity transition zones from the center to the side. The droplet velocities in these two transition zones are in the rage of 0.4–0.7 m/s. Thus, the overall droplet velocity distribution of two nozzles is more uniform than that of
Fig. 9. The effect of nozzle-to-surface distance on the droplet velocity distribution.

Fig. 10. The effect of nozzle-to-surface distance on the mass weighted average droplet velocity.

Fig. 11. The effect of the nozzle number on the droplet SMD distribution.
single nozzle. When the nozzle number is three, the velocities of more than 75 percent are in the rage of 0.1–0.45 m/s. There exist three velocity transition zones from the center to the side. And the droplets velocity distributions tend to be most uniform than the above two cases. It can be concluded that increasing the nozzle number can improve the uniformity of droplet velocity distribution significantly.

Fig. 13 shows the variations of droplet SMD and mass weighted average velocity of droplets with the number of nozzles at the same nozzle-to-surface distance of 0.011 m. It is found that when the nozzles are arranged linearly, the droplet size varies not much while the mass weighted average droplet velocity decreases clearly when increasing the number of nozzles.

4. Conclusions

In this study, the multi-nozzle spray characteristics were investigated using the CFD simulations. A two-phase flow mathematical model was presented based on the Eulerian–Lagrangian approach. Several grid arrangements were tested to ensure grid independence. The data obtained from the model were compared with the experimental results to verify the computational method. The effects of nozzle inlet pressure, mass flux, nozzle-to-surface distance and nozzle numbers on the droplet SMD, droplet velocities and their distributions were examined. The conclusions are summarized as follows.

1) The model was tested using available experimental data. It showed a deviation below 10%.
2) The droplet SMD decreases while the mass weighted average droplet velocity increases with the increase of the inlet pressure in both the cases using two and three nozzles. The inlet pressure influences more significantly on the mass weighted average droplet velocity than on the droplet SMD.
3) The droplet SMD increases almost linearly proportional to the mass flux. With the increase of the mass flux, the mass weighted average droplet velocity also increases.
4) The droplet velocity distribution is not a monotonic function of the nozzle-to-surface distance. The velocity value and the velocity distributions are two different targets when optimizing the nozzle-to-surface distance.
5) With a linear arrangement of nozzles, increasing nozzle number will decrease the mass weighted average droplet velocity; but not affect the droplet size obviously. The increase of nozzle number can improve the distributions of droplet size and droplet velocity significantly.
Acknowledgements

This research is supported by the National Natural Science Foundation of China (50906083) and the National Basic Research Program of China (2011CB710705).

References