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

Contents lists available at ScienceDirect

Computers and Electronics in Agriculture

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

Original papers

Comparative study of CFD models of the air flow produced by an air-assisted sprayer adapted to the crop geometry

Jorge Badules[a], Mariano Vidal[a], Antonio Boné[a], Jordi Llop[b], Ramón Salcedo[b], Emilio Gil[b], F. Javier García-Ramos[a,*]

[a] Escuela Politécnica Superior - Huesca, Universidad de Zaragoza, Spain
[b] Escuela Superior de Agricultura de Barcelona, Universidad Politécnica de Cataluña, Spain

ARTICLE INFO

Keywords:
Modelling
Vineyard
Mesh
Boundary conditions
Crop treatment

ABSTRACT

A computational fluid dynamics (CFD) model for simulating the air flow produced by an air-assisted sprayer can be developed according to any one of several sets of criteria and for various specific applications. Thus far, most CFD models have focused on the characterization of the air flow generated by the sprayer. However, such models may not be the best when assessing the effectiveness of air-assisted sprayers adapted to crop geometry, such as those used in vineyards. In this study, the air flow produced by an air-assisted sprayer adapted to the geometry of a vineyard was simulated using four CFD models: Model 1 was designed to simulate air velocity measurements at the outlets; Model 2 was designed to simulate air velocity measurements at a certain distance from the outlets; Model 3 was dedicated to the modelling of the internal geometry of the air ducts rather than the characterization of the air flows generated by the sprayer; and Model 4, which was developed as a variant of Model 3, was created to perform the same calculation in several stages. The models were validated with actual measurements of the air velocity near the sprayer outlets. The results showed that although Models 1 and 2 (both of which have been used in most existing studies) simplified the calculation, they are impractical for simulating different air flows. By contrast, Models 3 and 4 provided complex meshes that complicate the convergence of the calculation and require a suitable treatment of both the viscosity and the flow near the walls. Model 3 showed the smallest error (16%) in the air velocity estimated in the treatment plane. Model 4 showed potential for future implementation of the dispersed phase and crop-air interaction because it mitigated the problems arising from the complexity of the meshes. It should also be noted that there are errors inherent to the implementation of the CFD model, errors related to inaccuracies in the geometry of the air ducts, inaccuracies related to the measurement of the air velocities, inaccuracies in the quantification of the air flow, and simplifications linked to the turbulent model.

1. Introduction

Computational fluid dynamics (CFD) facilitates the study of numerous transport phenomena associated with fluids. One example is the modelling of the pesticide application process that is performed using agricultural sprayers (Bartzanas et al., 2013), which facilitates the estimation of drift loss (Da Silva et al., 2006; Baetens et al., 2007, 2009; Nuyttens et al., 2011), thereby improving the prediction accuracy of the efficacy of the treatment. Consequently, it is possible to reduce both the treatment cost and the resulting environmental pollution (Svensson, 2001; Ako, 2011; DEFRA, 2001). Field trials, which have been carried out for decades, are tedious and expensive; moreover, they are not repeatable because of the influence of atmospheric conditions, varying crop characteristics, etc. By contrast, CFD models are advantageous, as they can estimate the characteristics of the treatments carried out with the sprayers under a wide range of real-world conditions (Teske et al., 2011).

In recent years, several researchers have developed CFD models to simulate the performance of the air-assisted sprayers used in fruit orchards (Delele et al., 2005; Endalew et al., 2010a, 2010b; Dekeyser et al., 2013; Salcedo et al., 2013). These studies have shown that the air flow generated by certain air-assisted sprayers can be simulated successfully with this technique. With regard to a specific sprayer, a validated CFD model may be a faster and more economical tool than an analysis conducted using collected experimental data (DEFRA, 2001), especially when further investigating the influence of that various parameters, such as air flow rate, forward velocity, and the phenological state of the treated crop (Da Silva et al., 2002).

The objective of this study is to analyse the advantages and disadvantages of different strategies for developing CFD models to

---

* Corresponding author at: Escuela Politécnica Superior de Huesca, 22071 Huesca, Spain.
E-mail address: fjavier@unizar.es (F.J. García-Ramos).

http://dx.doi.org/10.1016/j.compag.2017.09.026
Received 25 November 2016; Received in revised form 14 September 2017; Accepted 20 September 2017

Please cite this article as: Badules, J., Computers and Electronics in Agriculture (2017), http://dx.doi.org/10.1016/j.compag.2017.09.026

Nomenclature

ρ	density
$\vec{u}$	vector of velocity
p	pressure at a point
μ	dynamic viscosity
k	turbulent kinetic energy
ε	energy dissipation ratio
E	error
V_m	value measured at a point
V_c	value calculated by CFD simulation
N	number of measured points

estimate the air flow generated by an air-assisted sprayer adapted to a specific crop geometry. Toward this end, a commercial sprayer that can simultaneously spray two rows of a vineyard from both sides was employed. The design of the equipment (Fig. 1) generates complex and highly vortex air currents, all of which can complicate not only the computational modelling but also experimental data collection.

For this study's purpose, four different CFD models were compared. Model 1 was designed to measure air velocity as close as possible to the machine outlets (Dekeyser et al., 2013). Model 2 was designed to measure air velocity along a line at a certain distance from the air outlets (Endalew et al., 2010b; Salcedo, 2015). Model 3 was dedicated to implementing the internal geometry of the air ducts and the total air flow supplied by the fan without the need to characterize air velocity at the machine outlets (as a prerequisite for developing the CFD model). Model 4 was designed to fulfill a purpose similar to that of Model 3, but it performs the essential calculation in two stages, the solution of the first stage being the input boundary condition of the second stage: stage 1 consists of the geometry of the duct, while stage 2 consists of the adjacent-air component the model.

2. Materials and methods

2.1. Sprayer characteristics

The spraying device analysed in this study was an IRIS multi-row air-assisted sprayer (Ilemo-Hardi, S.A.U., Lleida, Spain) adapted to the line geometry of a vineyard (Fig. 1). The sprayer consisted of a centrifugal turbine that directed the air flow to four vertical ducts parallel to the sides of the vineyard line. Each vertical duct housed four groups of air-fluid outlets. Each group consisted of two hydraulic nozzles and one air outlet with internal deflectors that could be oriented in a one of several directions. Each air outlet was divided into four individual outlets each having dimensions of 3 × 5 cm.

A hedge of the vineyard was enclosed between a pair of vertical ducts that stood opposite to each other and were separated by approximately 1.65 m such that the crop could be treated on both sides at the same time. Thus, the machine, which was designed specifically for spraying vineyard lines, could treat two rows of vines simultaneously.

In order to take empirical measurements, one of the vertical ducts was dismounted and moved to a laboratory. The duct was connected to a centrifugal fan, model S & P CRRT/2-401 RD 275 5.5 kW (Soler & Palau, Barcelona, Spain), which was able to provide an air flow similar to that provided by the sprayer. The laboratory temperature was about 25 °C, with no appreciable changes in temperature occurring between the air inlet of the fan and the air outlet of the duct.

The air flow produced by the fan was measured according to ISO 9898:2000 (García-Ramos et al., 2015) using a TESTO 0635 1041 hot-wire anemometer (Testo AG, Lenzkirch, Germany) having an accuracy of 0.03 m/s and a measurement range of 0–20 m/s. Based on all the measurements, an average flow rate of 1010 m^3/h was obtained.

2.2. Fundamentals of CFD models

ANSYS Fluent 15.0 (ANSYS, Inc., Canonsburg, PA, USA) was used as the CFD software for the resolution of all the models. The basis of CFD is the resolution of the Navier-Stokes equations, which describe the movement of Newtonian fluids. Basically, they include the momentum and continuity equations. Eq. (1) represents the momentum equation in vector notation, while Eq. (2) represents the continuity principle.

$$\rho\frac{D\vec{u}}{Dt} = \rho\left\{\frac{\partial\vec{u}}{\partial t} + (\vec{u}\cdot\nabla)\vec{u}\right\} = -\nabla p + \mu\cdot\nabla^{2}\vec{u} \tag{1}$$

$$\nabla\vec{u} = 0 \tag{2}$$

Considering turbulent flows (those with a high Reynolds number, as in the case of sprayers), vortices of different sizes are generated (Salcedo, 2015), and these transfer energy (from the largest to the smallest) without loss by a process called energy cascading. However, upon reaching a certain scale, the fluid viscosity begins to control the phenomenon, converting the turbulent energy into heat. Ideally, the mesh of the model in the regions of viscous dissipation should be less than half of the scale at which this phenomenon begins to occur. However, this scale is so small that a resolving of Eqs. (1) and (2) would require an irresolvable mesh using conventional computers (Versteeg and Malalasekera, 1995). To solve this problem, several CFD techniques have been developed, each using mesh cell sizes that are much larger than the viscous dissipation scale. The most commonly use of these techniques at present relies on the RANS model (Reynolds Average

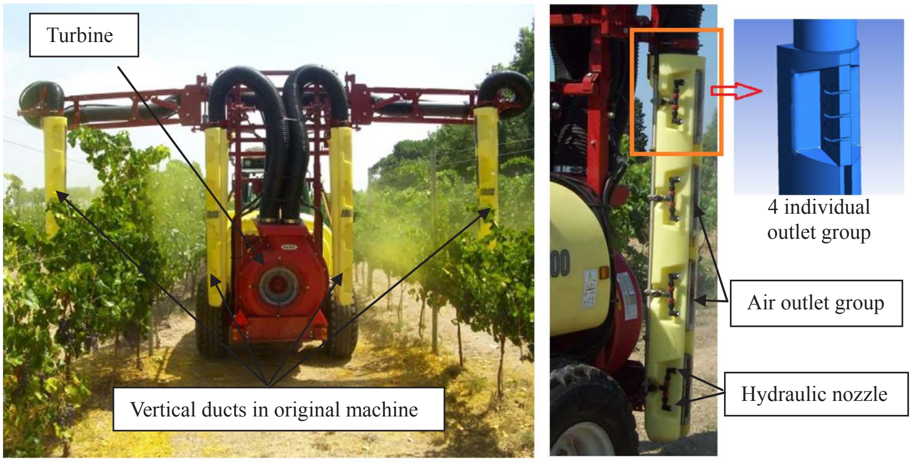

Fig. 1. IRIS multi-row air-assisted sprayer (Ilemo-Hardi, S.A.U., Lleida, Spain) adapted to the vineyard geometry. The centrifugal turbine directs the air flow to four vertical ducts (left), each having four air-liquid outlet groups, parallel to the sides of the vineyard line. In each air-liquid outlet group, there are two hydraulic nozzles and four air outlets (right).