Numerical simulation of three-dimensional transient flow characteristics for a dual-fluid atomizer

Yu Minghao\textsuperscript{a}, Wang Rui\textsuperscript{a}, Liu Kai\textsuperscript{a} and Yao Jiafeng\textsuperscript{b}

\textsuperscript{a}School of Mechanical and Precision Instrument Engineering, Xi’an University of Technology, Xi’an, People’s Republic of China; \textsuperscript{b}College of Mechanical and Electrical Engineering, Nanjing University of Aeronautics and Astronautics, Nanjing, People’s Republic of China

**ABSTRACT**

This paper aims to study the three-dimensional transient flow-field properties of the gas–liquid dual-fluid atomizer, which has high flow capacity and low energy consumption. The two-phase atomization process of air and water, the negative pressure gradient near the porous medium and the recirculation flow inside the atomizer were numerically simulated and analyzed. Primarily, the influence of the air flow velocity on the water flow velocity at the water inlet was numerically studied. Furthermore, it was proven that the simulated results agree well with the experimental data. Thereafter, the variations in the inlet’s water flow velocity and vortex current zone size were studied in detail with changes in the orifice diameter and outlet pipe length. The results indicated that the water absorption improves as the air flow velocity increases. There is a suitable orifice diameter to maximize water flow and maximize the secondary atomization. The water flow velocity becomes larger and the secondary atomization improves as the length of the outlet pipe increases. The larger the vortex region, the smaller the droplet size. This study could provide theoretical data and guidance for the optimization design of dual-fluid atomizers.

**ARTICLE HISTORY**

Received 21 May 2019
Accepted 7 September 2019

**KEYWORDS**

dual-fluid atomizer; transient; vortex; numerical simulation

1. Introduction

Atomizers are widely used in modern life and industrial fields, such as for air humidification, farm irrigation, fire extinguishers and absorption of toxic gases (Mohan, Yang, & Chou, 2014; Yao, Yu, Zhao, Kawahara, & Sadatomi, 2017; Zhang, Wu, & Pu, 2006). Atomizers are mainly of four types: flat nozzle atomizers, pressure swirl atomizers, gas-assisted atomizers and bubble atomizers (Yuan, Tan, Xu, & Yuan, 2015). Many studies on bubble atomizers have been carried out and have achieved many results (Liang, 2005; Liang & Wang, 2008; Yuan et al., 2015). However, there are few reports on gas-assisted atomizers. It is difficult to characterize the internal flow process of the gas-assisted atomizer by general experimental methods because of its complexity and fast gas flow velocity. However, visualization of the atomization process can be achieved by numerical simulation (Yu, Shademan, Barron, & Balachandar, 2012). Therefore, the optimal design of such atomizers is usually carried out and assisted by numerical simulation to save time and reduce economic costs (Xia, Wang, Zhang, & Ge, 2018).

The dual-fluid atomizer studied in this paper is a gas-assisted atomizer. There are two kinds of atomization process. One is primary atomization, where the liquid becomes droplets or irregularly shaped liquid separated by the high-speed gas through the porous medium. The other is secondary atomization, where droplets are further broken into smaller ones by the high-speed gas. From the available literature, primary atomization has mainly been simulated by volume of fluid (VOF) multiphase flow (Fritsching, 2004; Liu et al., 2015; Zeoli, Tabbara, & Gu, 2011), and secondary atomization mainly by the discrete phase method (DPM) (Firmansyah et al., 2012; Thompson, Hassan, Rolland, & Sienz, 2016). This has been studied by some scholars through experiments. For example, Nguyen and Rhodes (1998) found that these two factors have less effect on atomization if the length and the diameter of the mixing chamber are 20–45 mm and 17–23 mm, respectively. The experimental results of Kushari (2010) revealed that the droplet size decreases as the length of the mixing chamber increases. Both Ferreira, Garcia, Barreras, Lozano, and Lincheta (2009) and Kushari (2010) have shown that the ratio of the outlet area of the internal mixing dual-fluid atomizer to the air injection area affects the average droplet size of the spray, and the bubble atomizer (Chin & Lefebvre, 1995) has the same effect. In addition, Shafaee, Banitabaei, Esfahani, and Ashjaee (2011) found that the spray cone angle decreased slightly with increasing length of the mixing chamber. El-Shanawany and Lefebvre (1980) conducted
a series of experiments on a blower atomizer, and concluded that the atomization quality decreased as the size of the atomizer was increased. In summary, although all these results are helpful for the optimal design of the dual-fluid atomizer, most of these studies are about the steady-state flow characteristics of the atomizer. There is still no detailed and comprehensive study on the three-dimensional (3D) transient flow characteristics inside the dual-fluid atomizer. Therefore, it is necessary to perform this study to improve our understanding of the 3D transient turbulent flow properties, and to enable further improvements in the construction of air–liquid dual-fluid atomizers.

The present study evaluates the performance of the atomizer from three aspects: water absorption, atomized particle size and atomization efficiency. Water absorption and particle size were used to evaluate the performance of the two-fluid atomizer with different construction parameters. FLUENT software was employed to simulate the internal flow of the atomizer. The computational fluid domain was constructed by the software SOLIDWORKS. The interface tracking function of the VOF multiphase flow model was used to control the main atomization process of the dual-fluid atomizer. First, the influence of air flow velocity on the water flow velocity of the inlet is studied through numerical simulation. Second, the simulated results are compared with relevant experimental data to verify the accuracy of the numerical methods used in this study. Third, the changes in the inlet’s water flow velocity and vortex current zone size are studied in detail, with variations in the orifice diameter and outlet pipe length. Finally, the influences of the physical structure of each part of the dual-fluid atomizer on the water velocity of the inlet are summarized to accurately grasp the physical characteristics of the gas–liquid two-phase flow inside the atomizer. The numerical results could provide basic theoretical data and guidance for the optimal design of the dual-fluid atomizer.

2. Physical model of the dual-fluid atomizer

The physical structure of the dual-fluid atomizer is shown in Figure 1. It includes five main parts: inlet pipe, orifice, porous medium, shell of water inlet and outlet pipe. The working principle of the atomizer is as follows.

The compressed air enters via the inlet pipe, passing through the orifice, forming a negative pressure in the porous medium and generating vortex current, and a pressure difference using the external atmospheric pressure. Thereby, water is inhaled through the porous medium to generate primary atomization, and secondary atomization occurs when a droplet is separated from the porous medium and crushed by high-speed air.

Compared with other atomizers, the dual-fluid atomizer has the advantages of a good atomization effect and uniform atomization particle diameter (Lefebvre, 1989). To simulate the internal flow characteristics of the atomizer more accurately, we established a hybrid dense computational mesh, which is mainly hexahedral, for the simulation. The total number of grids is about 305,367. There are 208,455 inlet pipe grids, 29,883 medium pipe grids and 74,010 outlet pipe grids. To check the effect of grid size on the numerical accuracy of the present simulations, several additional simulations were performed, with the total grid numbers being 70,145, 90,622, 120,661, 197,375, 246,503, 305,367, 448,038 and 668,524 for case 1. It was found that when the number of grids was greater than 250,000, the numerical results such as the water velocity changed only very slightly. So, to save the computational cost, about 300,000 grids were used in the computational mesh for all simulations in this study. The calculation domain of the flow-field and the 3D mesh is shown in Figure 2.

3. Mathematical model

3.1. Governing equations

In this paper, it is assumed that the flow state in the dual-fluid atomizer is transient and turbulent. The fluid
is incompressible and unaffected by heat exchange. Then, the mass conservation and momentum conservation equations can be written as follows.

Mass conservation equation:

\[
\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} + \frac{\partial w}{\partial z} = 0 \quad (1)
\]

where \( u, v \) and \( w \) are the velocity components in the \( x, y \) and \( z \) directions.

Momentum conservation equations:

\[
\frac{\partial (\rho u)}{\partial t} + \text{div}(\rho u U) = -\frac{\partial p}{\partial x} + \frac{\partial \tau_{xx}}{\partial x} + \frac{\partial \tau_{yx}}{\partial y} + \frac{\partial \tau_{zx}}{\partial z} + F_x \quad (2)
\]

\[
\frac{\partial (\rho v)}{\partial t} + \text{div}(\rho v U) = -\frac{\partial p}{\partial y} + \frac{\partial \tau_{xy}}{\partial x} + \frac{\partial \tau_{yy}}{\partial y} + \frac{\partial \tau_{zy}}{\partial z} + F_y \quad (3)
\]

\[
\frac{\partial (\rho w)}{\partial t} + \text{div}(\rho w U) = -\frac{\partial p}{\partial z} + \frac{\partial \tau_{xz}}{\partial x} + \frac{\partial \tau_{yz}}{\partial y} + \frac{\partial \tau_{zz}}{\partial z} + F_z \quad (4)
\]

where \( p \) is the fluid pressure; \( \tau_{xx}, \tau_{xy}, \tau_{zx} \) are the viscous stress components; and \( F_x, F_y, F_z \) are the inertial forces in the \( x, y \) and \( z \) directions. In this study, only gravity is considered as the inertial force, so \( F_x = F_y = 0 \) and \( F_z = -\rho g \).

### 3.2. Turbulence model

Several turbulence models, such as the standard \( k-\epsilon \), realized \( k-\epsilon \), renormalization group (RNG) \( k-\epsilon \), standard \( k-\omega \), baseline (BSL) \( k-\omega \) and shear stress transport (SST) \( k-\omega \) models (Tian & Lu, 2013), are available and were selected for the present study. After analysis, as shown in Figure 3, the SST \( k-\omega \) model, which was proposed by Menter (1993), seems the most suitable and accurate for the present study. The SST \( k-\omega \) model combines the advantages of the \( k-\omega \) model in the near-wall calculation and the advantages of the \( k-\epsilon \) model in the far-field calculation, while increasing the lateral dissipation derivative term. It considers the transport process of turbulent shear stress in the definition of turbulent viscosity. The SST \( k-\omega \) model is more applicable for flow calculations with back-pressure gradient, airfoil calculations and transonic speed shock calculations. Because the air expands too rapidly after the air passes through the orifice for this atomizer, the air no longer flows close to the outlet pipe wall, and there is a reverse pressure gradient. So, the SST \( k-\omega \) model was adopted in all the simulations performed in this study. The equations of the model can be expressed as follows.

The SST model’s transport equations are given by (Goldberg & Batten, 2015):

\[
\frac{D(\rho k)}{Dt} = \tilde{P}_k - \beta_k^* \rho k \omega + \nabla \cdot [\kappa \omega \nabla k] \quad (5)
\]

\[
\frac{D(\rho \omega)}{Dt} = \frac{\gamma}{\nu_t} - \beta \rho \omega^2 + \nabla \cdot [\kappa + \sigma_{k\omega} \frac{1}{\omega} \nabla k \nabla \omega] + 2(1 - F_1) \rho \sigma_{k\omega} \frac{1}{\omega} \nabla k \nabla \omega \quad (6)
\]

where the eddy viscosity is

\[
\nu_t = \frac{\mu_t}{\rho} = \frac{a^1_k}{\max\{a^1_k, SF^2\}} \quad (7)
\]

and the production terms are

\[
P_k = [\mu_1(U_{ij} + U_{ji} - \frac{2}{3} U_{k,kh})]U_{ij} \quad (9)
\]

\[
\tilde{P}_k = \min\{P_k, 10^8 \beta^* \rho k \omega \} \quad (10)
\]

\( F_1 \) and \( F_2 \) are the blending functions, which depend on the wall distance, and all other symbols are given in the list of symbols at the end of this article.

### 4. Boundary conditions

The multiphase flow VOF model was adopted in the simulation. The air inlet boundary was set as the velocity inlet boundary condition. The air speed was increased from 25 m/s to 60 m/s. The water inlet was set as the pressure inlet boundary condition. The inlet pressure was the standard atmospheric pressure. The pressure and other flow properties were interpolated from inner grids at the outlet. The surface tension coefficient of the two gas–liquid phases was set to 0.075 N/m (Wright, Bean, & Aguilera, 2008). The working fluid is air and water.
Table 1. Physical properties of fluids.

| Fluid | Density (kg/m³) | Viscosity (Pa•s) |
|-------|----------------|-----------------|
| Water | 998.2          | $1.003 \times 10^{-3}$ |
| Air   | 1.225          | $1.7894 \times 10^{-5}$ |

and the physical properties of air and water are shown in Table 1.

5. Results and discussion

5.1. Influence of air flow velocity on the absorbed water

Figure 3 shows the influence of air velocity on the water velocity at the water inlet. Six simulations were carried out with six turbulence models: the standard $k$-$\epsilon$, real-
ized $k$-$\epsilon$, RNG $k$-$\epsilon$, standard $k$-$\omega$, BSL $k$-$\omega$ and SST $k$-$\omega$ models. The simulation conditions are as follows: $l_{in} = 20.5$ mm, $l_{out} = 20.5$ mm, $d_1 = 21$ mm, $d_2 = 5$ mm, $D = 13.8$ mm and $\phi = 0.1$ (Yao et al., 2017). Finally, the simulated results of these six cases were compared with the measured experimental data (Yao et al., 2017) in Figure 3 to check the evaluation accuracy of the different turbulence models.

The water velocity increased at the water inlet with the increase in the supplied air velocity, as shown in Figure 3. This is because the high-speed air flow will generate a negative pressure at the orifice when the air flows through the orifice, and then the increased negative pressure will result in an increase in the water velocity of the inlet. The simulated values of the six turbulence models are compared with the experimental values (Yao et al., 2017) in this paper, as shown in Figure 3. It is found that the simulated values of the SST $k$-$\omega$ model are consistent with the experimental values, and the relative error is 6.7%.

5.2. Atomization process

Figure 4 shows the time-variation distribution of the water volume fraction inside the atomizer. The simulation was carried out under following conditions: $l_{in} = 20.5$ mm, $l_{out} = 20.5$ mm, $d_1 = 21$ mm, $d_2 = 5$ mm, $D = 13.8$ mm, $\phi = 0.1$ and $V_{air} = 50$ m/s (Yao et al., 2017). The initial state of the entire fluid domain is nothing but air.

Water begins to enter the porous medium from the water inlet owing to the negative pressure at about 0.8 ms (Figure 4a); then, water enters the outlet pipe through the porous medium at about 1.8 ms, and some of the water is rapidly atomized (Figure 4b); there is water flowing through the shell of water inlet which is farthest from the inlet at about 13 ms, and atomization occurs through the porous medium (Figure 4c). At about 20 s, the shell of water inlet is almost completely filled with water, and the flow velocity of the water inlet is stable. The VOF model can provide the water fraction in the outlet pipe; as the water fraction in the outlet pipe may be 30%, one can speculate that many small droplets will exist (Figure 4d).  

As shown in Figure 5(a), the distribution of the water phase volume shows clearly that a significant eddy flow is formed in the outlet pipe after the water enters the porous medium. As will be shown later (see Section 5.4),
Figure 5. (a) Water phase volume fraction changes with time; (b) pressure animation; (c) velocity animation. (Animations are as the attachments in the online article.)
Table 2. Different orifice conditions.

| Case | \( l_w \) (mm) | \( l_{\text{out}} \) (mm) | \( d_1 \) (mm) | \( d_2 \) (mm) | \( V_{\text{air}} \) (m/s) | \( \phi \) |
|------|----------------|----------------|-------------|-------------|----------------|-------|
| 1    | 20.5          | 20.5          | 21         | 5           | 100            | 0.1   |
| 2    | 15            | 15            | 14         | 5           | 100            | 0.1   |

Figure 6. Influence of orifice diameter on flow velocity at the inlet.

The eddy flow has a significant effect on the refinement of the diameter of the atomized particles. Figure 5(b) shows the internal pressure distribution of the atomizer. At the beginning, the gauge pressure is 0; thereafter, a negative pressure is generated at the porous medium; then, the negative pressure gradually expands to the entire outlet pipe; and finally, the negative pressure remains stable. Figure 5(c) shows the internal velocity distribution of the atomizer. The initial state of air flow velocity is 50 m/s and the water flow velocity is 0 m/s; then, the air velocity increases at the orifice and flows through the entire outlet pipe. The velocity at the center axis of the outlet pipe is the greatest, while the velocity at the pipe wall is the smallest. When the water is absorbed into the outlet pipe, it first breaks into droplets along with the air vortex at the pipe wall, and then moves to the outlet pipe gradually near the center axis. Therefore, the diameter of atomized particles for this dual-fluid atomizer is smaller than for a general atomizer.

5.3. Influence of the orifice size on water velocity at the inlet

The size of the orifice diameter has a relatively large influence on the water velocity of the inlet. There would be too low a negative pressure, or even no negative pressure, if the orifice area was too large, while the negative pressure would be too low in the wall of the outlet pipe if the orifice area was too small. Both of these scenarios would lead to poor water absorption. Therefore, a suitable
Table 3. Different outlet length conditions.

| Case | l_out (mm) | D0 (mm) | d1 (mm) | d2 (mm) | V_air (m/s) | φ |
|------|------------|---------|---------|---------|-------------|---|
| 3    | 20.5       | 13.8    | 21      | 5       | 50          | 0.1|
| 4    | 20.5       | 12      | 21      | 5       | 50          | 0.1|
| 5    | 20.5       | 17      | 21      | 5       | 50          | 0.1|

Figure 8. Effect of outlet pipe length on flow velocity at the water inlet.

Figure 9. Internal streamline distribution of the atomizer under different outlet length (l_out) conditions.

orifice diameter is needed to maximize the water velocity of the inlet. To study the influence of orifice size on water velocity at the inlet, several simulations were carried out with different orifice diameters for cases 1 and 2 in Table 2 (Yao et al., 2017). The diameters of the orifices which maximize the water velocity of the inlets for cases 1 and 2 are 17 mm and 9 mm, respectively, as shown in Figure 6.

Figure 7 shows the internal streamline distribution of the atomizer under different orifice diameters. As can be seen in Figure 7(a), air will generate a vortex current after flowing through the orifice. The larger the vortex area, the longer the droplet moves in the outlet pipe, and the easier it is to break into smaller droplets. This leads to better secondary atomization. Comparative analysis of the streamlines in the atomizer is also given for orifice diameters of 10, 15 and 20 mm. It can be seen from Figure 7 that there is a small vortex when the diameter of the orifice diameter is 10 mm. The vortex gradually becomes larger as the diameter of the orifice increases. The recirculation flow zone with the 15 mm orifice diameter is larger than that for the 10 mm orifice diameter, but the vortex zone almost disappears when the orifice diameter is increased to 20 mm (Figure 7a–(c). Thus, it is concluded that there is a suitable orifice diameter that can maximize the secondary atomization effect.

5.4. Influence of outlet pipe length on water velocity at the inlet

To study the influence of the outlet pipe length on water velocity at the inlet, nine simulations were carried out
with different outlet pipe lengths for cases 3 and 4 (from Yao et al., 2017) and case 5, as listed in Table 3.

Figure 8 shows the variation tendency of the water velocity at the water inlet under the different outlet pipe lengths. Because the vortex current zone will increase as the air passes through the orifice, extension of the outlet pipe length will lead to the increment of the negative pressure and the water velocity of the water inlet. In addition, with the diameter of the orifice of 13.8 mm, a comparative analysis of the streamline distribution is given with the length of the outlet pipe being 10, 15 and 20 mm. It can be easily seen from Figure 9(a)–(c) that the vortex area for the 20 mm outlet pipe is relatively large, while the vortex areas are reduced when the outlet pipe is 10 or 15 mm. Kushari (2010) points that droplet size would be decreased with the increase in the length of the mixing chamber. Therefore, it is a significant conclusion that the larger the vortex region, the smaller the droplet size, and the better the secondary atomization effect.

6. Conclusions

The transient gas–liquid two-phase flows inside an atomizer were numerically simulated by solving the 3D viscous Navier–Stokes equations considering the SST k-ε turbulence and the VOF multiphase flow models. In this study, the formation of the vortex, the automatic absorption process of the water and the atomization process of air–water inside the atomizer were successfully reproduced. The effects of different air flow velocity and atomizer geometry parameters (e.g. different orifice diameters and outlet pipe lengths) on the water absorption capacity of the atomizer were also studied and revealed. However, the influence of outlet pipe diameter on atomization performance was not studied. The conclusion, that the larger the vortex region, the smaller the droplet size, could be verified by the DPM model. The main conclusions can be summarized as follows:

(1) The water absorption improves as the air flow velocity increases.

(2) A suitable orifice diameter will maximize the water flow and allow the most effective secondary atomization. When \( l_{in} = 20.5 \, \text{mm} \), \( l_{out} = 20.5 \, \text{mm} \), \( d_1 = 21 \, \text{mm} \), \( d_2 = 5 \, \text{mm} \) and \( \phi = 0.1 \), the orifice diameter \( D_0 = 17 \, \text{mm} \) maximizes the water flow velocity and there is an optimal orifice diameter which will make the secondary atomization best.

(3) The water flow velocity increases and the secondary atomization improves as the length of the outlet pipe increases.

(4) The larger the vortex region, the smaller the droplet size.

Disclosure statement

No potential conflict of interest was reported by the authors.

Funding

This study was supported by the National Natural Science Foundation of China [grant number 11705143], the Natural Science Basic Research Plan in Shaanxi Province of China [grant number 2018JQ1016] and the China Postdoctoral Science Foundation [grant number 2018M643814XB].

List of symbols

- \( D_0 \) orifice diameter [mm]
- \( d_1 \) diameter of the mixing chamber [mm]
- \( d_2 \) diameter of the water inlet [mm]
- \( l_{in} \) length of outlet part [mm]
- \( l_{out} \) length of outlet part [mm]
- \( Q \) volume flow rate \([m^3 \cdot s^{-1}]\)
- \( V \) velocity \([m/s]\)
- \( \phi \) porosity
- \( F_1, F_2 \) blending functions
- \( k \approx 0.41 \) von Karman constant
- \( \beta, \gamma \) coefficients in the \( k-\omega \) model
- \( \beta^* = C_{\mu} \)
- \( P_k \) production of turbulent kinetic energy \([m^2/s^3]\)
- \( \delta \) boundary layer thickness [m]
- \( \delta_{ij} \) Kronecker delta (1 if \( i = j \), 0 otherwise)
- \( \mu \) dynamic molecular viscosity \([kg/(m \cdot s)]\)
- \( \mu_t \) dynamic eddy viscosity \([kg/(m \cdot s)]\)
- \( \nu = \mu/\rho \) kinematic molecular viscosity \([m^2/s]\)
- \( \rho \) density \([kg/m^3]\)
- \( \sigma_k \) Prandtl number in \( k \)-equation diffusion term
- \( \sigma_\omega \) Prandtl number in \( \omega \)-equation diffusion term
- \( \sigma_{\omega 2} = 0.856 \)
- \( \tau \) timescale [s] or shear stress [Pa]
- \( \omega \) turbulence inverse timescale [s\(^{-1}\)]

References

Chin, J. S., & Lefebvre, A. H. (1995). A design procedure for effervescent atomizers. Journal of Engineering for Gas Turbines and Power, 117(2), 266–271. doi:10.1115/93-GT-337

El-Shanaway, M. S., & Lefebvre, A. H. (1980). Airblast atomization: The effect of linear scale on mean drop size. Journal of Energy, 4(4), 184–189. doi:10.1115/80-GT-74

Ferreira, G., Garcia, J. A., Barreras, F., Lozano, A., & Lincheta, E. (2009). Design optimization of twin-fluid atomizers with
an internal mixing chamber for heavy fuel oils. Fuel Processing Technology, 90(2), 270–278. doi:10.1016/j.fuproc.2008.09.013

Firmansyah, D. A., Kim, S. G., Lee, K. S., Zahaf, R., Yong, H., & Lee, D. (2012). Microstructure-controlled aerosol-gel synthesis of ZnO quantum dots dispersed in SiO2 nanospheres. Langmuir the ACS Journal of Surfaces & Colloids, 28(5), 2890–2896. doi:10.1021/la203730a

Fritsching, U. (2004). Spray simulation. Cambridge: Cambridge University Press.

Goldberg, U. C., & Batten, P. (2015). A wall-distance-free version of the SST turbulence model. Engineering Applications of Computational Fluid Mechanics, 9(1), 33–40. doi:10.1080/19942060.2015.1004791

Kushari, A. (2010). Effect of injector geometry on the performance of an internally mixed liquid atomizer. Fuel Processing Technology, 91(11), 1650–1654. doi:10.1016/j.fuproc.2010.06.014

Lefebvre, A. H. (1989). Atomization and sprays. New York, NY: Hemisphere Publishing Corporation.

Liang, X. Y. (2005). Experimental research and numerical simulation on effervescent atomizer. Nanjing, China: Southeast University.

Liang, X. Y., & Wang, X. L. (2008). Numerical simulation on the inner and outlet downriver of effervescent atomizers nozzle. Boiler Manufacturing, 5, 13–16. doi:10.1016/j.ssa.2014.05.090

Liu, Y., Li, Z., Zhang, G. Q., Xu, W. Y., Yuan, H., Liu, N., & Zheng, L. (2015). Flow field of double layer atomizer. Journal of Aeronautical Materials, 35(5), 63–69.

Menter, F. R. (1993). Zonal two equation k-ω turbulence models for aerodynamic flows. NASA STI/Recon Technical Report N, 93. doi:10.2514/6.1993-2906

Mohan, B., Yang, W., & Chou, S. (2014). Cavitation in injector nozzle holes – a parametric study. Engineering Applications of Computational Fluid Mechanics, 8(1), 70–81. doi:10.1080/19942060.2014.11015498

Nguyen, D. A., & Rhodes, M. J. (1998). Producing fine drops of water by twin-fluid atomisation. Powder Technology, 99(3), 285–292. doi:10.1016/S0032-5910(98)00125-9

Shafaee, M., Banitabaei, S. A., Esfahanian, V., & Ashjaee, M. (2011). An investigation on effect of geometrical parameters on spray cone angle and droplet size distribution of a two-fluid atomizer. Journal of Mechanical Science & Technology, 25(12), 3047–3052. doi:10.1007/s12206-011-0901-2

Thompson, J. S., Hassan, O., Rolland, S. A., & Sienz, J. (2016). The identification of an accurate simulation approach to predict the effect of operational parameters on the particle size distribution (PSD) of powders produced by an industrial close-coupled gas atomiser. Powder Technology, 291, 75–85.

Tian, C., & Lu, Y. (2013). Turbulence models of separated flow in shock wave thrust vector nozzle. Engineering Applications of Computational Fluid Mechanics, 7(2), 182–192. doi:10.1080/19942060.2013.11015463

Wright, J. D., Bean, V. E., & Aguilera, J. (2008). NIST calibrations services for hydrometers. Gaithersburg, MD: National Institute of Standards and Technology.

Xia, M., Wang, P., Zhang, X. H., & Ge, C. C. (2018). Computational fluid dynamic investigation of the primary and secondary atomization of the free-fall atomizer in electrode induction melting gas atomization process. Acta Physica Sinica, 67(17), 1–12. doi:10.7498/aps.67.20180584

Yao, J. F., Yu, M. H., Zhao, T., Kawahara, A., & Sadatomi, M. (2017). Investigation of CO2 absorption performance in a gas-liquid two-phase flow atomizer on the basis of a gas diffusion model. Transactions of the Canadian Society for Mechanical Engineering, 41(4), 645–656. doi:10.1139/tcsme-2017-1045

Yu, Y., Shademan, M., Barron, R. M., & Balachandar, R. (2012). CFD study of effects of geometry variations on flow in a nozzle. Engineering Applications of Computational Fluid Mechanics, 6(3), 412–425. doi:10.1080/19942060.2012.11015432

Yuan, Y. J., Tan, L. B., Xu, Y. Y., & Yuan, Y. D. (2015). Numerical simulation of gas-liquid two-phase flow in effervescent atomizer. Journal of Shaanxi University of Science & Technology, 33(5), 135–140.

Zeoli, N., Tabbara, H., & Gu, S. (2011). CFD modeling of primary breakup during metal powder atomization. Chemical Engineering Science, 66(24), 6498–6504. doi:10.1016/j.ces.2011.09.014

Zhang, L., Wu, W. S., & Pu, G. (2006). An experimental study on spray performances of twin fluid nozzle for flue-gas decontamination. Environmental Engineering, 24(2), 40–43. doi:10.1016/S1872-2040(06)60041-8