Numerical Simulation on Deposition of Atomization Droplet of Air-assist Boom Spraying in Air Flow

Wei Zhong¹, Huitao Zhou¹, Weidong Jia¹* and Fei Xue²
¹Key Laboratory of Modern Agricultural Equipment and Technology, Ministry of Education, Jiangsu University, Zhenjiang 212013, China
²Agricultural machinery technology extension station, Yangzhou
Email:2211816039@stmail.ujs.edu.cn

Abstract. Air-assist Boom Spraying is a crucial technique and spray method of plant protection which has a significant effect on improving the efficiency of pesticide application and reducing droplet drift. Spraying drift is the main reason for the low utilization rate of pesticides, at the same time, the interaction of natural wind and the air bag wind is the main cause of spray drift. But at present, the regularity of spray droplet deposition and motion trajectory under the common influence of natural wind and air bag wind is not yet clear. The kinematics characteristics of the droplets deposition of air-assist boom spraying in the plants is investigated with numerical simulation method. The Phase Doppler Particle Analyzer (PDPA) is used to validate the numerical simulation results and it indicates that the deposition process can be well reproduced. The results show that the bigger droplets (>150 μm) get more kinetic energy from the vertical airflow and locate in windward side. These droplets are not sensitive to the horizontal airflow and are easier to deposit within a short time. The droplets with smaller size (<150 μm) are minimally influenced by the vertical airflow and suspend in the upside of the flow field. The vertical airflow not only prevents the horizontal airflow drift the droplets, but also introduces a vortex region near the canopy of the plants, which rotates the plant canopy and increases the droplet deposition on backside of the leaves. This study improves the understanding of the droplets deposition under the multi-wind fields and provides the theoretical basis for the innovation of the air-assist boom spraying system.

1. Introduction
Plant protection, a vital means to prevent crop diseases, insects and weeds, is indispensable in agricultural production process. Air-assist boom spraying is an important technique of plant protection, which improves the efficiency of pesticide application and reduces spray droplets drift substantially.

Domestic and foreign scholars have launched a wide range of research on the air-assist boom spraying system. Panneton et al. (1996) investigated the effects of different airflow velocities, airflows and air jet angles on the adhesion of droplets on the leaf[1]. Ringel and Anderson studied the impacts of the air release angle, air velocity on the drift and deposition via using Hardi Twin sprayer on the winter wheat[2]. Brown (1995) tested and evaluated the characteristics of airflow-assisted sprayers on three different situations in wind tunnel: non-windy, vertical downward airflow, and forward 30° auxiliary airflow[3]. For determining the optimal operating parameters, Philion et al. (1995) tested airflow-assisted sprayers in the spray chamber.

In recent years, CFD simulation have been applied in the research of spray flow field characteristics of plant protection machinery. Due to the limiting factor of experiment and other conditions, it is difficult to determine the velocity vector distribution of the air flow field, trajectory of the droplet and so on through the experiments. So using CFD to complete mathematical modeling and
simulation based on fluid mechanics theory has been an important auxiliary method in spray flow field research, that can provide theoretical guiding in studies. In 1992, Reichard et al. (1992) simulated trajectory of droplet in wind tunnel by Fluent software [4-6]. Tsay et al. (2004) used Fluent software to simulate the anti-drift effect of ordinary spray, air-assist shield spray and wind-curtain spray under different conditions, and optimized the operating parameters of the air-assist shield spray[7]. Tsay and Ozkan et al. (2002) investigated the characteristics of air-assist shield spray through two-dimensional and three-dimensional CFD, simulated and compared the functions of anti-drift in six sprayers[8]. Weiner and Parkin (1993) used Fluent to establish three-dimensional model of the spray flow field, the calculation results revised by boundary condition and measured data, were consistent with measured results[9]. Sidahmed and Brown (2001) simulated flow field, droplet trajectory and sedimentary characteristics of the forestry blast sprayer in three dimensions and two dimensions by using Fluent, for verifying the correctness of the Fluent 3D free jet model. Above research provide a strong support for the reasonable application of numerical simulation in air-assist boom spraying field.

Research and optimizations on gas-liquid two-phase flow of the air-assist boom spraying could provide fundamental support for spray system optimization design, and improve the spray quality more economically and rapidly. Dynamic simulation of the gas-liquid two-phase flow of the air-assist boom spraying is investigated by Ansys Fluent software, simultaneously, Discrete Phase Model (DPM) and RNG k-ε turbulence model were used to get the internal flow field of air-assist boom spraying and the distribution of droplets in the flow field at different times. The droplets sizes and distributions of velocities were measured by PDPA. The reliability of the simulation was verified by experiments, could provide theoretical basis and basic parameters for multi-mode optimizations of air-assist boom spraying.

2. Numerical Method

2.1. Governing Equation

Evaporation and heat transfer during the movement were neglected, assuming the gas-liquid two-phase flow as incompressible Newton fluid, meanwhile, there was no phase transition on the gas-liquid interface in the simulation, and the droplets followed the conservations of mass, momentum and energy in the process of motion, must declaring that the mass transfer and heat transfer were not considered in isothermal process.

Mass conservation equation:

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \mathbf{U}) = 0$$ (1)

Where: \( t \) is the time; \( \nabla \cdot \) express divergence; \( \mathbf{U} \) is the velocity vector; \( u, v, w \) are components of the velocity vector \( \mathbf{U} \) in \( x, y, z \) directions; And \( \rho \) is density of fluid. For incompressible steady flow, the divergence of velocity is zero.

$$\nabla \cdot \mathbf{U} = \frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} + \frac{\partial w}{\partial z} = 0$$ (2)

Momentum conservation equations for incompressible viscous fluids:

$$\frac{\partial u}{\partial t} + \nabla \cdot (u \mathbf{U}) = \nabla \cdot \left( \frac{1}{\rho} \frac{\partial p}{\partial x} \right) + \nabla \cdot (v \mathbf{u})$$ (3)

$$\frac{\partial v}{\partial t} + \nabla \cdot (v \mathbf{U}) = \nabla \cdot \left( \frac{1}{\rho} \frac{\partial p}{\partial y} \right) + \nabla \cdot (v \mathbf{v})$$ (4)

$$\frac{\partial w}{\partial t} + \nabla \cdot (w \mathbf{U}) = \nabla \cdot \left( \frac{1}{\rho} \frac{\partial p}{\partial z} \right) + \nabla \cdot (v \mathbf{w})$$ (5)

Where: \( p \) is pressure; \( v \) is the kinematic viscosity; \( \nabla \cdot \mathbf{u} = \frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} + \frac{\partial w}{\partial z} \), and \( S_u, S_v, S_w \) are generalized source items, \( S_u=F_1/p, S_v=F_2/p, S_w=F_3/p \).

This paper still used the Reynolds-Averaged Navier-Stokes equation(RANS), because the average flow field of time scale is able to reflect physical natures of gas flow from the view of spray
application engineering application. The equation was obtained by averaging the Navier-Stokes equations as follow:

\[
\frac{\partial u_i}{\partial t} + u_j \frac{\partial u_i}{\partial x_j} = -1 \frac{\partial p}{\partial x_i} + \frac{1}{\rho} \frac{\partial}{\partial x_j} \left( \nu \frac{\partial u_i}{\partial x_j} - u_i u_j \bar{u}^2 \right)
\]  
(6)

Where: \( \mu' \) is the velocity fluctuation.

Neglecting the bulk force in the incompressible steady flow, the equation above could be written as following form:

\[
u \frac{\partial u_i}{\partial x_j} = -1 \frac{\partial p}{\partial x_i} + \frac{1}{\rho} \frac{\partial}{\partial x_j} \left( \nu \frac{\partial u_i}{\partial x_j} - u_i u_j \bar{u}^2 \right)
\]  
(7)

2.2. Continuous Phase Setting

Considering air as the continuous phase in the air-liquid two-phase flow field of air-assist boom spraying, assuming that the value of the assist-air inlet velocity of air bag system was constant. In this study, gas was considered as the continuous phase in the air-liquid two-phase flow field of air-assist boom spraying, assuming that the size of the inlet air velocity was stable, and the natural wind in horizontally had no other directions of the component.

The control measures of continuous phases include continuity equation, momentum equation and turbulence model, and SIMPLEC was used in velocity-pressure coupling. The turbulence energy and diffusion rate of Reynolds-stress were calculated using the RNG model of the standard k-ε model based on the Boussinesq hypothesis, where the value of the model empirical constants as follows: \( C_m=0.045 \), \( C_{1}\epsilon=1.42 \), \( C_{2}\epsilon=1.68 \).

2.3. Discrete Phase Setting

Droplets were the discrete phase in the air-liquid two-phase flow field of air-assist boom spraying, which were assumed as spherical. The DPM (Discrete Phase Models) method was used to describe and track the movement of droplets in the air flow field, and droplet trajectories were obtained by solving the momentum equilibrium equation of droplets based on Lagrangian discrete phase model. Using the Flat-Fan Atomizer model as the atomization model. Taylor analogy crushing model (TAB) was used to analyze the droplet broken in the spray process.

\[
\begin{align*}
\frac{d^2 y}{dt^2} &= \frac{C_b \rho_l u_l y^2}{C_s \rho_l u_l} \frac{C_k \sigma}{\rho_l u_l^2} \frac{C_m u_l dy}{\rho_l u_l^2 dt} \\
\end{align*}
\]  
(8)

Where: \( \rho_l, \rho_g \) are the density of liquid phase and gas phase, respectively; \( u \) is the relative-velocity of the liquid phase; \( r \) is the radius of the droplet before deformation; \( \sigma \) is the surface tension of the liquid phase, \( \mu_l \) is the viscosity of the liquid phase, \( C_b \) is a constant (usually equals to 0.5); The dimensionless constants \( C_F, C_k, C_d \) are derived from experimental data and theoretical derivation, if breakup is assumed to occur when the distortion is equal to half the droplet radius, that is, oscillations at the north and south pole with this amplitude will meet at the droplet center, where breakup now occurs for \( y>1 \).

Dynamic drag force model equation:

\[
F_D = \frac{18 \mu}{\rho_d u_p^2} \frac{C_D Re}{24}
\]  
(9)

Where: \( u \) is the velocity of continuous phase; \( u_p \) is the velocity of discrete phase; \( \mu \) is the dynamic viscosity of continuous phase; \( \rho \) is the density of continuous phase; \( \rho_p \) is the density of discrete phase; \( d_p \) is the size of the discrete phase particles; \( Re \), the Reynolds number of the discrete phase particles, was represented as:

\[
Re = \frac{\rho d_p |u_p-u|}{\mu}
\]  
(10)

The equation used to determine the drag coefficient \( CD \) was:
Where: for spherical particles, \(a_1, a_2, a_3\) are constants when the Reynolds number were in a certain range.

In order to distinguish between wipe-passing and merging of droplets, introduced the critical number \(\chi_{cr}\): when the distance \(\chi \leq \chi_{cr}\), the droplets merged after collision; when \(\chi > \chi_{cr}\), the droplets passed after collision. The expression of collision and merger of droplets was \([10-11]\):

\[
\chi_{cr}^2 = (r_1 + r_2)^2 \min[1, 0.24f(\gamma)/We] \tag{12}
\]

Where: \(\rho T_{rel}/\sigma(T_d)\), \(\sigma(T_d)\) is the surface tension of droplets which is related to the average temperature \(T_d\) of the droplets; \(T_d = (r_1^3 T_1 + r_2^3 T_2)/(r_1^3 + r_2^3)\). \(T_1\) and \(T_2\) are the temperatures of the droplets; \(f(\gamma) = \gamma^3 - 2.4\gamma^2 + 2.7\gamma, \gamma = \gamma_2/\gamma_1 \geq 1\). \(f(\gamma)\) is the function of radius ratio \(\gamma\). Basing on the law of conservation of momentum, the velocities of the droplets after collision and passing could write as follow:

\[
\begin{align*}
    &u_{1\text{new}} = \\
    &\frac{u_1 m_1 + u_2 m_2 + m_2 (u_1 - u_2) + \chi_{cr}}{m_1 + m_2} \tag{13}
\end{align*}
\]

The velocities of the droplets merging after collision was \(u_{\text{new}} = (u_1 m_1 + u_2 m_2)/(m_1 + m_2)\), where \(m_1\) and \(m_2\) are the mass of small droplets and large droplets, \(u_1\) and \(u_2\) are velocities of small droplets and large droplets, and the combined droplet size is \(r_{\text{new}} = (r_1^3 + r_2^3)^{1/3}\).

The mutual transmission of quality and momentum had been going on between the gas-liquid two phases during the process of gas-liquid two-phase flow (regardless the energy transfer between the phases because the spraying operations were carried out in room temperature condition). Therefore, it is necessary to consider the coupling between the liquid phase (the discrete phase) and the gas phase (continuous phase) in the calculation, the control equations of the continuous phase and the discrete phase were calculated alternately until the solution of the two phases converged.

The values of momentum and mass transmitted from the continuous phase to the discrete particles were solved by calculating the momentum and mass change of the particles, when the whole particle along with the momentum and mass carried by particle entered and moved out of the control body \([12-14]\). The change in particle momentum was:

\[
F = \sum \left[ \frac{18\mu C_D Re}{\rho p d_p^{24}} (u_p - u) + F_{\text{other}} \right] \dot{m}_p \Delta t \tag{14}
\]

The change in particle mass was:

\[
M = \frac{\Delta m_p}{\dot{m}_p,0} \dot{m}_p,0 \tag{15}
\]

Where: \(\mu\) is the viscosity of continuous phase fluid, \(\rho_p\) is the density of discrete phase particles, \(d_p\) is the size of particles, \(Re\) is the relative Reynolds number of particles, \(u_p\) is the velocity of particles, \(u\) is the velocity of the continuous phase fluid, \(C_D\) is the drag coefficient, \(\dot{m}_p\) is the mass flow rate of the particles, \(\Delta t\) is the time step, \(F_{\text{other}}\) is the other force between two phases.

### 2.4. Computational Model and Boundary Conditions

#### 2.4.1 Computational model. The size of the continuous flow field which was selected to simulate calculation area is length \(\times\) width \(\times\) height = 8 \(\times\) 2.8 \(\times\) 1.3 m, the simulation calculation area is shown in Figure 1. An area of 2.8 m long and 0.3 meters wide in the interior of the flow field was provided as the air bag outlet, four operating nozzles were set in the horizontal direction of the air bag outlet according to the experiment requirements, the nozzle spacing 0.5 m. Arranging two rows of plants under the flow field according to the data processing capacity of computer, the width of rows was 0.8 m. There were 8 trees which was 0.5 m high in each row, the width between trees was 0.5 m.
For each plant, there were 3 layers of leaves, every 0.2 m arranged two leaves from the height of 0.4 m. The area of each leaf was 0.3 m$^2$, the leaves spreading width of each layer was 0.3 m (Figure 2), and the vertical distance between the top of the plants and the air bag outlets was 0.5 m.

2.4.2 Boundary condition. The front interface of the calculation area (the natural wind inlet ADLI) and air bag outlet were set as Velocity-Inlet, the wind speed in the air bag outlet was 10.74 m/s when the natural wind speed was 5 m/s and the speed of the fan was 1947 r/min. Regarding the side (ABJI, CDLK), top (IJKL) and back (BCKJ) of flow field as the outlets of calculation area, and setting the boundary condition as Pressure-Outlet, the total pressure at the pressure outlet was 0 MPa, and the middle surface (EFGH) which was the interface of the relative movement, was set as the interface. The running velocity of the spray boom was 5 m/s.

The locations of four Lechler 110-05 standard fan-shaped nozzles were selected as the outlets of the droplet discharging, and adopting the Flat-Fan Atomizer model as the atomization model. The width and length of the nozzle were 0.05 m and 0.1 m, the flow rate was 0.033 kg/s when the spray pressure and the atomization angle was 0.3 MPa and 102.8 °. Supposing the number of droplets sprayed by each nozzle was 3000, and droplets were linear distribution.

Since most of the components of the liquid in the actual field spray operation are water, it is false in the simulation study. The droplets are all pure water droplets. In the entire simulated flow field, the droplets are finally only terminated by the species, which are: Deposition, drift and evaporation. Because the amount of evaporation generated by the droplets during the deposition process is only a small proportion, Therefore, in this study, evaporation during the droplet deposition process is not considered, and it is assumed that the droplets move to each wall.

Ignoring the evaporation in the process of droplet deposition and the rebound phenomena of droplets moving to the walls, and assuming that the droplets were pure water droplets, the ground and the surface of plants leaves were set as “trap” in the simulated area. When the droplet moved to this interface, the trajectory calculation was terminated, and supposed that the droplets were deposited this area. At the "Escape" boundary condition, which was the side (ABJI, CDLK), the top (IJKL), and the back (BCKJ) of flow field, the droplets would be considered to be drifting and then the calculation was terminated.

Thinking the plant leaves surface and the ground as sampling surface, the time of sampling was 0.19 s, where each step lasted 0.01 s, totally 1900 steps. These droplets which didn’t deposit on the sampling surface of the calculation area after 0.19 s were drift droplets. After the end of sampling, the sampling surfaces under different working conditions were analyzed, and the accumulative values of droplet deposition from the beginning to the end of the simulation for about 1900 steps were counted.
as the deposition amount of droplets in each working condition.

2.5. Grid Divisions
Establishing the calculation area by using Ansys ICEM software based on the simulation area of spray process is shown in Fig 1. And then dividing the flow field of calculation area to two parts according to the order of line-face-body. The upper part which was divided into 214970 units using the regular hexahedral mesh was the area including the air bag outlets, and the lower part of the target plant area, was divided into 968851 units by using the positive tetrahedral mesh. The mesh results of the actual simulation area are shown in Figure 3.

![Figure 3. The model mesh](image)

3. Experimental Setup
The air-assist boom spraying test system is shown in Figure 4. The system included three parts: the system of air bag operation and adjustment, boom spray operation and regulation system and PDPA test system, laboratory temperature of 23 °C and relative humidity of 71%.

![Figure 4. Schematic diagram of air-assist boom spraying test system](image)

1. Axial flow fan; 2. Frequency convertor; 3. Air-curtain; 4. Spray boom; 5. Nozzle; 6. Pressure sensor; 7. Flow control valve; 8. Pump; 9. tank; 10. Compute; 11. PDPA;

4. Result
4.1. Validation of the Numerical Method
In order to verify the rationality of the calculation model, the part of the data of the numerical simulation results and the experimental results of the PDPA experiment of the gas-liquid two-phase flow field were compared. Comparisons between the calculation results of the average size of droplets, the average velocity of the vertical direction of the droplet in the measuring line position and PDPA test results are shown in Figure 5 and Figure 6. From the two figures, it can be seen that the calculation results of the droplets and experimental results are basically the same, and the droplet size data of simulated spraying are less than the experimental data. This phenomenon mainly dues to the theoretical model which used in the software simulation of the spray process was an empirical model.
under ideal, but the actual experiments would be effected by certain external environment, and also there will be some other impurities in the natural air. Therefore the average particle size of the droplets will decrease with these factors. In general, the influences on droplet deposition and drift properties can be negligible because the difference in the average droplet size is less than 10 μm. These values and their variation trend of calculation model are consistent with the experimental test. Thus the calculation model established in this paper can achieve the results consistent with the actual situation. So it can be considered that the computational model can achieve a more accurate simulation of the spray process of the air-assist boom spraying to a certain extent and scope.

Figure 5. The simulation and experimental results of droplet diameter

Figure 6. The simulation and experimental results of droplet vertical (Z direction) velocity

4.2. Spray Flow Field

Velocity distribution of the spray simulation of the flow field in 0.01 seconds is shown in Figure 7, and selecting profile of the calculation area which was located on the center of the nozzle at 250 mm from the center of the boom as a viewing surface. It can be seen from the figure that a high-speed flow field was generated at the outlet of the curtain, and the flow velocity in the downward direction of the flow field was large, which could weaken the movement of the droplets to the target crop, shorten the movement time of the droplets in the air, and reduce the loss of liquid caused by droplet evaporation. Thereby high-speed flow field can reduce the droplet drift potential, and have a significant effect of preventing the droplet drift.
Figure 7. Continuous phase velocity distribution

The distribution of droplets in the flow field during simulation is shown in Figure 8. The color of the figure represents the particle size of the droplets. As can be seen from the figure, the particle size distribution of the droplet was hierarchical under the action of natural wind. And the particle size of droplets in the wind surface closest to the natural wind was the largest, the closer the droplet is to the upper flow field, the smaller the size is. So considering that the whole droplets were moving along the wind direction under natural wind. However, the larger droplets are deposited earlier because of their own mass, smaller droplets which are susceptible to natural winds are floating over the upper part of the flow field and difficult to deposit.

(a) The droplet size distribution when t = 0.05 s.  (b) The droplet size distribution when t = 1 s.
(c) The droplet size distribution when t = 1.9 s.  (d) The droplet size distribution when t = 2 s.

Figure 8. Particle traces colored by particle diameter

Figure 9 shows the continuous phase velocity vector of the air bag outlet. The velocity of the upper flow field mainly determined by natural winds was about 5 m/s. The airflow from the air curtain airbag constitutes a wind curtain, covered the upper part of the droplet group. The droplets located in the upper part of the air flow field were forced to move downward the droplets because of the downward pressure generated by wind curtain, so that it is possible to suppress the drift of the partial droplets. The speed in the lower part of the flow near the plant was small, and natural wind was blocked by plants. A high velocity flow field was generated in the outlet of the air bag, and the wind speed vector pointed to the lower right direction. There was a vortex area of airflow at the left bottom of the air bag. The direction of the wind curtain is vertical down, the direction of the natural wind from
left to right, which the vector direction is the resultant vector of the air-bag’s wind and the natural wind.

Figure 9. Continuous phase velocity vector vector diagram at air bag outlets

Velocity vector of discrete phase droplet is shown in Figure 10, the droplets has a certain speed when away from the nozzle. And due to the role of gravity and inertia, most of the droplets move downward. As the distance of the droplet away the nozzle increases, the velocity of the droplet decreases gradually due to the resistance of the air. The movement speed in horizontal direction generated the droplets due to the influence of the natural winds. With the droplets rises, the velocity of these droplets increases. When the droplets reached a certain height, the velocity of the droplets was same with the velocity of natural wind. At this time the droplets are completely controlled by nature wind and then move without falling, which means drift.

Figure 10. Particle traces colored by velocity magnitude

5. Conclusion
Dynamic simulation of the gas-liquid two-phase flow of the air-assist boom spraying is investigated by Ansys Fluent software, and simultaneously, Discrete Phase Model (DPM), RNG k-ε turbulence model are used to get internal flow field of air-assist boom spraying and the distribution of droplets in the flow field at different times. The simulation results are compared with the results measured by PDPA, and it is concluded that the calculation model can simulate the spraying process of the air-assist boom spraying accurately. The results shows that:

(1) The resistance of the air decreases the downward velocity of the droplets. And the downward velocity of these droplets away from the nozzle was 0 m/s, which completely controlled by the natural wind and along the wind direction and no longer decline, namely drift.

(2) The particle size distribution is hierarchical in the process of falling droplets. The droplets on the windward side with larger particle sizes are deposited earlier because of their own mass, and the smaller droplets near the upper part of the flow field which are susceptible to natural winds are
floating over the upper part of the flow field and difficult to deposit.

(3) The direction of airflow field with high speed generated by the airflow from air curtain airbag is determined by the vector sum of the air curtain flow direction and the natural wind direction. The airflow field with high speed mentioned above constitutes a wind curtain, and covered the upper part of the droplet group. The droplets located in the upper part of the air flow field were forced to move downward because of the downward pressure generated by wind curtain, so that it is possible to suppress the drift of the partial droplets.

6. Acknowledgements
This research was financially supported by the Ningxia Key R&D Program (Grant No. 2018BBF02020), the authors also want to express their thanks to those graduate students for their wonderful work.

7. References
[1] Panneton, B., H. Philion and R. Theriault. Spray chamber evaluation of air-assisted spraying [J]. ASAE Paper, 1996, No. 96-1078.
[2] May, M. J. Early studies on spray drift, deposition manipulation and weed control in sugarbeet with two air assisted boom sprayers [J]. In air assisted Spraying in Crop Protecting-BCPC Monograph, 1991, No. 46.
[3] Brown, R.B. General principles of air-assisted sprayers[C]//Proceedings of National Conference on Pesticide Application Technology. University of Guelph, Guelph, Ontario, Canada. 1995,97-102.
[4] Philion, H., B. Panneton, P. Dutilleul, et al. Statistical comparison of a full factorial design and a central composite design for spray droplet deposition[J]. CSAE Paper, 1995, No. 95-406.
[5] Reichard D L., Zhu H., Fox R D , et al. Wind Tunnel Evaluation of a Computer Program to Model Spray Drift[J]. Transactions of the Asae, 1992, 35(3):755-758.
[6] Reichard D L., Zhu H., Fox R D , et al. Computer Simulation of Variables that Influence Spray Drift[J]. Transactions of the Asae, 1992, 35(5):1401-1407.
[7] J.Tsay, L.Liang, L.H.Lu. Air-assisted boom spraying system under a no-canopy condition using cfd simulation [J]. Transactions of the ASAE, 2004, 47(6):1887~1897.
[8] J.Tsay, R.D.Fox, H.E.Ozkan, et al. Evaluation of a pneumatic-shielded spraying system by cfd simulation [J]. Transactions of the ASAE, 2002, 45(1):47~54.
[9] Weiner K L., Parkin C S. The use of computational fluid dynamic code for modeling spray from a mistblower [J]. Journal of Agricultural Engineering Research, 1993, 55(4): 313~324.
[10] Sidahmed M M, Brown R B. Simulation of spray dispersal and deposition from a forestry airblast sprayer-part 1: air jet model [J]. Transactions of ASAE, 2001, 44(1): 5~10.
[11] R. D. Reitz. Mechanisms of Atomization Processes in High-Pressure Vaporizing Sprays [J]. Atomization and Spray Technology, 1987, 3:309-337.
[12] G. I. Taylor. The Shape and Acceleration of a Drop in a High Speed Air Stream [M]. In G.K. Batchelor, editor. The scientific papers of Sir Geoffrey Ingram Taylor, vol. 3, Cambridge University Press, 1963:457-464.
[13] O’Rourke P. J. Collective drop effects on vaporizing liquid sprays [D]. Mechanical and aerospace engineering. Princeton University, USA, 1981, Ph.D. Thesis.
[14] Panneton B , Philion H , Roger Theriault, et al. Spray Chamber Evaluation of Air-Assisted Spraying on Broccoli[J]. Crop ence, 2000, 40(2):444-448.