Computers and Electronics in Agriculture 93 (2013) 151–167

Contents lists available at SciVerse ScienceDirect

Computers and Electronics in Agriculture

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

Computational fluid dynamics applications to improve crop production systems

T. Bartzanas [a,*], M. Kacira [b], H. Zhu [c], S. Karmakar [d], E. Tamimi [b], N. Katsoulas [e], In Bok Lee [f], C. Kittas [a,e]

[a] Center for Research and Technology, Thessaly, Institute of Technology and Management of Agricultural Ecosystems, Technology Park of Thessaly, 1st Industrial Area of Volos, PO Box 15, 38500 Volos, Greece
[b] Agricultural and Biosystems Engineering, The University of Arizona, Tucson, AZ 85721, USA
[c] USDA-ARS, Application Technology Research Unit, 1680 Madison Ave., Wooster, OH 44691, USA
[d] Faculty of Agricultural Engineering, Bidhan Chandra Krishi Viswavidyalaya, Mohanpur, Nadia 741 252, West Bengal, India
[e] University of Thessaly, Sch. of Agricultural Sciences, Department of Agriculture, Crop Production & Rural Environment, Fytokou St., N. Ionia, GR-38446 Volos, Greece
[f] Aero-Environmental and Energy Engineering Laboratory, Department of Rural Systems Engineering and Research Institute for Agriculture and Life Sciences, College of Agriculture and Life Sciences, Seoul National University, 599, Gwanakno, Gwanakgu, 151-921 Seoul, Republic of Korea

ARTICLE INFO

Article history:
Received 2 November 2011
Received in revised form 19 March 2012
Accepted 12 May 2012

Keywords:
Tillage
Harvesting machines
Sprayers
Greenhouse
Decision support tools

ABSTRACT

Computational fluid dynamics (CFD), numerical analysis and simulation tools of fluid flow processes have emerged from the development stage and become nowadays a robust design tool. It is widely used to study various transport phenomena which involve fluid flow, heat and mass transfer, providing detailed information for spatial and temporal distributions of flow speed and direction, pressure, temperature and species concentration. The CFD tools provide a cost-effective way of carrying out equipment and process design and optimization, and can reduce risk in equipment modification and process scale-up. In recent years, CFD modeling has been gaining attraction from the agri-food industry. The present paper provides a state-of-the-art review on various CFD applications to improve crop farming systems such as, soil tillage, sprayers, harvesting, machinery, and greenhouses. The challenges faced by modelers using CFD in precision crop production are discussed and possibilities for incorporating the CFD models in decision support tools for Precision Farming are highlighted.

1. Introduction

Historically agriculture has covered the expansion of human population by boosting its productivity. In the last century most of this increased productivity was the result of genetic improvement of crops and use of chemicals and machinery. In the next years agriculture is expected not only to produce food and raw material but also to maintain the landscape and contribute to the reduction of Greenhouse Gases (GHGs) in the atmosphere. It is anticipated that the new increase in productivity in the next 50 years can also be the result of Information and Communication Technology (ICT) use in agriculture.

The introduction of advanced ICT in agriculture has enabled farmers to acquire huge amounts of site-specific data for their farms, with the ultimate aim to improve their decision-making process (Blackmore, 2000). Precision Agriculture, Site-Specific Management, Precision Farming are all synonymous of innovative agricultural techniques to improve production and reduce environmental pollution. Precision Agriculture (PA) can be defined as "the application of technologies and principles to manage spatial (in-field) and temporal (over time) variability associated with all aspects of agricultural production for the purpose of improving crop performance and environmental quality" (Blackmore et al., 2003). The first applications of PA started in the early nineties but the initial adoption started at the end of nineties.

The PA is intrinsically information intensive, and farmers face many difficulties in efficiently managing the enormous amount of data they collect. They may lack sufficient time or are reluctant to invest the time needed to analyze the data and interpret the information. Additionally, the economic benefits of PA practices have not been proven yet, especially applications such as variable rate of nitrogen application and patch spraying (Swinton and Lowenberg-DeBoer, 1998; Pedersen, 2003). The challenge is to identify the usefulness, importance and relevance of the gathered data for optimizing farm efficiency. It became clear that it is not a lack of available data that will impede progress in PA. Farmers have to think systematically about their information needs, the costs of information, alternative sources and the value of the information, identifying what is the necessary data to collect before making decisions (Fountas et al., 2006). There is no "cook-book" available on how to systematically analyze the gathered data to help them make farm management decisions. This has become a drawback on the wide adoption of PA worldwide. It becomes more evident that the analysis and interpretation of PA data will be a

* Corresponding author. Tel.: +30 24210 96740; fax: +30 24210 96750.
E-mail address: bartzanas@cereteth.gr (T. Bartzanas).

0168-1699/$ - see front matter © 2012 Elsevier B.V. All rights reserved.
http://dx.doi.org/10.1016/j.compag.2012.05.012

new role for agronomists, as separate service or as complimentary to agrochemical and fertilizer suppliers.

Numerical modeling techniques such as CFD can offer an effective way of accurately quantifying the influence of machinery design, environment parameters and weather conditions within a virtual environment. Thus, the amount of physical experimentation can be reduced considerably, although, as of yet, not eliminated. CFD is a simulation method that can efficiently estimate both spatial and temporal field fluid pressure as well as other chemical and environmental scalars, and the method has proven its effectiveness in system design and optimization within the chemical, aerospace, and hydrodynamic industries (Zhang et al., 2006). The ubiquitous nature of fluids and their influence on system performance has caused a widespread take-up of CFD by many other disciplines. As a developing modeling technique, CFD has received extensive attention throughout the international research and industrial community. As a result, CFD became an integral part of the engineering design and analysis environment of many companies because of its ability to predict the performance of new designs or processes prior to manufacturing or implementation (Schaldach et al., 2000).

The application of CFD in precision crop production has been concentrated mainly on greenhouse systems and the optimization of sprayers. Few works have been done to optimize the design and functioning of harvesting machines and some to analyze soil characteristics during tillage.

The present work reviews the main issues in the context of the current status and capabilities of CFD in precision crop production and identifies its potential for contributing in the design of more efficient decision support systems for Precision Agriculture applications.

2. Applications of CFD in precision crop production

2.1. Tillage

Tool interaction with agricultural soil during tillage deals with soil cutting, inversion, pulverization, compaction and traction. Tillage is associated with large soil deformation and soil translocation. Studies of soil-tool interaction in tillage focus investigation on energy requirement and soil failure pattern. Pressure exerted by soil on the tillage tool that is associated to draft requirement and its distribution with respect to tool wear, is an important parameter in determining tool size and shape. Numerical methods have been preferred over analytical and empirical methods in recent past toward optimizing design parameters in tillage mechanics. Even though last few decades, computational methods have been applied to study tillage mechanics, some unanswered aspects remain mainly due to the complexity of tillage dynamics. Fluid flow approach using computational fluid dynamics (CFD) modeling has been found to have great relevance in analyzing and predicting large soil deformation in tillage.

2.1.1. Soil dynamics in tillage

Though tillage is a dynamic process, most of the previous studies are based on quasi-static analysis following the passive earth pressure theories. During tillage, soil particles move ahead and around the tool as they fail in shear. As the tool engages soil during tillage, undisturbed stiff soil sustains the exerted tool thrust up to its elastic limit before failing in shear. Soil shear failure can be visualized and analyzed as a flow pattern in front and around the cutting tool. Soil failure front, the maximum longitudinal distance of the disturbed soil from the tool face, is associated with slip surfaces generated by yielding and plastic deformation. The advancement of the soil failure front depends on the operating speed, tool shape and size, tool orientation, and the soil conditions.

The assumptions involved with most of the analytical and numerical models based on earth pressure theory neglect the inertial forces and are suitable only for predicting the forces on a narrow tine moving at very slow speed. Moreover, due to the limitations of the constitutive relationships, the dynamic nature of tillage has not been represented in a realistic way with the widely studied finite element method (FEM) approach (Yong and Hanna, 1977; Liu and Hou, 1985; Chi and Kushwaha, 1990; Wang and Gee-Clough, 1991). Conservation tillage necessitates high speed of operation for energy efficiency and optimization of soil disturbance. It was suggested that other numerical tools should be combined with FEM to enable evaluation of overall behaviors of tillage implements (Plouffe et al., 1999).

2.1.2. CFD in tillage mechanics

Traditional stress–strain relationships of agricultural soil deformation that rely on the empirical geotechnical engineering practices fail to address key features of soil structural dynamics (Ghezzehei and Or, 2001). The engineering soil mechanics approach is based on equilibrium state stress–strain relationships for the study of soil deformation, while deformations in agricultural soils rarely reach equilibrium (Or, 1996). Soil shear rate with respect to the tool operational speed plays a very important role in analyzing and optimizing high speed tillage. Computational fluid dynamics was applied to the flow of sludge in a manure spreader (Thirion, 2002). Davison et al. (2002) studied soil flow over augers using a CFD code.

Analytical methods have their limitations to address the problem with complex tool geometry, dynamics of the tool and soil failure pattern. There have been studies on the dynamic analysis of the soil-tool interface using numerical methods like FEM, discrete element method (DEM) and artificial neural networking (ANN). Though these numerical methods have been able to solve some of the complex system parameters, dynamic features related

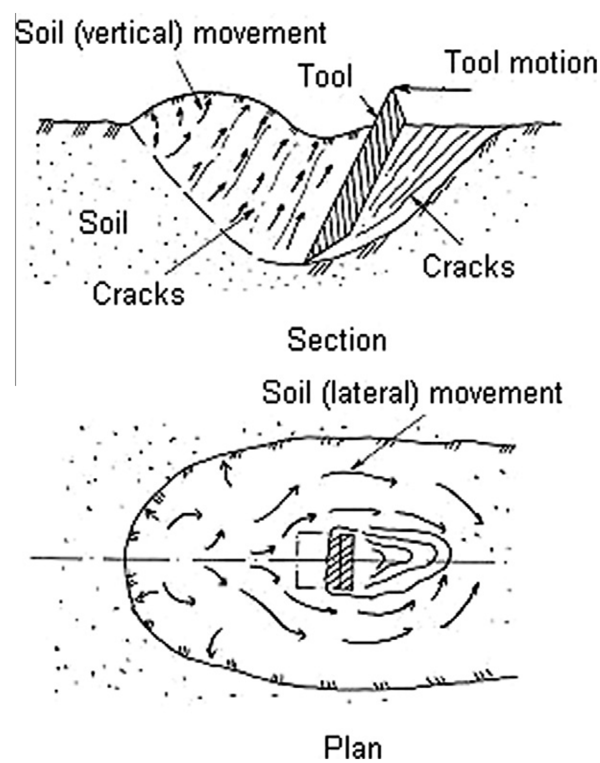

Fig. 1. Soil tool idealization (Desai and Phan, 1980).

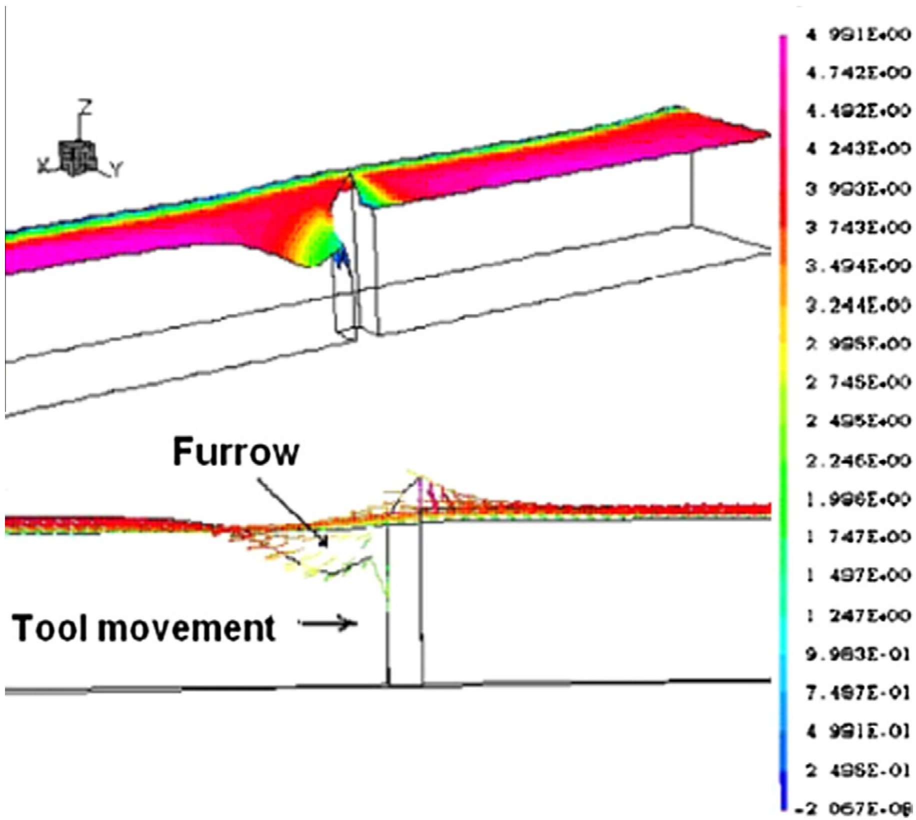

Fig. 2. Velocity profile showing furrow formation behind the tool and soil build-up in front of tillage tool (Karmakar, 2005).

to the soil failure front and soil deformation patterns have not been addressed to the level of satisfaction.

Soil is highly non-linear and needs characterization as nonlinear plastic or visco-plastic material (Desai and Phan, 1980). The interparticulate contact zones within the soil mass can be viscous in nature leading to a non-linear rate dependent response (Keedwell, 1984). Soil deformation under steady state stress can be described by a simple linear model of viscoplasticity, the Bingham rheological model (Vyalov, 1986). Study of soil-tool interface mechanism in connection with soil rheology from fluid flow perspective will lead to a better understanding of tillage dynamics.

Desai and Phan (1980) speculated and suggested a general case of the three-dimensional soil tool interaction in which the tool is moving relative to the soil as shown in Fig. 1. Thus the soil shear failure due to the translation of the tool is analogous to the fluid flow over a blunt body. The velocity vectors of the soil particles as they encounter with the tool and soil failure front propagation can be derived from a fluid mechanistic approach.

2.1.3. CFD modeling of tillage tools

Study in tillage tools modeling mainly focus on energy requirements to operate a tool at different soil, tool and system conditions. Soil pressure exerted on tool surface was also simulated using CFD (Karmakar, 2005). Mathematical modeling was based on Navier stokes relationships for incompressible, single phase soil flow in a free-surface flow channel with specific boundary conditions. Using CFD the soil was treated as a visco-plastic fluid using the Bingham-plastic material model. Dynamic analysis of soil-tool interaction with CFD helped understand visco-plastic soil flow phenomena and formation of furrow and soil build-up around the tool (Fig. 2). Free-surface simulation improved the prediction and description of the dynamics of soil-tool interaction and enabled visualization of soil surface deformation representing furrow formation at the back of the tool and ridge formation in front of the tool (Karmakar and Kushwaha, 2005).

Pressure bulbs in front and around the tool, depicting the soil stress due to the tool motion is shown in Fig. 3 (Karmakar et al., 2007). The contour lines describe the range of pressure on the horizontal plane at halfway below the soil surface i.e., at 50 mm depth. The normal stress on the soil in front of the tool at 50 mm depth and $1\ m\ s^{-1}$ operating speed decreased from 177 to 20 kPa along the direction of motion forming a set of pressure bulbs. A negative pressure zone existed behind the tool due to flow suction and pressure drop. Pressure distribution on the tool surface (50 mm $\times$ 100 mm) shown as color[1] fringe plots indicates that the maximum pressure zone (area of pressure concentration) lies near the tool tip (Figs. 4 and 5).

CFD analysis of dynamic soil-tool interaction revealed many interesting features of viscoplastic soil flow and the soil deformation pattern including pressure pattern on the tool surface towards modeling energy requirement. The flow dynamics near the tool in the flow domain is of major interest with respect to soil failure front. Soil disturbance zone due to the tool interaction was obtained from this axial velocity profile of the soil particles (Karmakar and Kushwaha, 2005).

Computational fluid dynamics modeling indicated dynamic viscoplastic soil deformation patterns with distinct plug and plastic flow regions (Karmakar and Kushwaha, 2007). Plastic soil failure pattern was nicely visible due to the formation of pressure bulbs. Pressure bulbs were the boundary or contours of the soil failure lines or slip lines. Soil failure front associated with tool speed was also obtained from the longitudinal velocity patterns and yield surfaces. Pressure concentration on the tool surface was found at

[1] For interpretation of color in Figs. 2–4, 6–11, 14–16, 19 and 20 the reader is referred to the web version of this article.

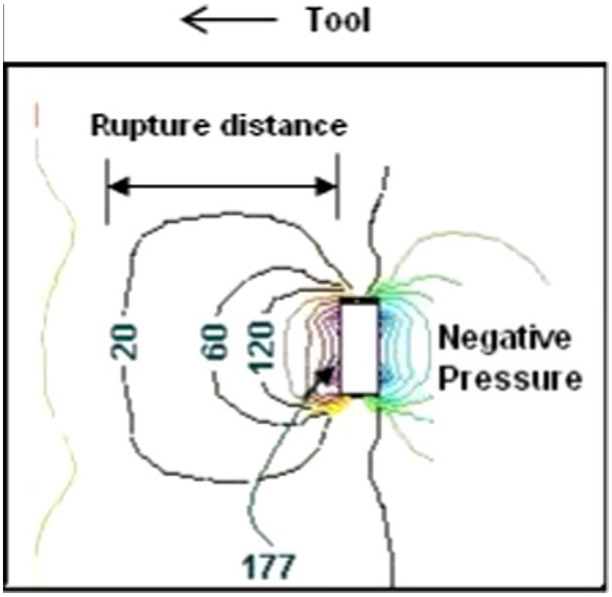

Fig. 3. Pressure (kPa) contours on the horizontal plane at 1 m s^{-1} tool speed (Karmakar et al., 2007).

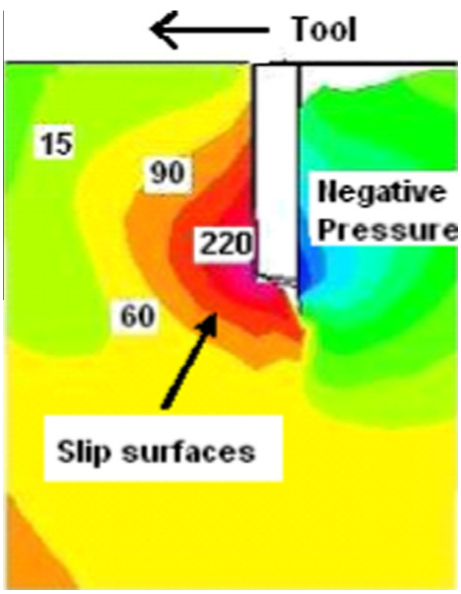

Fig. 4. Pressure (kPa) fringe plot on the vertical plane at 1 m s^{-1} tool speed (Karmakar et al., 2007).

the bottom. The critical speed range could also be found using this approach that would help optimize the tool operating speed for creating desired soil deformation for preparing seedbed.

Steady-state CFD only and transient two-way fluid-structure interaction simulations were run to predict draft and vertical forces on a spring-reset field cultivator standard (Barker, 2008). The results indicate that CFD method can be used to reasonably predict the draft force to within three standard deviations of the average measured draft. The method was successfully used to determine tool shape for optimizing draft requirement.

2.1.4. Validation of CFD modeling

Experimental validation of CFD modeling was done for a 40 mm wide narrow tillage tool operating at four different depths of 40, 80, 120, and 160 mm at four different operating speeds of 1, 8, 16 and 24 km h^{-1} in clay loam soil (Karmakar et al., 2009) Simulation results over-predicted tool draft in comparison to the experimental values. The difference between the predicted and measured draft were not consistent and ranged from 1% to 42%. The agreement of simulation data with experimental values was higher at shallow depth of operation and lower tool operating speed. The correlation coefficient between the simulation and experimental draft were found to vary from 0.9275 to 0.9914.

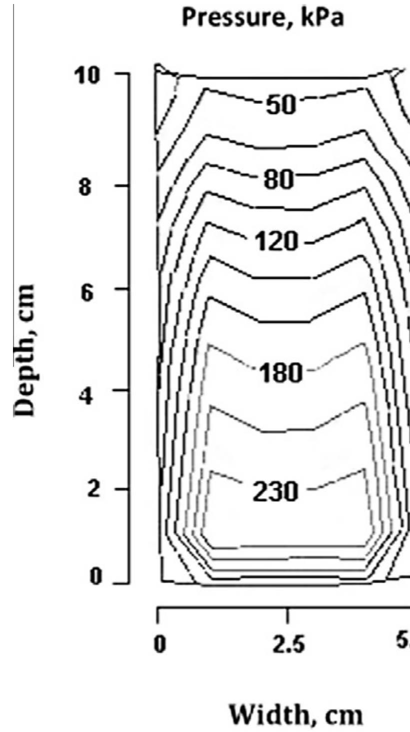

Fig. 5. Pressure distribution on the tool surface at 1 m s^{-1} tool speed (Karmakar, 2005).

Soil pressure on the tool surface and the resulting draft predicted by CFD modeling were very close when compared to published data based on finite element analysis on similar studies (Karmakar et al., 2007). CFD modeling of tillage indicated that soil pressure on the tool surface, which was considered as direct parameter in calculating draft requirement, increased with speed (Karmakar et al., 2007).

2.2. Sprayers

Despite ever more stringent market requirements, spray applications of pesticides and other pest or disease control strategies have ensured a continuous production of high quality crops. However, improper applications of pesticides can cause excessive or inadequate pesticide to be similarly delivered to target and non-target areas, resulting in greater production cost, worker exposure, environmental contamination, and adverse impacts on vulnerable ecosystems. Proficient applications of pesticides often are encumbered by use of various equipment and methods. Many factors influence the efficiency and effectiveness of sprayer performance to control pests. Among them are the droplet size and density, distance from atomizer to target, droplet velocity, direction of discharge from atomizer, volatility of spray fluid, wind velocity and direction, relative humidity, ambient temperature, atmospheric turbulence intensity, crop canopy structure, and crop planting pattern. Many field tests have been conducted to study the influence of these variables on sprayer performance. However, the uncontrollable limitations of weather conditions and crop structures and the influence of variables on spray performance may interact and vary during field tests. A seldom considered deficiency is our inability to integrate these uncontrollable variables into a decisive strategy.

Most computer simulations for pesticide spray applications focus on droplet movement and air flow after they are discharged from sprayers and spray droplet displacement under simplified field conditions. Lagrangian trajectory approach is used to calculate droplet movement in laminar or turbulence air flow fields. CFD accuracy to determine the relative effects of droplet size, wind speed, turbulent intensity, initial droplet velocity, droplet discharge height, temperature and relative humidity on droplet displacement (Reichard et al., 1992b; Zhu et al., 1994) and collection efficiency (Zhu et al., 1996) was validated in a wind tunnel (Reichard et al., 1992a). The greatest difference between measured and simulated droplet displacement in the wind tunnel was 5.4% for the droplet sizes ranging from 148 to 424 μm and wind velocities ranging from 0.5 to 6.2 m/s. Spray drift potentials from ground sprayers were also simulated (Brown and Sidahmed, 2001; Sidahmed and Brown, 2001; Baetens et al., 2007; Nuyttens et al., 2011), to improve sprayer performance (Weiner and Parkin, 1993; Molari et al., 2005), assist design of a pneumatic shielded spraying system for increasing spray deposition and reduce spray drift (Tsay et al., 2002a, 2002b), simulate jet flow in sprayer tanks to increase tank mixture uniformity (Ucar et al., 2001), and simulate air flow and droplet movements inside canopies for an air-assisted orchard sprayer to increase spray deposition on leaves (Delele et al., 2005; Endalew et al., 2010a, 2010b, 2010c). Tsay et al. (2002b) used CFD simulation to evaluate different designs of mechanical shields attached on boom sprayers and demonstrated that the addition of a double-foil shield (Fig. 6a) on a sprayer could reduce potential spray drift up to 50% while adding an inclined plate ahead of the shield (Fig. 6b) would worsen the shield performance. With CFD simulations Endalew et al. (2010c) were able to determine droplet trajectories inside pear canopies for choosing optimal nozzles for air-assisted tower sprayers to maximize droplet deposition on leaves and minimize off-target loss (Fig. 7).

However, the complexity of CFD operation limited its practical use in spray applications. Hence, the approach of CFD was simplified with the development of DRIFTSIM that is based on a large data base of drift distances calculated from the FLUENT program (Zhu et al., 1995). The DRIFTSIM program quickly estimates the mean drift distances of water droplets discharged from sprayer atomizers for spray applicators without any CFD experience. Holterman et al. (1997) developed a random-walk model (IDEFICS) incorporating travel speed and entrained air below the nozzles to estimate spray drift potentials from boom sprayers. The accuracy of these programs was tested under field conditions to assist users to choose a proper program to predict drift deposits from ground sprayers (Kruckeberg et al., 2011).

2.3. Harvesting and machinery design

2.3.1. Cleaning fan

In modern harvesting machines, one of the critical factors to fulfill the current demand of capacity and output under a wide range of field and crop conditions is the capacity of the cleaning fan. With the increasing power and output demands of the modern grain combine, the cleaning section capacity has become a limiting factor. In order to obtain an effective cleaning action, the fan has to generate a forceful and even air flow over and through the complete width of sieves. The most readily achieved method of increasing the cleaning capacity is by increasing the width of the combine and the sieves to spread the crop material across a wider area in a thinner veil. Increasing the width of the cleaning sieves, so as to increase cleaning section capacity, also involves having to modify the air flow across the increased size of the cleaning sieves. The inherently uneven air distribution of cleaning fans is accentuated with an increase in the width of the cleaning fans. Several researchers have studied air flow in combine cleaning shoes. Streicher et al. (1986) measured air velocities at multiple locations in the combine cleaning shoe during harvesting of wheat at material other than grain (MOG) flow rates of up to 10 kg s^{-1}. They found that in general, velocities in the chaffer decrease as total material flow rate (grain and MOG) increases. However at intermediate total material flow rates, velocities at the rear increase slightly.

Peters (1995) suggested geometric changes to the paddle fan to obtain an efficient cleaning action. They presented a cleaning fan with two outlets, with the first outlet having two ducts for improved air circulation. Jonckheere (1997) patented a centrifugal fan, modified to have a cross-flow inlet opening throughout its width as in a cross-flow fan in addition to the well known traditional side inlets, installed in generally volute-shaped fan housing with two outlets.

Gebrehiwot et al. (2010) investigates the effect of an additional inlet opening on the performance of the cleaning fan using CFD. They combined simulations with experimental measurements by hotwire anemometers, to study the influence of a cross-flow opening on the performance and flow distribution of a forward curved wide centrifugal fan with two parallel outlets. Three forward curved fans of the same dimension were considered (Fig. 8). Fan-I is an ordinary forward curved centrifugal fan with two axial inlets, while fan-II and fan-III have a cross-flow inlet in addition to the axial inlet opening in fan-I.

To assess the effect of the cross-flow opening on the performance of the centrifugal fan, CFD simulations were performed. The transient, three-dimensional, viscous, incompressible URANS equations were solved and the Re-Normalization Group (RNG) k–ε was used as a turbulence model. The calculation is performed unsteady, because of the highly transient flow in the blade channels. The velocity vector plots of the cross-flow opening of fan-II and fan-III at no load are shown in Fig. 9. The cross-flow opening of fan-II (Fig. 9a) is actually not fully an inlet, as air is coming out of some parts of it. There is a relatively large amount of air coming out of the frontal middle area across the width. In the other parts of the opening, air is entering the fan through this opening at a relatively smaller velocity. Thus, even when running at no load this opening is not completely an inlet. Close monitoring of the flow in this opening shows that a small net amount of air enters the fan. Fig. 9b shows that in fan-III, closing the mid section of the cross-flow opening through which flow is observed to come out in fan-II, did not improve the situation as large amount of air comes out through the frontal area of the cross-flow opening.

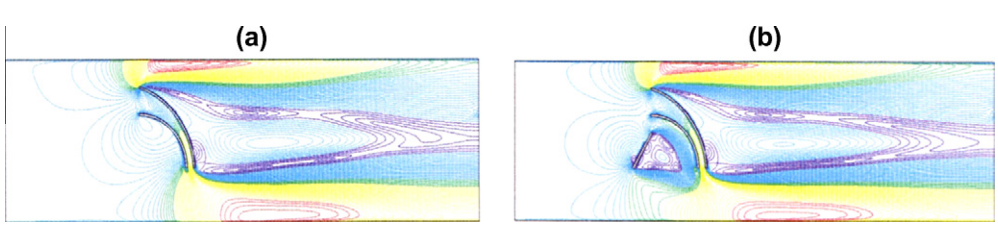

Fig. 6. Simulated convergent flow fields of two mechanical shields attached on spray booms to prevent droplet drift: (a) double-foil shield, and (b) tent shaped shield (Tsay et al., 2002b).

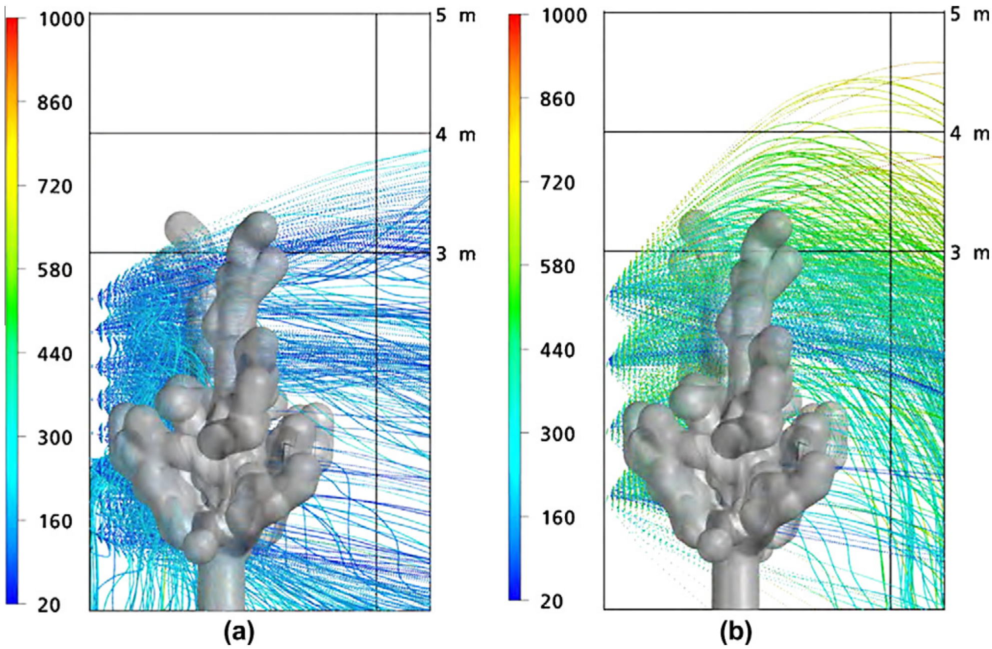

Fig. 7. CFD simulated droplet trajectories passing through a pear tree for the air assisted tower sprayer equipped with (a) hollow cone nozzles and (b) flat fan nozzles (Endalew et al., 2010c).

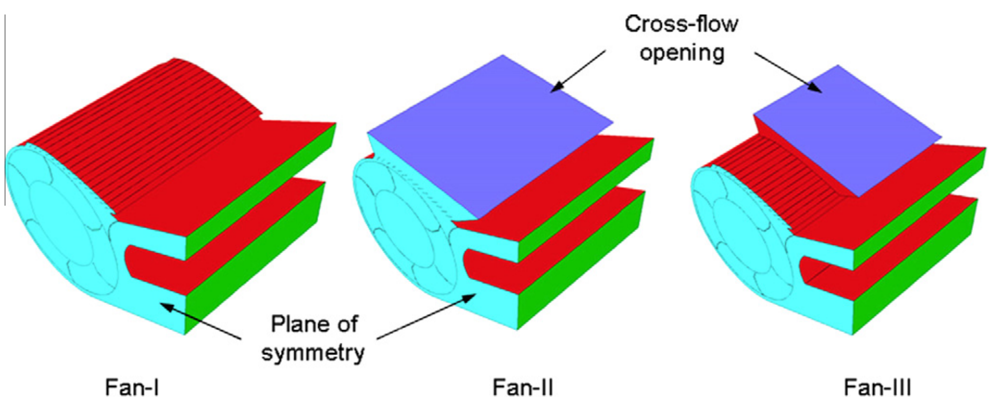

Fig. 8. Geometrical configuration of the fans under consideration (Gebrehiwot et al., 2010). Fan-I is an ordinary forward curved centrifugal fan with two axial inlets, while fan-II and fan-III have a cross-flow inlet in addition to the axial inlet opening in fan-I.

Using the CFD simulation results, Gebrehiwot et al. (2010) observed that availability of the cross-flow opening increases the flow rate created by the fan at low loads. At higher loads however, all fans generate similar flow rates. Comparison between measurements and simulations showed that the three-dimensional URANS CFD simulations could predict the performance of the cleaning fan with an error of less that 10%.

CFD modeling allows the prototypes of the harvesters to be developed and tested to obtain essential parameters for engineering design without building a physical prototype, which requires more time and is relatively expensive. Consequently, the CFD modeling allows the harvester designs to be more easily optimized, creating greater control over the physical process of gas-particle separation. In this research, the three dimensional design of nut harvester geometry was computationally generated based on an initial prototype of commercial almond-harvester developed by a commercial manufacturer (Flory Industries, Salida, CA).

Ponpesh and Ken Giles (2008) applied CFD modeling to improve the design of a nut harvester in order to reduce particulate matter emissions while maintaining product quality and harvesting speed. Fluent® was used for the simulations and the realizable k–ε and Stochastic Lagrangian Discrete Phase Models were used to study the gas-particle flow in the harvester. Model validation was performed using experimental measurements of static and dynamic pressures of the flow fields, particle flow pattern and collection efficiency. Fig. 10 shows the velocity vectors in the separating chamber. The velocity of the gas flow is relatively low and essentially equal to zero at the bottom of the separating chamber. As indicated by Ponpesh and Ken Giles (2008), this may benefit the collection efficiency of the particles by minimizing particle movement and the re-entrainment of the particles into the gas stream which proceeds toward the outlet. The velocity is increased during the semicircular section and is decreased toward the outlet. Larger (heavier) particles moving at a lower speed are exerted by greater centrifugal force become detached from the flow and driven toward the wall of the separating chamber. In addition, the simulated vectors of velocity predict some recirculation near the upper wall of the outlet section, slightly down stream from the semicircular section.

Concerning the comparison with experimental data, although in general there was a good agreement between measured and

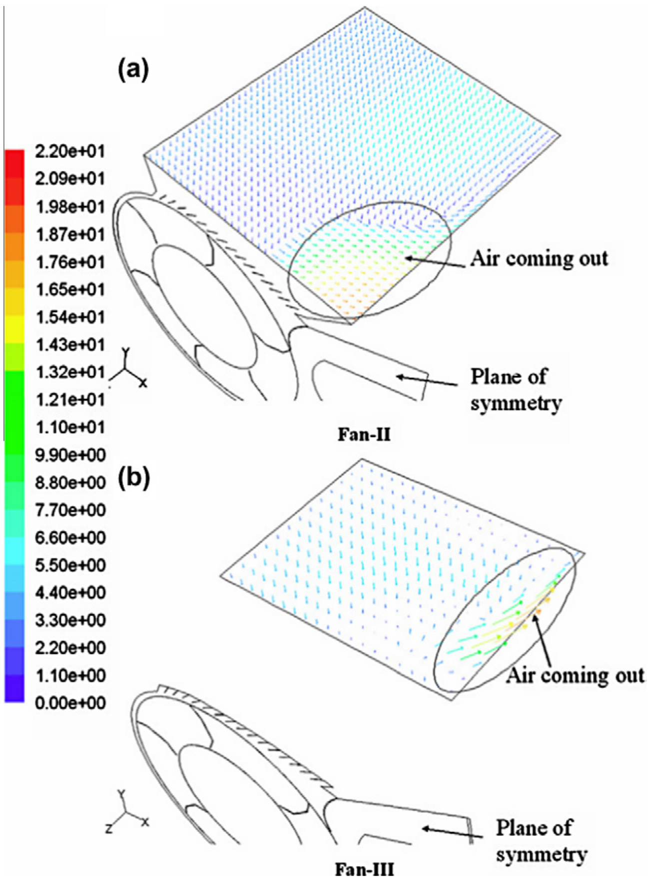

Fig. 9. Velocity vector plots of the cross-flow opening at no load: (a) fan-II and (b) fan-III (Gebrehiwot et al., 2010).

simulated (CFD) results, there was some deviation between the pressure from the experiments and the simulations at the location before the flow entering the separating chamber and in the outlet region. It is possible that this difference arises from the effects on the flow field of the pitot tubes due to their relatively large dimension and shape of the probe tips. Other factors such as the uncertainty of turbulent flow measurement, outlet boundary condition and the steady state assumption are also pointed out by the authors. In general, Ponpesh and Ken Giles (2008) concluded that the CFD model provided a convenient way to study the effects of essential parameters related to harvester geometry and the operational conditions which are important for design optimization.

2.3.2. Turbo machinery

CFD may be also employed to assess the performance of turbo machinery. Combined with measurements provide a complementary tool for simulation, design, optimization and analysis of the flow field inside a turbo machine. Zhang et al. (1996) computed the three-dimensional viscous flow in a blade passage of a backswept centrifugal impeller at the design point using the standard k–ε model. Studies by Dilin et al. (1998) and Thakur et al. (2002) for a centrifugal blower, Muggli et al. (2002) for a mixed flow pump, Yedidiah (2008) for design of a centrifugal pump and Miner (2000) for axial and mixed flow pumps also demonstrated the accuracy of CFD for turbo machinery performance prediction.

2.4. Greenhouse systems

Greenhouses play an important role in the field of agriculture, allowing the grower to have more control on the environmental factors that govern the behavior of crops. Failure to maintain consistent appropriate levels of climate conditions within greenhouses may lead to a decrease in the quantity, quality of the crop, and reduced resource efficiencies in crop production. Proper greenhouse climate control and management of resources in greenhouse crop production systems require feedback control systems that receive environmental measurements using sensors, which may have limitations in sensitivity and spatial distribution. Furthermore, experimental studies for greenhouse ventilation and aerodynamics analysis are costly, time consuming and unpractical. Hence, modeling can be utilized as a preferred tool to study the details of greenhouse environments. However, the behavior and interactions of the environmental factors of a greenhouse are complex involving a number of physical and chemical properties of matter that are challenging to model. Researchers have used various methods to understand these complex phenomena within controlled environments crop production systems.

Amongst various experimental methods used to study greenhouses are tracer gases, full-scale and small-scale greenhouses equipped with appropriately distributed sensors, energy balance methods, numerical wind-buoyancy effect models and wind tunnels (Lee and Short, 2000). Boulard and Draoui (1995) compared three types of tracer gases: CO_2, NO_2 and water vapor to measure ventilation rates in the presence and absence of crops. Molina-Aiz et al. (2004) used smoke pencils to determine the direction and overall streamline behavior of the air flow.

Besides conventional CFD methods, researchers have used other approaches to computationally simulate climate conditions. Jimenez-Hornero et al. (2005) used a computational numerical approach involving a lattice Bathnager, Groos and Krook (BGK) model to describe natural ventilation without considering the effect of buoyancy. The results were similar to that generated by CFD. This method has some limitations, including the need to empirically calculate geometry-dependent constants. Abbes et al. (2010) modeled a tunnel greenhouse using a pseudo bond graph model approach that coupled heat and mass transfer. The method assumed complete uniform homogeneous conditions within the greenhouse and therefore did not require the same computational capacity as conventional CFD.

2.4.1. Greenhouse ventilation

Bartzanas et al. (2004) and Fatnassi et al. (2006) simulated the use of nitrous oxide (N_2O) as tracer gas and fitting the data to an exponential decay equation calculated greenhouse ventilation rate. Hong et al. (2008) concluded that simulated tracer gas models, as opposed to experimental methods, were a more accurate method of calculating ventilation rates, as the vents could serve as both inlet and outlets at the same time, which is difficult to determine experimentally.

Natural ventilation is the most commonly used greenhouse cooling process around the world since most of greenhouses are naturally ventilated. Hoxey et al. (1993) studied wind load codes for different greenhouse designs generating detailed information on airflow profiles due to wind loads on structures. Mistriotis et al. (1997) focused on studying greenhouse ventilation systems with maximum efficiency and optimum vent configuration. Kacira et al. (1998) determined air exchange rates for multispan sawtooth greenhouses. The study showed that the side vent openings in all cases had a significant effect on ventilation, which was directly and linearly proportional to the outside wind velocity in the 0.5–2.5 $m\,s^{-1}$ range. Boulard et al. (1999) studied the incoming and outgoing airflow patterns in a single span greenhouse and compared the results with experimentally obtained values. Their results were often qualitative because the spatial density of the measurements was not dense enough to allow detailed comparisons with the simulation, except for some places such as in the opening. When

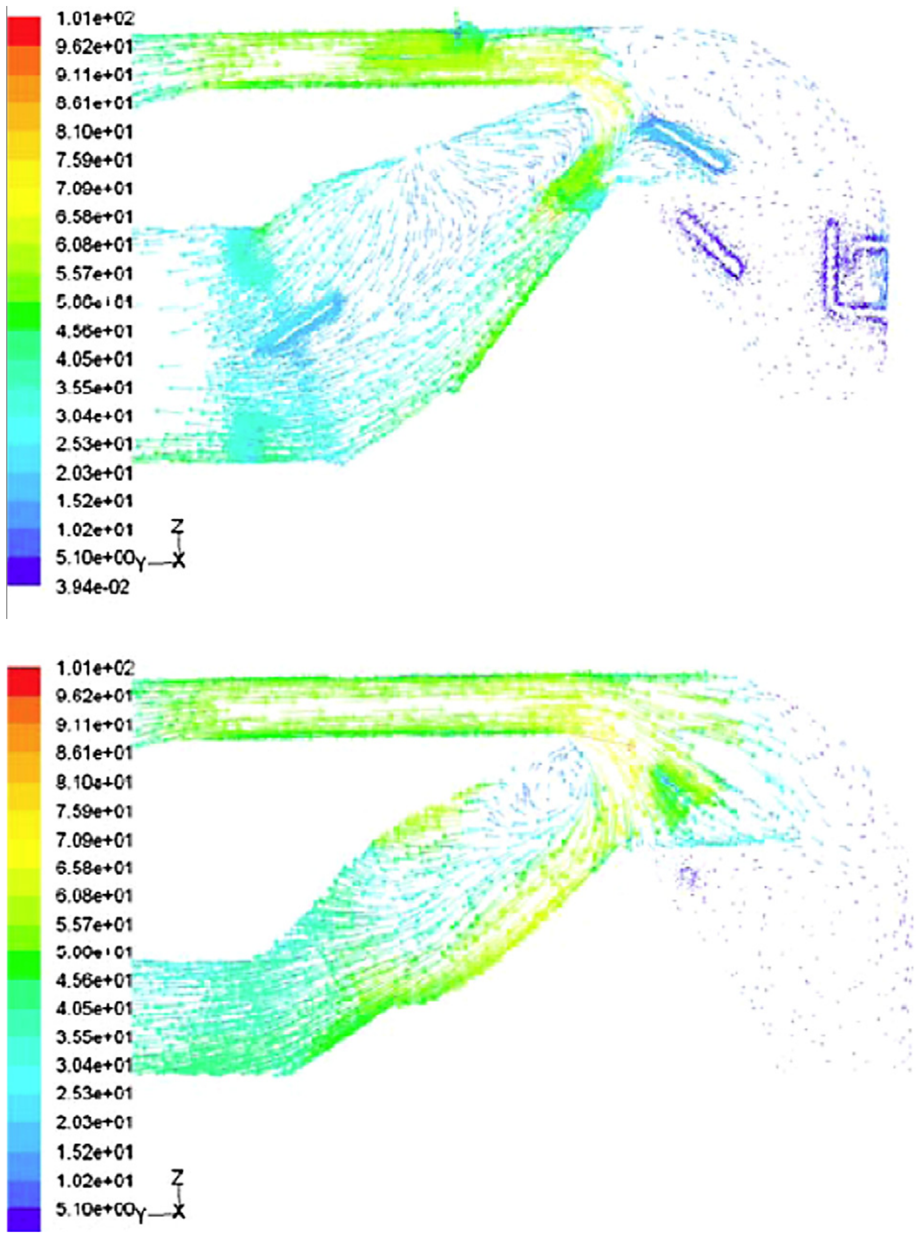

Fig. 10. Vectors of velocity on the midplane (top) and on the intersectional plane (bottom) within the separating chamber from CFD simulation using realizable k–ε model with 2nd order upwind computational scheme and the non-equilibrium wall functions, (Ponpesh and Ken Giles, 2008).

comparison was possible good agreement was found between measured and simulated values. They analyzed airflow and temperature patterns induced by buoyancy forces through greenhouse roof openings using Rayleigh-Benard convection pattern. The last authors showed that this method stressed the origin of the convective motion and the air renewal as being due to a temperature gradient taking place over the whole height of the greenhouse, rather than over the height of the vent section only.

Lee and Short (2000) considered the effects of wind speed, wind direction, vent opening sizes and the presence of crops on natural ventilation rates of a Quonset type greenhouse and concluded that CFD was an effective method for evaluating the relative performance of alternative designs of greenhouse structures and air distribution patterns inside and outside the greenhouse. However, the results presented in their study were not validated against experimental measurements. Campen and Bot (2003) showed that the ventilation flux for the whole greenhouse is linearly proportional to the wind speed and that wind direction can have a significant effect on ventilation rate, up to 50% in some cases. Depending on the wind direction, the difference between measured and simulated values was between 7% and 60%. The last authors did not take into account the buoyancy effect directly but only by imposing the 50% of incoming solar radiation on the ground which then transferred into sensible heat. Teitel et al. (2008) investigated the effect of different wind angles on ventilation rates and air flow pattern and found that the direction of wind affects the flow pattern significantly. Though quantitatively their numerically predicted results were in reasonable agreement with estimates of ventilation rates obtained by a model given in the literature, quantitatively, their CFD simulated air velocity values had large differences with the measured ones at most measurement points of the roof openings. Thus, they mention that there may be several reasons for this result. Usually in real greenhouses, the heat loads due to solar radiation on the crop, the structure elements and the greenhouse cover are not homogeneous while in simulations they are commonly assumed uniform. In addition, as indicated by Norton et al. (2007), there have been many situations where the standard k–ε model failed to sufficiently represent the turbulent regime, with predictions often proving to be totally inaccurate. Yet due to its favorable convergence behavior and reasonable accuracy the standard k–ε model is routinely used.

Shklyar and Arbel (2004) studied the velocity profiles and streamlines to understand the effects of wind direction and vent angles, whether they are windward or leeward (Fig. 11). They pointed out that the symmetry arguments used to simplify CFD modeling from 3D to 2D may not be valid in the general case, especially when the wind direction is arbitrary.

Kacira et al. (2004a) studied the effect of side vents and number of spans in a gothic greenhouse equipped with continuous roof vents on the leeward side of each ridge (Fig. 12). Without the existence of buoyancy effect in the computations, the ventilation rate increased linearly with the external wind speed. The ratio of the opening of the ventilator area to the greenhouse floor area studied was 9.6%, and it was found to be small compared to the recommended minimum ratios of 15–25% for proper greenhouse exchange rates. The results showed that a significant reduction in ventilation rate was determined as the number of spans was increased (i.e. from 6 to 24 spans) and an exponential decay described the relationship between the ventilation rate and the number of spans (Fig. 12a and b). However, it has to be noted that the above authors did not validated their simulated results by experimental measurements.

Kacira et al. (2004b) indicated that in naturally ventilated greenhouses, depending on the vent configuration, it is observed that air entering the greenhouse from vent openings tends to travel along the ceiling and leaves the greenhouse without reaching the plant canopy zone, thus limiting the effectiveness of air exchanges in the plant canopy zone for air renewal. The 3D CFD study (Fig. 13) showed that the maximum greenhouse ventilation rates were achieved when rollup type side vents, as opposed to butterfly side vents, were used in the side walls and both side and roof vents were fully open. The rollup side vents considerably enhanced the ventilation rate in the plant canopy zone for the two span greenhouse design investigated. The CFD model used in this study was validated against experimental data with acceptable quantitative agreements between the CFD simulations and experimental results.

Bartzanas et al. (2004) studied greenhouse velocity profiles and ventilation rates in a greenhouse with tomato crop and under four vent configurations. After validation of the CFD model against experimental values, they showed that the largest ventilation rate did not necessary correspond to the best greenhouse air temperature and velocity distribution. Molina-Aiz et al. (2004) modeled air ventilation within an Almeria-type greenhouse. The results showed a chimney effect in which air came in through the side vents and hotter air left the greenhouse through the roof vents and verified the importance of roof ventilation in multi-span naturally ventilated greenhouses. Their model was validated by comparing the numerical results with experimental data. The differences between values predicted by the CFD models and those measured were from 0.0 to 0.36 m s^{-1} for air velocities, and from 0.1 to 2.1 °C for air temperatures. A canarian type greenhouse with insect screens was simulated by Majdoubi et al. (2009). The numerical solution showed flow patterns that indicated a strong flow from the windward openings to the leeward openings, with air loops being developed due to buoyancy forces. Simulated data concluded that even low wind velocities created a strong wind-wise current above the canopy with a lower reversed wind profile in the canopy area.

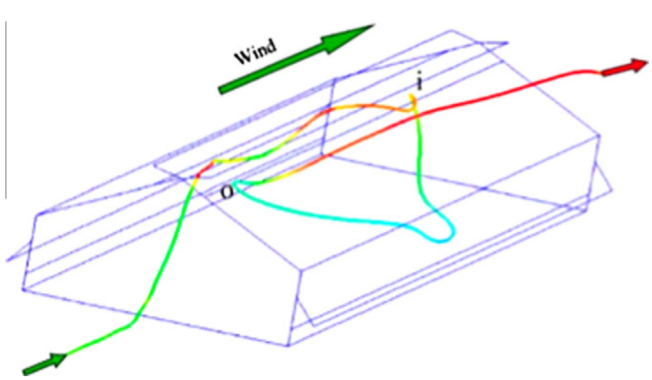

Fig. 11. CFD generated streamline illustrating the airflow through a pitched roof greenhouse (Shklyar and Arbel, 2004).

Greenhouse ventilation flow is usually associated with turbulent motion. A study by Mistriotis et al. (1997) assessed the use of different turbulence models in CFD. The more popular and widely used model is the k–ε model. Researchers have updated the original k–ε model by adding specific volumetric terms that do well to predict the dissipation nature of the airflow, such as the Chen–Kim (CK) model and the ReNormalization Group (RNG) model (Mistriotis et al., 1997). To verify that the k–ε model being used is accurate, Mistriotis et al. (1997) simulated published experimental air flow profiles using the three turbulence models discussed above to both validate the models. The results were qualitatively compared and the CK and RNG models seemed to be most accurate in recreating the experimental results. Bournet and Boulard (2010) stated the realizable k–ε model is more accurate than the standard κ–ε model in describing turbulent flow with recirculation and rotation and strong pressure gradients. Teitel et al. (2008) used the standard k–ε model due to the dimensions and velocities existed in their study and the flow was turbulent. They indicated that the standard k–ε model has a compromise between acceptable computational cost and modeling accuracy for turbulent flow simulation, thus it has a favorable convergence performance with reasonable accuracy and is routinely used. It was pointed out that inaccuracy in modeling results occur when predicting flows under adverse pressure gradients, impingement flows and flows with strong streamline curvature which are common in greenhouse ventilation processes. However, some studies using k–ε equation (Fidaros et al., 2010) mentioned that, the realizable k–ε model did not provide significantly different results, but required more computational time (Shklyar and Arbel, 2004).

2.4.2. Greenhouse heating

Buoyancy plays an important role in airflow in greenhouses. Various researchers considered the buoyancy effect using Boussinesq model in CFD simulations (Bartzanas et al., 2004; Majdoubi et al., 2009; Fidaros et al., 2010). Lee and Short (2000) and Kacira et al. (2004a) did not consider the buoyancy effect, as it was considered negligible compared to the air exchange rate studied. Campen and Bot (2003) confirmed the validity of this assumption experimentally and concluded the effect of buoyancy was minimal in many cases.

CFD simulations by Tadj et al. (2010) described the greenhouse climate heterogeneity caused by buoyancy forces using different heating configurations. Combinations of heating pipes and air heaters were investigated. They validated their CFD model against experimental data and their measured air velocity values were 11–16% lower than the simulated values and their measured air temperature values were 12–17% higher than the simulated ones. Fig. 14 presents the air temperature distribution in a tunnel plastic greenhouse heated by heating pipes placed close to the grow gutter (about 0.3 m from greenhouse ground). The isotherm spreads along the heating pipes from the center towards the walls. These

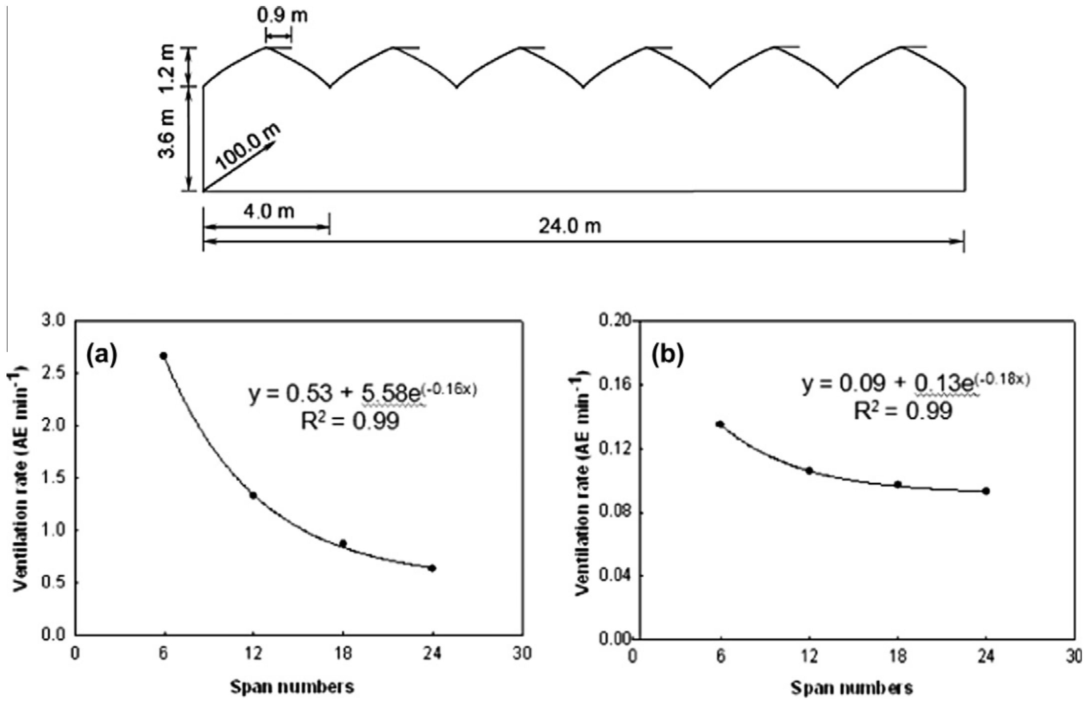

Fig. 12. The effect of span numbers on greenhouse ventilation rate with fully open windward and leeward side vents and roof vents (a) and only roof vents (b) (Kacira et al., 2004a).

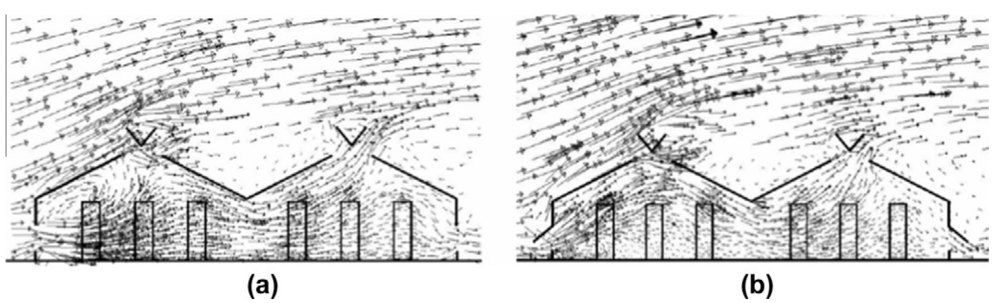

Fig. 13. The effect of side vent configuration on plant canopy zone ventilation, air flow pattern and air exchange process with (a) roll up side vents and (b) butterfly type side vents (Kacira et al., 2004b).

two regions split off in the middle of the greenhouse to ascend away from the heating pipes to greenhouse roof forming a thermal plume. The same temperature pattern was observed by Pretot et al. (2000) who investigated the natural convection on a horizontal plane. Most of the temperature gradients occurred near the greenhouse ground and the roof, while air temperature remains almost constant and stable in most of the greenhouse volume. Tadj et al. (2010) mention that the combined use of heating pipes and air heater enhanced plant activity and reduced the condensation rate. However, this combination led to an increase in energy consumption of up to 19% and created a more heterogeneous climate distribution compared to the case in which only heating pipes were used. The greenhouse air volume was split into two regions: one occupied by the crop where natural convection dominated, and one above the crop where the hot air from the air heater resulted in a different microclimate from the lower part of the greenhouse (crop level), and the convection mode changed to mixed or forced depending on the distance from the air heater. The above findings suggest that in case a combined system is used, the position of the exit of hot air from the air heater is very important and it would be probably better to be located at the level of the crop, homogeneously distributed along the greenhouse by a perforated tube.

2.4.3. Radiation transfer and distribution

The effect of solar radiation in the greenhouse system is a dynamic phenomenon (Fidaros et al., 2010) (Fig. 15). However, in the majority of the earlier studies that used CFD, there was no simultaneous solving of the coupled radiative and convective transfers. In fact, the radiative transfer equation (RTE) was not solved using a radiation model but by imposing wall temperature at an assumed value to simulate solar radiation, as was the case for Kacira et al. (1998) and Bartzanas et al. (2004). Other authors (e.g. Mistriotis et al., 1997) set specific heat flux physical boundary conditions. In order to account for multiple radiative exchanges between the different components of a greenhouse, suitable models were integrated (i.e. solar and atmospheric radiative components, Bournet et al., 2007). Although different approaches for solving the RTE exist, i.e. the P1 radiation model, it seems that the Discrete Ordinate (DO) model has been the most utilized in greenhouse applications (Bournet et al., 2007), as is also the case in the study by Kim et al. (2008), which allows for the solution of radiation at semi-transparent walls.

Bournet et al. (2007) used a more complex model in which shortwave and longwave radiation was considered separately, while accounting also for the interactions between the different parts of the greenhouse and the radiation. Recently, Baxevanou et al.

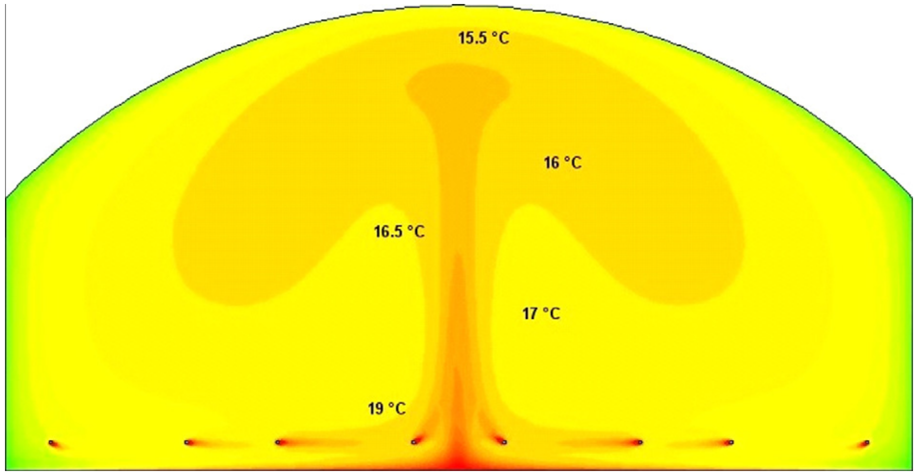

Fig. 14. Vertical cross sectional view of air temperature contours in the middle of a greenhouse heated by pipes (Tadj et al., 2010).

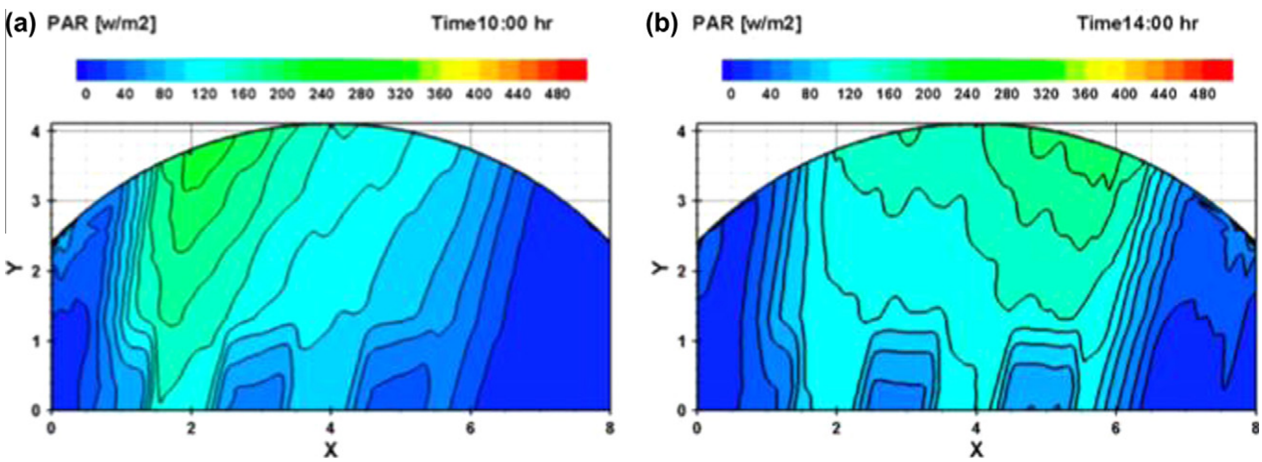

Fig. 15. Simulated PAR radiation distribution in high tunnel greenhouse at constant ambient temperatures at two different times of the day (Fidaros et al., 2010).

(2010) developed a CFD model to investigate the distribution of solar radiation in a naturally ventilated arc type tunnel greenhouse and its influence on greenhouse microclimate, taking into account the external incident radiation distributed in three wavelength bands and the radiometric properties of the involved materials. The model was used for predicting solar radiation and air temperature distribution under different solar radiation inclinations and greenhouse covering materials. Fig. 16 presents the distribution of photosynthetically active radiation (PAR) for different values of solar radiation inclination. The cover material allows 83% of the incident radiation to be transmitted inside the greenhouse. Part of this radiation is reflected by the plants (22%) while that part of PAR that is absorbed heats the crop which radiates at much higher wave lengths. The intensity of PAR inside the greenhouse is time and angle dependent and presents almost uniform distribution over the plants for angles around solar noon (cases b–d). Nevertheless, in all cases the maximum PAR intensity is reaching roughly the same value, illustrated in this way the effectiveness in the homogeneous distribution of solar radiation of greenhouses with arc roof compared to greenhouses with other roof designs. Further elaboration of the above model could help greenhouse designers to find the best greenhouse shape for each location in order to optimize radiation interception according to the crop needs.

Tong et al. (2009) simulated temperature distribution in a Chinese solar greenhouse for varying solar conditions using time dependent CFD models and validated the results by experimental measurements. The study evaluated the cloudy and clear conditions and studied the heat flow during both scenarios. The model underestimated air temperature which suggests that greenhouse structure absorbed more heat than expected. The results found during a clear day indicate that only the surface temperatures on the northwall and the soil are higher than the interior air temperatures during the night, so only these surfaces contribute heat to the interior air to maintain the air temperatures during the night. However, on a cloudy day, the north wall and roof and the soil all contribute heat to the interior air throughout the day since the interior air is not heated significantly by the substantially reduced solar insulation on the cloudy day. The simulated values were in very good agreement with the measured one and accordingly the model could be used reliably for several case studies. Thus, further elaboration of the model will help to evaluate the effects of different structural design and climatic conditions on the greenhouse microclimate so as to optimize the greenhouse design.

2.4.4. Greenhouse cooling

Common techniques to cool greenhouses are natural ventilation, whitening, shade screens and evaporative cooling (Perdigones et al., 2008). Since natural ventilation is the least expensive and requires little maintenance, it is more common than other cooling methods. However, high ventilation rates are not necessarily the best methods to cool greenhouses as increasing the ventilation rate after a certain level increases the transpiration level, which decreases the

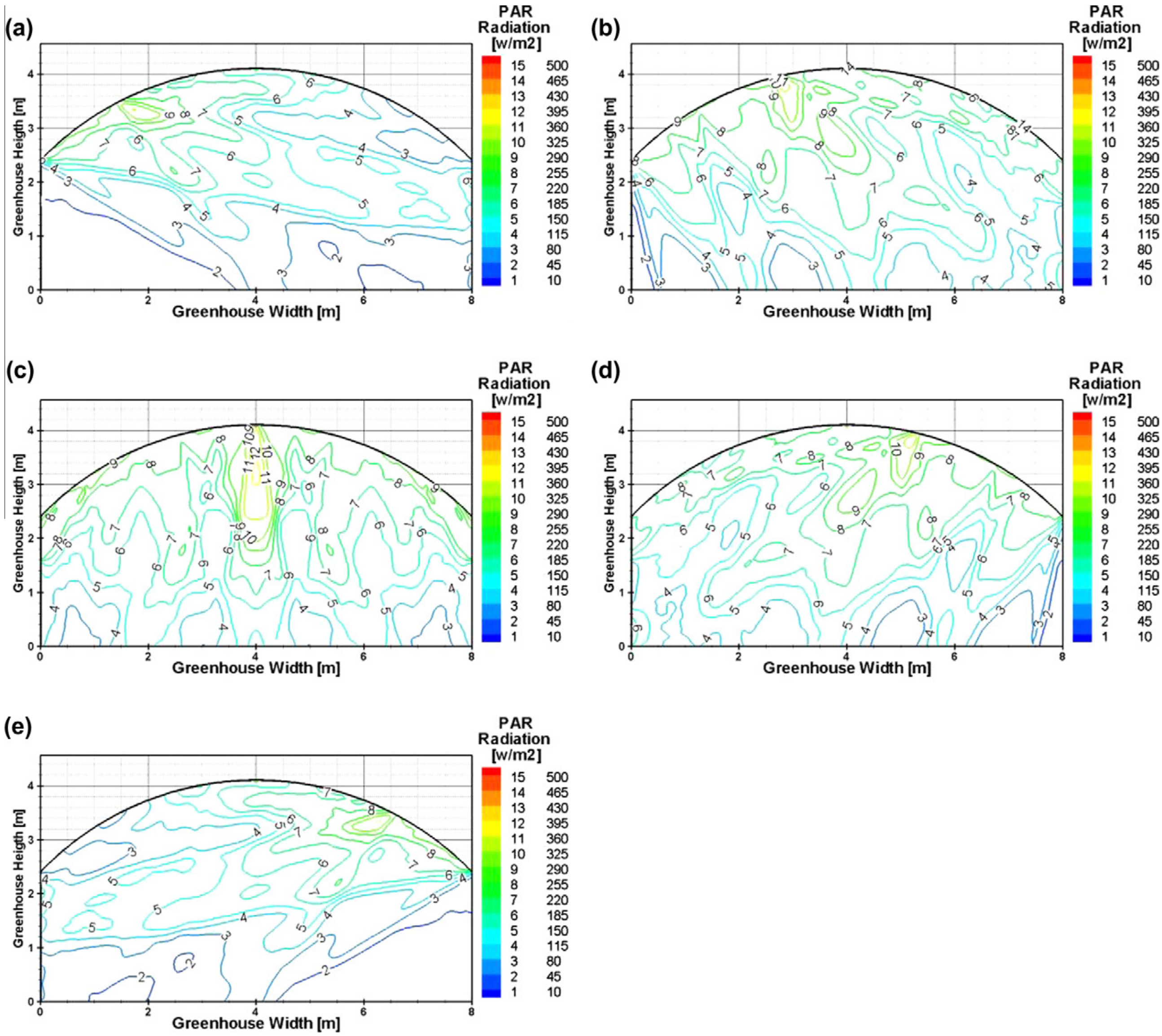

Fig. 16. PAR distributions for different inclinations of the incident solar radiation (a–e) (a) −60°, (b) −30°, (c) 0°, (d) 30° and (e) 60° (Baxevanou et al., 2010).

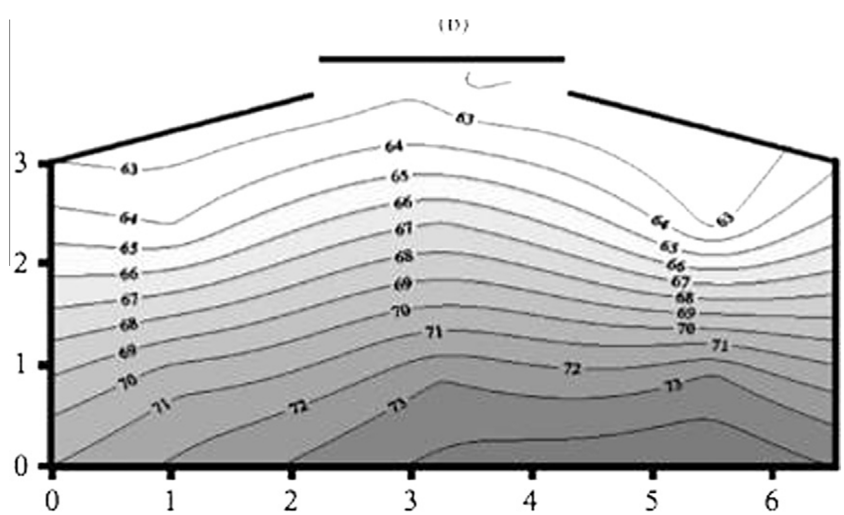

Fig. 17. Cross sectional simulated contour map of relative humidity within a greenhouse equipped with fogging (Kim et al., 2008).

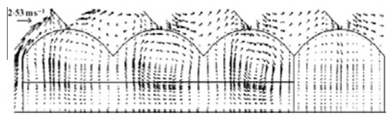

Fig. 18. Velocity vectors of airflow through a greenhouse with side vents open equipped with insect screens (Fatnassi et al., 2006).

overall positive contribution of ventilation (Sapounas et al., 2007). Perdigones et al. (2008) suggested that evaporative methods of cooling are the most efficient way of maintaining lower temperature and increasing humidity in greenhouses. Franco et al. (2011) modeled different geometries of evaporative pads using a validated CFD model. The relationship between air velocity, porosity and pressure drops was investigated. Differences between simulated and experimental results from a custom built pad apparatus ranged between 9% and 16%. Although there have been researchers who modeled humidity using an integrated species model (Sapounas et al., 2007), there are few cases that use CFD to model humidity levels of a greenhouse equipped with fogging. Kim et al. (2008) used the discrete phase model in which the equation predicted the trajectory a fog droplet by integrating the force balance equating the particle inertia with the forces acting on the particle (Fig. 17). Heat balances were utilized to predict convective and latent heat transfers between the particle and the continuous phase. The simulation data was in good agreement with the experimental data, with errors that ranged between 0.1% and 18.4%.

2.4.5. Crop simulation

Transpiration from greenhouse crops is a very involved process because it alters the local climate, which ultimately modifies the conditions that effect transpiration. Bartzanas et al. (2004) utilized a user-defined function (UDF) for crop water and heat exchanges without the use of a radiation model. Sensible and latent heat exchanges of the crop were modeled using equations involving stomatal resistance. The experimental and simulated results varied by 12–15%. However, Wang and Boulard (2000) found that the most important factor that affects transpiration was solar radiation. Complete set of equations for the transpiration rate and solar radiation estimations were developed and reported by Boulard and Wang (2002). The last authors used a UDF to simulate solar energy partition into sensible heat and latent heat, and couple solar radiation with convection and plant physiological properties. Other authors (Sapounas et al., 2007) used a crop porous medium model with UDF to simulate crop evapotranspiration. Fatnassi et al. (2006) used a dynamic model that described the physical effects of the porous medium, which also involved a macro-model of heat and mass transfer between the crop cover and air. However, to the best of our knowledge, none of the above studies was able to take into account and simulate the thermal negative feedback effects linked to the energy balance and the hydrological negative feedback effects linked to the water balance that contribute to the reduction of stomatal control on transpiration rate (Aubinet et al., 1989). Accordingly, it seems that much work needs to be done in order to be able to simulate crop sensible and latent heat exchanges under suboptimal climate conditions i.e. conditions that induce crop stress and may lead to stomatal and transpiration rate reductions, as is the case many times in real situations.

2.4.6. Crop protection

New insect pests have made the use of insect-proof screens necessary in controlled environments. Fatnassi et al. (2006) used CFD to study the effect of screens on airflow behavior so that greenhouse design may be updated to accommodate screens use (Fig. 18). The simulated data concluded low wind efficiency, which was due to the presence of the screen and the presence of partitions within the multi-ispan greenhouse used. Teitel (2010) investigated the validity of certain existing models to estimate the permeability and inertial factor of different insect screens. The

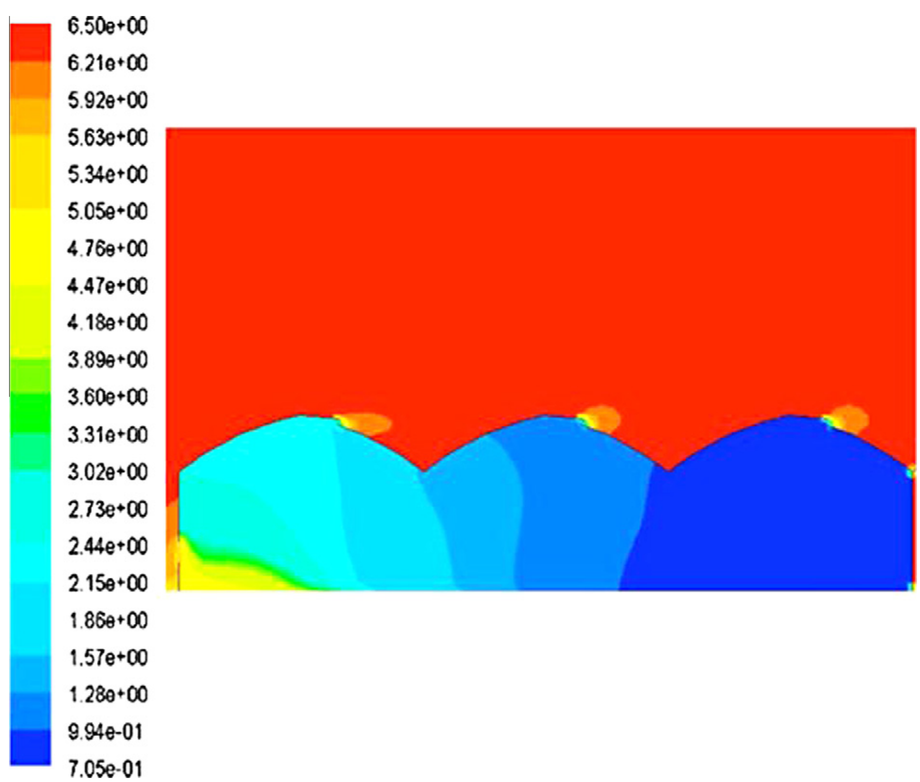

Fig. 19. Simulated distribution of spores concentration within a multi-span greenhouse and domain (Boulard et al., 2010).

study also compares the insect screen porous media method to that of a more realistic woven screen model. Results from both methods, using the k–ε turbulence model, were in good agreement.

Researchers modeled insect screen as porous media explicitly by means of the Darcy–Forcheimer equation (Molina-Aiz et al., 2004; Majdoubi et al., 2009). Boulard et al. (2010) combined a particle dispersal Eulerian model with validated greenhouse microclimate dynamic CFD models to study *Botrytis cinerea* spore dispersal within a greenhouse domain during day and night (Fig. 19) The particle dispersal model was validated using spore balance experimental data using a Petri dish method, which found good agreement with simulated data. The last authors concluded that spore development is governed by the ventilation configuration and insect screens use.

3. Discussion

3.1. Future challenges for CFD modeling in precision crop production

CFD is the art and science of analyzing and simulating systems in which a fluid flow is of central interest and in which heat and mass transfer and chemical reaction may take place. Its advantages over conventional experimental studies are substantial reductions in lead times and development cost, availability to study systems where experiments are not possible and ease of performing a large range of parametric studies for optimization. Although it starts from the design process in industry it has been rapidly used to other application including the greenhouse sector. In the present paper the basic applications of CFD for precision crop production was presented and analyzed. However there is a lot of work to be done and challenges for CFD modelers.

Analyzing soil dynamic behavior using CFD simulation will help in optimizing the design of tools with different shapes in order to reduce energy demand and to help modeling deferent types of soil-tool interaction based on their visco-plastic parameters. Important issues that are needed to be addressed for predicting performance of tillage tools with a numerical approach include tool geometry, dynamic nature of the task, complex soil characteristics and effectiveness of constitutive models. Since most of the recent studies on tillage mechanics using CFD considered single phase soil medium, further study should be conducted on the compressible soil considering the pore spaces at different soil conditions. Sensitivity studies need to be conducted on the effect of soil visco-plastic parameters on soil failure front, soil furrow formation and energy requirement. Study need to be conducted for soil deformation pattern and force prediction with different rake angles of the tool and with different shapes of the tool. Dynamic analysis of tillage for an implement with multiple tines for optimization of soil disturbance with the perspective of conservation tillage should also be investigated.

The interest for growing crops on artificial substrate in greenhouses is mainly due to the strong dependence of plant production on environmental conditions, and to the fact that optimum growing conditions are rarely met under open field conditions because of a poor water and mineral availability. While the volume of water consumed by the plants is known almost instantaneously, moisture variations within the growing medium (soil or substrate for greenhouse soilless crops), particularly dry and saturated zones are not precisely located. Accordingly, it would be important to determine the dynamics of water and nutrients in the growing medium because this would allow better time and space management of the water and nutrient supply according to plant needs during each step of the crop cycle. The motion of water in rockwool slabs used as growing substrate for a sweet pepper crop was simulated using CFD by Bougoul and Boulard (2006) but similar studies have not been found for water or nutrients motion in other growing medium or in the soil.

CFD simulations provided useful tools to determine relative effects of variables on droplet trajectories under complicated field conditions and assisted engineers to choose optimal sprayer parameters to maximum pesticide spray application efficiency. However, due to field conditions are mostly uncontrollable, current CFD simulations on pesticide spray applications are incapable of accurately predicting droplet movements and air flow from field sprayers. Consequently, only the simple models are available to determine the relative influence of different variables on sprayer performances. This situation will continue until we have new technologies that are able to incorporate these uncontrollable variables from real time field conditions into CFD simulations.

The CFD model has provided a convenient way to study the effects of essential parameters related to harvester and machinery geometry and the operational conditions which are important for design optimization. However more work is needed for accurately validating the CFD results and extending the use of CFD not only for optimizing the designing process but for operational control also.

Although a lot of works have already been published for analyzing the spatial distribution of greenhouse microclimate and its structural and design specifications, applications or incorporations of CFD models in general greenhouse climate models is not yet standard. Solving such complex problems requires a great deal of computational power. However, high-performance computing (HPC) is becoming increasingly accessible, scalable and affordable and they can easily operate, and integrate with existing infrastructure and tools. Development has to be focussed on the application of modeling the climate spatial distribution in the greenhouse system, enabling design of the system with homogeneous climate at design conditions. Problems to be solved are quantification of the interaction between the crop geometry with the flow pattern, modeling of evaporation, modeling of thermal and solar radiation, modeling of ventilation and choosing the proper sub-model for the effect of turbulence. The application in control is far away due to the complexity and time consuming calculations of this kind of modeling. Combination of CFD, mechanistic and black box modeling with on line parameter estimation as used in the principle of imperfect mixing may be applicable for a more precision greenhouse climate control.

To ensure CFD simulations are more than just theoretical exercises, experimental validation is necessary. However, there are usually differences between experimental and numerical studies. These differences are mainly due to the use of empirical values for transpiration, equivalent porous and turbulence models which are very sensitive to air velocity and relative humidity variations. For example in greenhouse environments the flow can be laminar or turbulent according to the prevailing climate variables. In many cases, various modes of turbulent flow co-exist. However, the usually used high Reynolds number k–ε turbulence model, cannot treat the flow accurately in the whole computational domain, as the flow is separated in regions with high and low Reynolds flows and even in subregions where the flow is allocated to transitional mode. The turbulence models $k - \varepsilon, k - \omega$ $k - \bar{u}^2$, SST are proposed to overcome significant spatial resolution problems, while the LES and DNS approaches appear serious difficulties to be applied in mixed low and high Reynolds flows due to mesh and CPU demanding requirements, according to Wilcox (2006). Recently, a predictive laminar to turbulent flow transition model has been incorporated in the ANSYS CFX 10.0 (Anon, 2006) software. However, no research employing this model is yet available and the approach has not been experimentally validated to date.

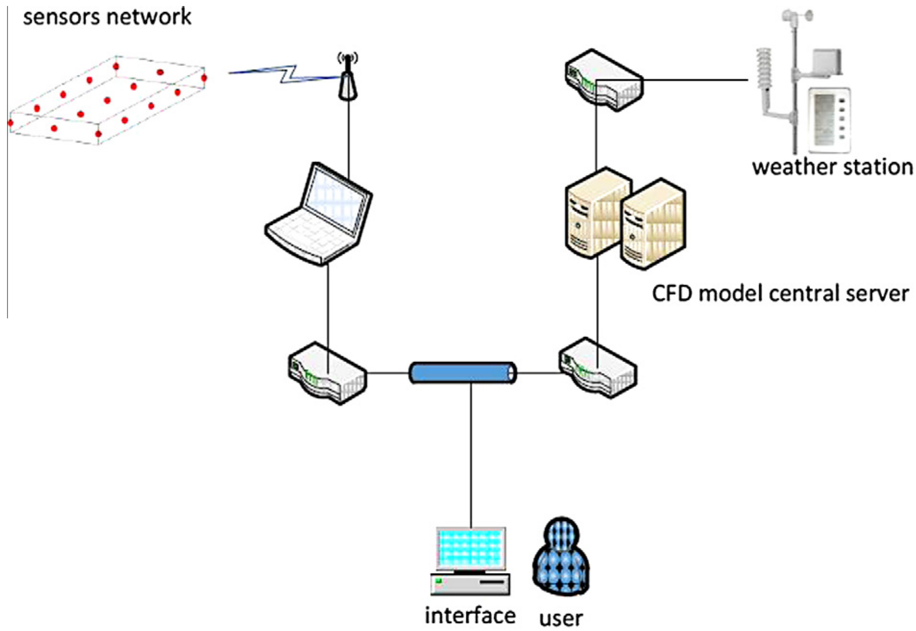

Fig. 20. The basic physical components of the diagnostic system (Bochtis et al., 2011).

3.2. CFD in decision support systems for precision crop production

The complexity and importance of agricultural operations management has increased as agriculture has adapted capital-intensive production systems, thereby stimulating the development of more formal planning techniques. Also, during recent years the general trend towards sustainable farming practices has shifted the very nature of farm planning. From mainly dealing with the traditional way of planning what to do – which crops to grow and which machines to use – the focus has been moved towards problems of how to schedule and carry out different operations. CFD can be combined with other models and experimental devices and it can be effectively be a part of decision support tools in precision crop farming.

Recently Bochtis et al. (2011), presents a diagnostic tool which involves a combination of a sensors network for the acquisition of real-time data in order to provide the actual conditions within the biomass storage facility and a real-time computational fluid dynamics (CFD) modeling approach fitted with selected updated weather data in order to provide the predicted conditions within the storage facility assuming ideal conditions with no interfering events. The two conditions (actual and predicted) are depicted graphically in a user interface allowing the user to evaluate the deviation between the two conditions and decide if preventive actions are required. The principal components of the diagnostic system (Fig. 20) include, a sensor network functioning as a data acquisition unit, a wireless communication unit for the transmission of sensor data to an on-site data storage unit, a data storage unit, a central server running the CFD model and a user interface for visualization purposes. In the near future it is anticipated the CFD models will be used more often in such systems enabling a more precision crop production.

Further studies are also needed to analyze the effect of CFD simulation on resource savings and climate control applications. As an example, Franco et al. (2011) evaluated the effect of several cellulose evaporative cooling pads used in Mediterranean greenhouses. Since evaporative pads make it more difficult for outside air to flow into the greenhouse, less resistance to air flow means lower energy costs, as the fans require less energy to maintain the ventilation rate. Therefore, CFD modeling was used to simulate new pad designs that reduce resistance to the flow of air and therefore increasing their efficiency as a key factor for reducing costs.

The perspectives of this diagnostic system, and maybe similar systems using CFD models, is to be integrated on farm management information systems where the visualization and alerting can involve multi-delivery mechanisms such instant messaging through e-mail, or wireless devices (e.g., cellular phones). This continuously updated information will provide the manager a solid basis for necessary preventive measures in precision crop farming.

4. Concluding remarks

CFD modeling has been successfully used for precision crop farming. Application of CFD to tillage has been proved to be an important tool in addressing many unresolved issues related to dynamic soil-tool interaction. Modeling of energy requirement and predicting of soil failure patterns in large soil deformation during tillage have been successfully analyzed by CFD modeling. CFD simulations also advanced the improvement of pesticide sprayer efficiency for crops comprising of many varieties and species. Future CFD models should accommodate inconsistent canopy structures and weather conditions with sprayer design and operation to demonstrate real-time droplet trajectories. However the majority of CFD studies for precision crop production are focusing on greenhouses. CFD modeling enables researchers to model and evaluate wide ranges of climatic conditions for a given greenhouse system offering to study and compare a number of scenarios to optimize greenhouse aerodynamics and designs. However, experimental data collection and reference data are essential for both establishing the boundary conditions required for the CFD models used and validating the model and results obtained. Until today, majority of the CFD modeling studies for greenhouse aerodynamics were primarily based on two-dimensional analysis. This helped researchers to minimize the complexity of the modeling involved in the problem and also reduced the computational time. However, three dimensional modeling was needed to study the circulating patterns of wind and varying external wind directions and is especially needed to characterize the uniformity of greenhouse interior climate under various crop, cooling system settings and greenhouse designs. There have been studies on greenhouse air flow, ventilation rates, inclusion of radiative heat transfer and crop resistance, however, research on greenhouse cooling with evaporative cooling especially with high pressure fogging under naturally vented conditions has been lacking. Future CFD studies must identify what kind of exchange processes between greenhouse system, its surrounding and crop and complexity of the model must be involved in the analysis to establish more realistic simulations for a given problem.

References

Abbes, M., Farhat, A., Mami, A., Dauphin-Tanquy, G., 2010. Pseudo bond graph model of coupled heat and mass transfers in a plastic tunnel greenhouse. Simul. Modell. Pract. Theory 18, 1327–1341.

Anon, 2006. ANSYS CFX Release 10.0 Technical Specifications. Public Notice. ANSYS, Inc., Southpointe, Canonburg, PA, USA.

Aubinet, M., Deltour, J., de Halleux, D., Nijskens, J., 1989. Stomatal regulation in greenhouse crops: analysis and simulation. Agric. For. Meteorol. 48 (1–2), 21–44.

Baetens, K., Nuyttens, P., De Schampheleire, M., Nicola, B., Ramon, H., 2007. Predicting drift from field spraying by means of a 3D computational fluid dynamics model. Comput. Electron. Agric. 56, 161–173.

Barker, M.E., 2008. Predicting Loads on Ground Engaging Tillage Tools Using Computational Fluid Dynamics. PhD Thesis. Iowa State University, Iowa, USA, 196pp.

Bartzanas, T., Boulard, T., Kittas, C., 2004. Effect of vent arrangement on windward ventilation of a tunnel greenhouse. Biosyst. Eng. 88 (4), 479–490.

Baxevanou, C., Fidaros, D., Bartzanas, T., Kittas, C., 2010. Numerical simulation of solar radiation distribution in a naturally ventilated agricultural building. CIGR E.-J. 12 (3–4), 58–69.

Blackmore, S., 2000. Developing the principles of precision farming. In: ICETS 2000: Proceedings of the ICETS 2000. China Agricultural University, Beijing, China, pp. 11–13.

Blackmore, S., Godwin, R., Fountas, S., 2003. The analysis of spatial and temporal trends in yield map data over six years. Biosyst. Eng. 84, 455–466.

Bochtis, D., Sørensen, C.G., Green, O., Bartzanas, T., 2011. A diagnostic system for improving biomass quality based on a sensor network. Sensors 11 (5), 4990–5004.

Bougoul, S., Boulard, T., 2006. Water dynamics in two rockwool slab growing substrates of contrasting densities. Sci. Horticult. 107 (4), 399–404.

Boulard, T., Draoui, B., 1995. Natural ventilation of a greenhouse with continuous roof vents: measurements and data analysis. J. Agric. Eng. Res. 61, 27–36.

Boulard, T., Wang, S., 2002. Experimental and numerical studies on the heterogeneity of crop transpiration in a plastic tunnel. Comput. Electron. Agric. 34, 173–190.

Boulard, T., Haxaire, R., Lamrani, M.A., Roy, J.C., Jaffrin, A., 1999. Characterization and modelling of the air fluxes induced by natural ventilation in a greenhouse. J. Agric. Eng. Res. 74, 144–195.

Boulard, T., Roy, J.C., Fatnassi, H., Kichah, A., Lee, I.-B., 2010. Computer fluid dynamics prediction of climate and fungal spore transfer in a rose greenhouse. Comput. Electron. Agric. 74, 280–292.

Bournet, P.-E., Boulard, T., 2010. Effect of ventilator configuration on the distributed climate of greenhouses: a review of experimental and CFD studies. Comput. Electron. Agric. 74, 195–217.

Bournet, P.E., Ould Khaoua, S.A., Boulard, T., 2007. Numerical prediction of the effect of vent arrangements on the ventilation and energy transfer in multi-span glasshouse using a bi-band radiation model. Biosyst. Eng. 98, 224–234.

Brown, R.B., Sidahmed, M.M., 2001. Simulation of spray dispersal and deposition from a forestry airblast sprayer. Part II: droplet trajectory model. Trans. ASAE 44, 11–17.

Campen, J.B., Bot, G.P.A., 2003. Determination of greenhouse-specific aspects of ventilation using three-dimensional computational fluid dynamics. Biosyst. Eng. 84 (1), 69–77.

Chi, L., Kushwaha, R.L., 1990. A non-linear 3-D finite element analysis of soil failure with tillage tools. J. Terramech. 27, 343–366.

Davison, J., Calay, R.J., Sands, T.B., England, M., 2002. CFD simulation of soil flow over augers. In: 2002 ASME PVP Conference, Vancouver, British Columbia, Canada.

Delele, M.A., De Moor, A., Sonck, B., Ramon, H., Nicolaï, B.M., Verboven, P., 2005. Modelling and validation of the air flow generated by a cross flow air sprayer as affected by travel speed and fan speed. Biosyst. Eng. 92, 165–174.

Desai, C.S., Phan, H.V., 1980. In: Oden, J.T. (Ed.), Computational Methods in Nonlinear Mechanics. North Holland Publishing Company, New York, pp. 205–224.

Dilin, P., Sakai, T., Wilson, M., Whitfield, A., 1998. A computational and experimental evaluation of the performance of a centrifugal fan volute. Proc. Inst. Mech. Eng. Part A: J. Power Energy 212, 235–246.

Endalew, A.M., Debaer, C., Rutten, N., Vercammen, J., Delele, M.A., Ramon, H., Nicolaï, B.M., Verboven, P., 2010a. A new integrated CFD modelling approach towards air-assisted orchard spraying. Part I. Model development and effect of wind speed and direction on sprayer airflow. Comput. Electron. Agric. 71, 128–136.

Endalew, A.M., Debaer, C., Rutten, N., Vercammen, J., Delele, M.A., Ramon, H., Nicolaï, B.M., Verboven, P., 2010b. A new integrated CFD modelling approach towards air-assisted orchard spraying—Part II: validation for different sprayer types. Comput. Electron. Agric. 71, 137–147.

Endalew, A.M., Debaer, C., Rutten, N., Vercammen, J., Delele, M.A., Ramon, H., Nicolaï, B.M., Verboven, P., 2010c. Modelling pesticide flow and deposition from air-assisted orchard spraying in orchards: a new integrated CFD approach. Agric. For. Meteorol. 150, 1383–1392.

Fatnassi, H., Boulard, T., Poncet, C., Chave, M., 2006. Optimisation of greenhouse insect screening with computational fluid dynamics. Biosyst. Eng. 93 (3), 301–312.

Fidaros, D.K., Baxevanou, C.A., Bartzanas, T., Kittas, C., 2010. Numerical simulation of thermal behavior of a ventilated arc greenhouse during a solar day. Renew. Energy 35, 1380–1386.

Fountas, S., Wulfsohn, D., Blackmore, B.S., Jacobsen, H.L., Pedersen, S.M., 2006. A model of decision-making and information flows for information-intensive agriculture. Agric. Syst. 87 (2), 192–210.

Franco, A., Valera, D.L., Pena, A., Perez, A.M., 2011. Aerodynamics analysis and CFD simulation of several cellulose evaporative cooling pads used in Mediterranean greenhouse. Comput. Electron. Agric. 76, 218–230.

Gebrehiwot, M.G., De Baerdemaeker, J., Baelmans, M., 2010. Effect of a cross-flow opening on the performance of a centrifugal fan in a combine harvester: computational and experimental study. Biosyst. Eng. 105 (2), 247–256.

Ghezzehei, T.A., Or, D., 2001. Rheological properties of wet soils and clays under steady and oscillatory stresses. Soil. Sci. Soc. Am. J. 65, 624–637.

Holterman, H.J., van de Zande, J.C., Porskamp, H.A.J., Huijsmans, J.F.M., 1997. Modelling spray drift from boom sprayers. Comput. Electron. Agric. 19, 1–22.

Hong, S.-W., Lee, I.-B., Hwang, H.-S., Seo, I.-H., Bitog, J.P., Yoo, J.-I., Kim, K.-S., Lee, S.-H., Kim, K.-W., Yoon, N.-K., 2008. Numerical simulation of ventilation efficiencies of naturally ventilated multi-span greenhouses in Korea. Trans. ASABE 51 (4), 1417–1432.

Hoxey, R.P., Robertson, A.P., Basara, B., Younis, B.A., 1993. Geometric parameters that affects wind loads on low-rise buildings: full-scale and CFD experiments. J. Wind Eng. Ind. Aerod. 50, 243–252.

Jimenez-Hornero, F.K., Gutierrez de Rave, E., Hidalgo, R., Giraldez, J.V., 2005. Numerical study of the natural airflow in greenhouses using a two-dimensional Lattice model. Biosyst. Eng. 91 (2), 219–228.

Jonckheere, M.R.M., 1997. Cleaning Means for an Agricultural Harvesting Machine. United States Patent, Number 5, 624-315.

Kacira, M., Short, T.H., Stowell, R.R., 1998. A CFD evaluation of naturally ventilated, multi-span, sawtooth greenhouses. Trans. ASAE 41 (3), 833–836.

Kacira, M., Sase, S., Okushima, L., 2004a. Effects of side vents and span numbers on wind-induced natural ventilation of a gothic multi-span greenhouse. J. Agric. Res. Quart. 38 (4), 227–233.

Kacira, M., Sase, S., Okushima, L., 2004b. Optimization of vent configuration by evaluating greenhouse and plant canopy ventilation rates under wind-induced ventilation. Trans. ASABE 47 (6), 2059–2067.

Karmakar, S., 2005. Numerical Modeling of Soil Flow and Pressure Distribution on a Simple Tillage Tool Using Computational Fluid Dynamics. PhD Thesis. University of Saskatchewan, Saskatoon, Canada, p. 220.

Karmakar, S., Kushwaha, R.L., 2005. Simulation of soil deformation around a tillage tool using computational fluid dynamics. Trans. ASAE 48, 923–932.

Karmakar, S., Kushwaha, R.L., 2007. Development and laboratory evaluation of a rheometer for soil visco-plastic parameters. J. Terramech. 44, 197–204.

Karmakar, S., Kushwaha, R.L., Laguë, C., 2007. Numerical modelling of soil stress and pressure distribution on a flat tillage tool using computational fluid dynamics. Biosyst. Eng. 97, 407–414.

Karmakar, S., Ashrafizadeh, S.R., Kushwaha, R.L., 2009. Experimental validation of computational fluid dynamics modeling for narrow tillage tool draft. J. Terramech. 46, 277–283.

Keedwell, M.J., 1984. Rheology and Soil Mechanics. Elsevier Applied Science Publishers, p. 323.

Kim, K., Yoon, J., Kwon, H., Han, J., Son, J.E., Nam, S., Giacomelli, G., Lee, I., 2008. 3-D CFD analysis of relative humidity distribution in greenhouse with a fog cooling system and refrigerative dehumidifiers. Biosyst. Eng. 100, 245–255.

Kruckeberg, J., Hanna, M., Darr, M., Steward, B., 2011. Relative Accuracy of DRIFTSIM as a Real-time Spray Drift Predictor. ASABE Paper No. 1111491. ASABE, St. Joseph, Mich.

Lee, I.B., Short, T.H., 2000. Two-dimensional numerical simulation of natural ventilation in a multi-span greenhouse. Trans. ASAE 43 (3), 745–753.

Liu, Y., Hou, Z.M., 1985. Three-dimensional nonlinear finite element analysis of soil cutting by narrow blades. Proc. Int. Conf. Soil. Dyn. Auburn, AL 2, 338–347.

Majdoubi, H., Boulard, T., Fatnassi, H., Bouirden, L., 2009. Airflow and microclimate patterns in a one-hectare Canary type greenhouse: an experimental and CFD assisted study. Agric. For. Meteorol. 149, 1050–1062.

Miner, S.M., 2000. Evaluation of blade passage analysis using coarse grids. ASME J. Fluid. Eng. 122, 345–348.

Mistriotis, A., Bot, G.P.A., Picuno, P., Scarascia-Mugnozza, G., 1997. Analysis of the efficiency of greenhouse ventilation using computational fluid dynamics. Agric. For. Meteorol. 85, 217–228.

Molari, G., Benini, L., Ade, G., 2005. Design of a recycling tunnel sprayer using CFD simulations. Trans. ASAE 48, 463–468.

Molina-Aiz, F.D., Valera, D.L., Alvarez, A.J., 2004. Measurement and simulation of climate inside Almeria-type greenhouses using computational fluid dynamics. Agric. For. Meteorol. 125, 33–51.

Muggli, F.A., Holbein, P., Dupont, P., 2002. CFD calculation of a mixed flow pump characteristic from shutoff to maximum flow. ASME J. Fluid. Eng. 124, 798–802.

Norton, T., Sun, D.W., Grant, J., Fallon, R., Dodd, V., 2007. Applications of computational fluid dynamics (CFD) in the modelling and design of ventilation systems in the agricultural industry: a review. Bioresour. Technol. 98, 2386–2414.

Nuyttens, D., De Schampheleire, M., Baetens, K., Brusselman, E., Dekeyser, D., Verboven, P., 2011. Drift from field crop sprayers using an integrated approach: results of a five-year study. Trans. ASABE 54, 403–408.

Or, D., 1996. Wetting induces soil structural changes: the theory of liquid phase sintering. Water Resour. Res. 32, 3041–3049.

Pedersen, S.M., 2003. Precision Farming – Technology Assessment of variable rate Treatment in Cereals. Ph.D. Thesis. Technical University of Denmark, Lyngby, 343pp.

Perdigones, A., Garcia, J.L., Romero, A., Rodriguez, A., Luna, L., Raposo, C., de la Plaza, S., 2008. Cooling strategies for greenhouses in summer: control of fogging by pulse width modulation. Biosyst. Eng. 99, 573–586.

Peters, L.W., 1995. Two Outlet Cleaning Fan. United States Patent, Number 5387154.

Plouffe, C., Laguë, C., Tessier, S., Richard, M.J., McLaughlin, N.B., 1999. Mouldboard plow performance in a clay soil: simulations and experiment. Trans. ASAE 42 (6), 1531–1539.

Ponpesh, P.O., Ken Giles, D., 2008. Modeling Turbulent Flow for Design of Almond Harvesters with Low Power Demand and Reduced Emission of Particulate Material, ASABE Paper Number: 084016, ASABE Annual Int. Meeting, Rhode Island Convention Center, Providence, Rhode Island (June 29–July 2).

Pretot, S.B., Zeghmati, G., Palec, Le., 2000. Theoretical and experimental study of natural convection on a horizontal plate. Appl. Therm. Eng. 20, 873–891.

Reichard, D.L., Zhu, H., Fox, R.D., Brazee, R.D., 1992a. Wind tunnel evaluation of a computer program to model spray drift. Trans. ASAE 35, 755–758.

Reichard, D.L., Zhu, H., Fox, R.D., Brazee, R.D., 1992b. Computer simulation of variables that influence spray drift. Trans. ASAE 35, 1401–1407.

Sapounas, A., Nikita, C., Bartzanas, T., Kittas, C., 2007. Aspects of CFD modeling of fan and pad evaporative cooling system in greenhouses. In: 2nd PALENC Conf. and 28th AIVC Conf. on Building Low Energy Cooling and Advanced Ventilation Technologies in the 21st Century, Crete Island, Greece.

Schaldach, G., Berger, L., Razilov, I., Berndt, H., 2000. Computer Simulation for Fundamental Studies and Optimisation of ICP Spray Chambers. Current Research Reports. ISAS (Institute of Spectrochemistry and Applied Spectroscopy), Berlin, Germany.

Shklyar, A., Arbel, A., 2004. Numerical model of the three-dimensional isothermal flow patterns and mass fluxes in a pitched-roof greenhouse. J. Wind Eng. Ind. Aerod. 92, 1039–1059.

Sidahmed, M.M., Brown, R.B., 2001. Simulation of spray dispersal and deposition from a forestry airblast sprayer. Part I: air jet model. Trans. ASABE 44, 5–10.

Streicher, E.A., Stroshine, R.L., Krutz, G.W., Hinkle, C.N., 1986. Cleaning shoe air velocities in combine harvesting of wheat. Trans. ASAE 29 (4), 923–928.

Swinton, S.M., Lowenberg-DeBoer, J., 1998. Evaluating the profitability of site-specific farming. J. Prod. Agric. 11 (4), 439–446.

Tadj, N., Bartzanas, T., Fidaros, D., Draoui, B., Kittas, C., 2010. Influence of heating system on greenhouse microclimate distribution. Trans. ASABE 53 (1), 225–238.

Teitel, M., 2010. Using computational fluid dynamics simulations to determine pressure drops on woven screens. Biosyst. Eng. 105, 172–179.

Teitel, M., Ziskind, G., Liran, O., Dubousky, V., Letan, R., 2008. Effect of wind direction on greenhouse ventilation rate, airflow patterns and temperature distributions. Biosyst. Eng. 101, 351–369.

Thakur, S., Lin, W., Wright, J., 2002. Prediction of flow in centrifugal blower using quasi steady rotor-stator models. J. Eng. Mech. 128, 1039–1049.

Thirion, F., 2002. Flow Simulation in a Sludge Spreader. Communication 02-PM-027 Presented at Ag-Eng 2002, Budapest, 8p.

Tong, G., Christopher, D.M., Li, B., 2009. Numerical modeling of temperature variations in a Chinese solar greenhouse. Comput. Electron. Agric. 68, 129–139.

Tsay, J., Ozkan, H.E., Brazee, R.D., Fox, R.D., 2002a. CFD simulation of moving spray shields. Trans. ASAE 45 (1), 21–26.

Tsay, J., Fox, R.D., Ozkan, H.E., Brazee, R.D., Derksen, R.C., 2002b. Evaluation of a pneumatic-shielded spraying system by CFD simulation. Trans. ASAE 45 (1), 47–54.

Ucar, T., Fox, R.D., Ozkan, H.E., Brazee, R.D., 2001. Simulation of jet agitation in sprayer tanks: comparison of predicted and measured water velocities. Trans. ASAE 44, 223–230.

Vyalov, S.S., 1986. Rheological Fundamentals of Soil Mechanics. Elsevier Science Publishing Co. Inc., Amsterdam, The Netherlands.

Wang, S., Boulard, T., 2000. Measurement and prediction of solar radiation distribution in full-scale greenhouse tunnels. Agronomie 20, 41–50.

Wang, J., Gee-Clough, D., 1991. Deformation and failure in wet clay soil. II. Simulation of tine soil cutting. In: Proc. IAMC Conf., Beijing, China, Session 2, pp. 219–226.

Weiner, K.L., Parkin, C.S., 1993. The use of computational fluid dynamic code for modelling spray from a mistblower. J. Agric. Eng. Res. 55, 313–324.

Wilcox, D.C., 2006. Turbulence Modeling for CFD, DCW Industries, third ed.

Yedidiah, S., 2008. A study in the use of CFD in the design of centrifugal pump. Eng. Appl. Comput. Fluid Mech. 2 (3), 331–343.

Yong, R.N., Hanna, A.W., 1977. Finite element analysis of plane soil cutting. J. Terramech. 14, 103–125.

Zhang, M.J., Pomfret, M.J., Wong, C.M., 1996. Three-dimensional viscous flow simulation in a backswept centrifugal impeller at the design point. Comput. Fluid. 25 (5), 497–507.

Zhang, Z., Liu, H., Zhu, S.P., Zhao, F., 2006. Application of CFD in ship engineering design practice and ship hydrodynamics. J. Hydrodyn. 18, 315–322.

Zhu, H., Reichard, D.L., Fox, R.D., Brazee, R.D., Ozkan, H.E., 1994. Simulation of drift of discrete sizes of water droplets from field sprayers. Trans. ASAE 37, 1401–1407.

Zhu, H., Reichard, D.L., Fox, R.D., Ozkan, H.E., Brazee, R.D., 1995. DRIFTSIM, a program to estimate drift distances of spray droplets. Appl. Eng. Agric. 11, 365–369.

Zhu, H., Richard, D.L., Fox, R.D., Brazee, R.D., Ozkan, H.E., 1996. Collection efficiency of spray droplets on vertical targets. Trans. ASAE 39, 415–422.