Computers and Electronics in Agriculture xxx (2017) xxx–xxx

Contents lists available at ScienceDirect

Computers and Electronics in Agriculture

journal homepage: www.elsevier.com/locate/compag

Original papers

CFD simulation of airflow inside tree canopies discharged from air-assisted sprayers

Se-Woon Hong [a], Lingying Zhao [a,*], Heping Zhu [b]

[a] Department of Food, Agricultural and Biological Engineering, The Ohio State University, 59 Woody Hayes Drive, Columbus, OH 43210, USA
[b] USDA-ARS Application Technology Research Unit, 1680 Madison Avenue, Wooster, OH 44691, USA

ARTICLE INFO

Article history:
Received 9 December 2016
Received in revised form 31 March 2017
Accepted 11 July 2017
Available online xxxx

Keywords:
Air-assisted sprayer
Airflow
Computational fluid dynamics
Porous modeling
Tree structure

ABSTRACT

Effective pesticide application is not only essential for specialty crop industries but also very important for addressing increasing concerns about environmental contamination caused by pesticide spray drift. Numerical analysis using computational fluid dynamics (CFD) can contribute to better understanding of the transport of spray droplets carried by strong air jets from sprayers. In this research, an integrated CFD model was developed to predict air velocity distributions inside and around tree canopies blown by an air-assisted pesticide sprayer. The sprayer motion was simulated by the sliding mesh technique, and the tree canopies were defined in the computational domain as virtual porous media without their geometric modeling. Validation of the CFD model was accomplished in three steps by comparing the CFD results with previous measurements. Air velocities and airflow pressures downwind from the sprayer agreed well with the measurements when the sprayer was both stationary and in motion. The model was also able to predict accurately the peak air velocity and airflow pressure inside the canopies with average RMS errors of 1.68 m s^{-1} and 0.89 kg m^{-2}, and relative errors of 29.2% and 20.2%, respectively. Although discrepancy existed between the field experiment measurement and CFD simulation this study can conclude that the simulation will give a reasonable prediction of air distributions discharged from air-assisted sprayers. The validated CFD model was applied to predict air distribution inside canopies with various canopy dimensions and densities. The complex airflow patterns obtained by the CFD model offered advantages of analyzing effects of various factors on sprayer performance.

1. Introduction

In specialty crop industries, high-quality crops must be produced to satisfy consumers' demands. Application of pesticides is essential for mass production of high-quality crops. The excessive use of pesticide and off-target spray deposition has raised concerns about dietary risks from pesticides and contamination of soil, surface water, and ground water. Improving safety and environmental friendliness of pesticide application is necessary for sustainability in high-value crop industries (Zhu et al., 2006b).

In most pesticide spray applications with air-assisted sprayers, air-jet streams carry spray droplets released from arrays of nozzles. Air jets play key roles in transporting spray droplets deeply into canopies by blowing droplets to and beyond the target (Endalew et al., 2010a). The strong air jets increase deposition by reducing flight time and reducing the influence of weather conditions on droplets (Weiner and Parkin, 1993). The airflow patterns from these sprayers are complex and three-dimensional interactions between the strong air jets and target crops. Insufficient knowledge about these airflow patterns leads to poor prediction of droplet movements and misapplication of chemicals. In order to improve the efficacy of air-assisted spraying and reduce off-target drift, it is essential to understand sprayer-induced airflow patterns in and around the target canopy as well as at locations far from the target.

The airflow patterns produced by air jets have been studied by experiments and modeling. Early studies stressed that field experiments are limited because wind cannot be controlled during a single spray event (Zhu et al., 1995). Measurements were taken to investigate the amount of drift rather than understand the aerodynamics of moving droplets (Tsay et al., 2002). In recent years, numerical analysis using computational fluid dynamics (CFD) has contributed to better understanding of transport of spray droplets. The three-dimensional airflow patterns from sprayers are determined by spraying injection, wind, and plant structure (Farooq and Salyani, 2004; Delele et al., 2005; Baetens et al., 2007; Endalew et al., 2010b). Computer simulations have benefited the

* Corresponding author.
E-mail address: zhao.119@osu.edu (L. Zhao).

http://dx.doi.org/10.1016/j.compag.2017.07.011
0168-1699/

prediction of long distance droplet drift while full-scale experiments are usually expensive and difficult to perform because of variability in meteorological conditions (Endalew et al., 2010a).

In previous CFD studies, the geometrical features of sprayers and tractors were not included in the CFD model due to their complicated shapes. In the case that the tractor or sprayer was moving during spraying, a pulse function was used to replace actual movement of the tractor or sprayer (Delele et al., 2005; Endalew et al., 2010b). A pulse function might not satisfy a mass continuity for air jets without well-structured meshes at the injector and accurate interpolation over the meshes. Furthermore, plants were not effectively modeled in most previous CFD works. Endalew et al. (2010a, 2010b) included the effect of trees into their CFD models. The geometric properties of full-scale pear trees, including all the branches, were modeled as a 3D object. Porous sub-domains were also added around the branches to simulate very thin branches, flowers, and leaves. In several other studies, plants were modeled as a porous media with cuboidal or spherical geometries (Mercer, 2009). While the former studies took advantage of more computational time and cost to offer a realistic representation of tree canopies, the latter might oversimplify the canopy for the sake of economy in the numerical calculations. Neither study achieved both realism and economy.

An accurate CFD representation of air-blast sprayers requires modeling the complex airflow patterns resulting from the interaction of the spray air jet and various types of plant canopies. The objective of this research was to develop an integrated CFD model to simulate spray jet performance of a moving air-assisted sprayer inside tree canopies. The developed CFD model will be applied to predict air velocity distribution through canopies of specialty crops with various dimensions, densities, and shapes. Eventually, the realistic and economic model will be helpful in understanding the effect of tree canopy structure on spray jet performance.

2. Materials and method

2.1. An air-assisted sprayer

A variable-rate air-assisted sprayer system was developed to control the spray discharges from 20 nozzles independently according to canopy density and distribution (Chen et al., 2012, 2013). Spray fluid is discharged from four specially-designed five-port-nozzle manifolds on both sides of the sprayer as shown in Fig. 1. The sprayer was designed for its spray pattern to cover target canopies of up to 3.2 m in height at 1.5 m distance from the sprayer. The heights of the four nozzle manifolds were 0.85, 1.35, 1.85, and 2.35 m above the ground. The manifolds were cast with five nozzle ports, each at a radial angle of 15° to improve spray penetration into dense nursery crops by dividing a conventional large air jet into five small jets (Zhu et al., 2006a). The inside diameter of the injection nozzle was 36 mm.

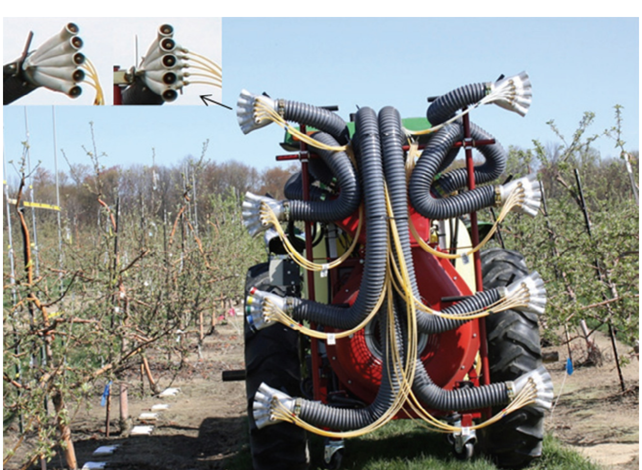

Fig. 1. An intelligent sprayer with four five-nozzle manifolds on both side of the sprayer (Chen et al., 2013).

2.2. Development of an integrated CFD model

CFD models were developed with a commercial CFD program Fluent 15.0 (ANSYS, Inc., PA, U.S.). The geometry and computational domain were generated with the ICEM CFD 14.5 (ANSYS, Inc., PA, U.S.). The CFD model using the sliding mesh technique for sprayer motion and virtual porous media for tree canopies was validated by previous experimental data for a variable-rate air-assisted sprayer system. Validation was carried out in terms of air jet velocities from the sprayer in an open field and air velocities in and around tree canopies (Gu et al., 2011, 2012, 2014).

2.2.1. Governing equations and models

In this study, the fluid phase was solved by the Reynolds-averaged Navier-Stokes (RANS) equations. The Reynolds stresses were simplified by two turbulence quantities, k and ω. The SST k-ω model was used for the turbulence modeling. The SST k-ω model has been increasingly used in the numerical simulation of flow around bluff bodies (Yang et al., 2009). The SST k-ω model has also provided more accurate and reliable predictions for a wide class of flow compared with widely-used k-ε models (Fluent, 2014). Delele et al. (2004) found the SST k-ω model gave relatively better accuracy for airflow prediction from a stationary air-assisted sprayer compared with the k-ε models.

A sprayer in motion was modeled by the sliding mesh technique that described flow situations involving interaction between stationary and moving parts. The stationary and moving parts represented the fluid phase and the sprayer, respectively, in this study. Two parts were in contact, and the contact surfaces between the two parts, the interface boundary, was continuously changing according to the location of the moving part. While the sprayer part moved relatively to the stationary fluid phase, the non-conformal interface algorithm (Fluent, 2014) computed fluxes across the interface boundaries satisfying a mass continuity through the interface.

The pressure based solver was used to solve incompressible flows, and the SIMPLE algorithm was used for steady and unsteady simulations to calculate pressure-velocity coupling. The pressure was discretized by the Second-order scheme. Momentum and turbulent quantities were discretized by the Second-order Upwind scheme. The convergence criteria of residuals for all variables were set as 1.0×10^{-4}.

2.2.2. Canopy architecture

Tree canopies were modeled as a porous media as most similar studies have done (Mercer, 2009). The trees were simulated using a sphere for leaves and branches and a cylinder for the stem. The classical modeling approach has strived to generate unstructured meshes consisting of such rounded geometries or, sometimes, realistic tree branches. However, it may be questioned whether it is worth the time and effort to make so-called 'perfect' meshes for irregular tree structures. Alternatively, this study modeled the tree canopies by setting porosity at structured cells where imaginary trees are assumed to be located, as shown in Fig. 2. This method avoids any complex processes for geometric modeling and meshing of actual trees but enables modeling of a number of trees or changing the shape of trees without repeating pre-processing for mesh generation.

Please cite this article in press as: Hong, S.-W., et al. CFD simulation of airflow inside tree canopies discharged from air-assisted sprayers. Comput. Electron. Agric. (2017), http://dx.doi.org/10.1016/j.compag.2017.07.011