Available online at www.sciencedirect.com

ScienceDirect

Bioresource Technology 98 (2007) 2386–2414

BIORESOURCE TECHNOLOGY

Review

Applications of computational fluid dynamics (CFD) in the modelling and design of ventilation systems in the agricultural industry: A review

Tomás Norton [a,b], Da-Wen Sun [a,*], Jim Grant [b], Richard Fallon [c], Vincent Dodd [a]

[a] Food Refrigeration and Computerised Food Technology Research (FRCFT) Group, Biosystems Engineering, National University of Ireland, University College Dublin, Earlsfort Terrace, Dublin 2, Ireland

[b] Teagasc Agricultural Research Centre, Kinsealy, Malahide Road, Dublin 17, Ireland

[c] Teagasc Agricultural Research Centre, Grange, Dunsany, Co. Meath, Ireland

Received 1 August 2006; received in revised form 10 November 2006; accepted 11 November 2006
Available online 5 January 2007

Abstract

The application of computational fluid dynamics (CFD) in the agricultural industry is becoming ever more important. Over the years, the versatility, accuracy and user-friendliness offered by CFD has led to its increased take-up by the agricultural engineering community. Now CFD is regularly employed to solve environmental problems of greenhouses and animal production facilities. However, due to a combination of increased computer efficacy and advanced numerical techniques, the realism of these simulations has only been enhanced in recent years. This study provides a state-of-the-art review of CFD, its current applications in the design of ventilation systems for agricultural production systems, and the outstanding challenging issues that confront CFD modellers. The current status of greenhouse CFD modelling was found to be at a higher standard than that of animal housing, owing to the incorporation of user-defined routines that simulate crop biological responses as a function of local environmental conditions. Nevertheless, the most recent animal housing simulations have addressed this issue and in turn have become more physically realistic.

Keywords: Airflow; Temperature; Animal house; Greenhouse; Simulation

1. Introduction

The ability to maintain desired environmental conditions in intensive production systems is dependant on the design and performance of the ventilation system. Convective heat and mass transfers dominate the exchange processes in ventilated structures (Roy et al., 2002). As a consequence, the indoor environmental parameters such as temperature, pollution and humidity are governed by airflow patterns. These airflow patterns form the essential link between the outdoor environment and the buildings microclimate; thus an understanding of the principles of air motion is necessary in order to provide the correct quantities of air and the proper distribution patterns to meet the needs of the application.

Many studies have elucidated the benefits associated with the uniform distribution of airflow in animal and crop production systems (Boulard et al., 2002; Gebremedhin and Wu, 2005). In animal production, control over airflow is needed to remove humidity and manure gases, and to provide shelter from low temperatures, rain and radiation extremes. The result of proper housing is a healthier and more productive environment for the animals and for the people working in the building (Spoolder et al., 2000). This makes the knowledge of principles governing the distribution of the indoor climatic variables necessary when designing production systems or optimising their performance (Kavanagh, 2003).

* Corresponding author. Tel.: +353 17165528; fax: +353 14752119.
E-mail address: dawen.sun@ucd.ie (D.-W. Sun).
URLs: www.ucd.ie/refrig, www.ucd.ie/sun (D.-W. Sun).

0960-8524/$ - see front matter © 2006 Elsevier Ltd. All rights reserved.
doi:10.1016/j.biortech.2006.11.025

Nomenclature

$\widehat{A}$	frontal area density
C	water concentration ($\mathrm{kg\,kg^{-1}}$)
C_a	specific heat capacity ($\mathrm{W\,kg^{-1}\,K^{-1}}$)
C_D	drag coefficient
C_F	Forchheimer drag coefficient ($\mathrm{m^{-1}}$)
C_μ	a constant fitting parameter
f_i	momentum source ($\mathrm{N\,m^{-3}}$)
fo	solution function
F_s	factor of safety
g	acceleration due to gravity ($\mathrm{m\,s^{-2}}$)
K	Darcy permeability ($\mathrm{m^2}$)
k	turbulent kinetic energy ($\mathrm{m^2\,s^{-2}}$)
K_v	von Karman constant
k_r	resistance coefficient
p	pressure (Pa)
p_o	formal order of accuracy of the convection scheme
R	gas constant ($\mathrm{J\,kmol^{-1}\,K^{-1}}$)
f	grid refinement ratio
s_T	thermal sink or source ($\mathrm{W\,m^{-3}}$)
T	temperature (K)
t	time (s)
U	inlet velocity
u	velocity component ($\mathrm{m\,s^{-1}}$)
u_*	frictional velocity ($\mathrm{m\,s^{-1}}$)
W_a	Molecular weight of air ($\mathrm{kg\,kmol^{-1}}$)
x	Cartesian coordinates (m)
z	vertical distance from ground
z_0	friction length in m

Greek letters

ρ	density ($\mathrm{kg\,m^{-3}}$)
μ	dynamic viscosity ($\mathrm{kg\,m^{-1}\,s^{-1}}$)
δ	Kroneckor delta
β	thermal expansion coefficient ($\mathrm{K^{-1}}$)
λ	thermal conductivity ($\mathrm{W\,m^{-1}\,K^{-1}}$)
σ	Prandtl number for enthalpy
ε	turbulent dissipation rate ($\mathrm{m^2\,s^{-3}}$)
θ	local mean age of air (s)

Subscripts

i, j, k	Cartesian coordinate index
c	water concentration
ref	reference
p_{at}	at atmospheric pressure
z	position z at inlet
1, 2	level of grid refinement

Crop production requires good control over the greenhouse environment; the production system needs to operate smoothly in the temperature range characteristic of normal plant growth and development for the crop during the growing season. Quantitative understanding of a microclimate can help growers to optimise fertilisation and irrigation systems and solve environmental problems, such as, the over-consumption of irrigation water and the loss of nitrogen to the environment (Boulard and Wang, 2002).

Due to the complexity of the phenomena involved in indoor production systems, the amount of information required to fully quantify the environmental variables is dependant both on the physics involved and the level of precision associated with the analysis tools. Although physical experimentation allows for accurate environmental measurements without the need for modelling assumptions, a comprehensive analysis would not only necessitate expensive equipment, but would also require large amounts of time (Cook, 1990). Moreover, as in the case of wind tunnel modelling, measurements may not always be representative of the full-scale phenomena, e.g. poor application of scaling laws. Numerical modelling techniques such as CFD can offer an effective means of accurately quantifying the indoor climatic variables of ventilated buildings under various design conditions within a virtual environment. Thus the amount of physical experimentation can be reduced considerably, although, as of yet, not eliminated.

Computational fluid dynamics is a simulation technique that can efficiently develop both spatial and temporal field solutions of fluid pressure, temperature and velocity, and has proven its effectiveness in system design and optimisation within the chemical, aerospace, and hydrodynamic industries. The ubiquitous nature of fluids and their influence on system performance has meant that there has been widespread take-up of CFD by many other disciplines. In the ventilation of buildings, engineers have turned to CFD to model airflow processes and accomplish comprehensive system evaluation and design (Sorensen and Nielsen, 2003). Such has been the interest associated with the unique attributes afforded by CFD that the agricultural engineering community have recently embraced its power in predicting the phenomena occurring in many types of systems. This is evidenced by the increase in peer reviewed papers of CFD applications in recent years (Fig. 1).

Although there are many CFD studies of ventilation in animal production systems, none provide a comprehensive review detailing the current state-of-the-art in research and development. Furthermore, the technical advancements seen both in computer efficacy and in the numerical description of air motion have warranted an updated review of CFD and its use in the advancement of greenhouse technology. This review, therefore, aims to address all these issues as well as to highlight the recent developments in physical models and numerical techniques within a typical commercial CFD workbench. In order to stress

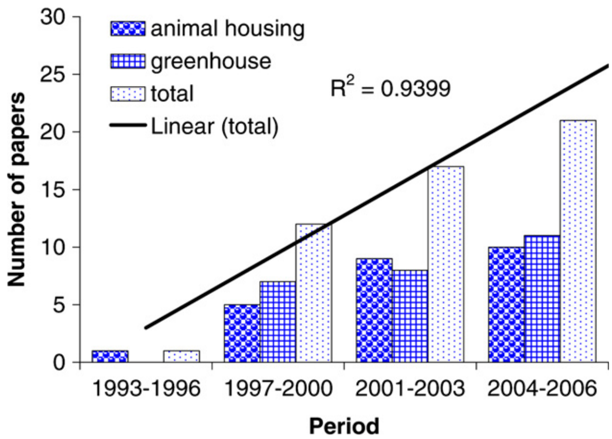

Fig. 1. The number of published peer-reviewed publications of CFD applied to the ventilation of agricultural buildings.

the essential need for realism in CFD, particular attention will be placed on the challenging issues that need to be addressed by CFD modellers, as well as modelling techniques that can be used to enhance the accuracy of CFD solutions.

2. Methods of ventilating agricultural buildings

Ventilation systems may be grouped into natural or mechanical ventilation systems according to the type of driving force employed. In mechanical systems electrically driven fans are used to drive airflow through the building. Mechanical ventilation offers a direct method of modifying the indoor environment by altering the velocity and psychometric properties of the incoming air jet, and has therefore raised the expectations for precise conditions in all weather extremes. Yet despite this being the most effective and commonly used technique, particularly for animals and crops with a narrow productive temperature range, the heavy reliance of mechanical ventilation systems on energy has meant that natural ventilation is now becoming more frequently investigated by the agricultural engineering community.

Natural ventilation is driven by two distinct mechanisms. The first driving mechanism is caused by thermal buoyancy, the so-called stack effect, which is dependent on the heating caused by incoming convective and radiative fluxes or by the metabolism of the occupant. The second mechanism, wind driven ventilation, is caused by the wind exerting pressure variations over the building envelope, and thus forcing airflow across ventilation openings. Under many practical situations natural ventilation can be designed based on strong physical arguments taking into account these two mechanisms (Montero et al., 2001). Moreover, natural ventilation can also be enhanced by adding some degree of automation to effect environmental control, the success of which is underpinned by the physics used to design the system (Hoff, 2004).

3. Mathematical modelling of agricultural buildings

3.1. The advantages afforded by mathematical models

As over many years agricultural buildings have become tailored for specialised production, the need to maintain control over the indoor climate so that the requirements of the production system can be continuously met has become ever more important. In essence it is necessary to understand the interaction of all climatic variables, alongside their contribution to the occupant's heat exchange with its surroundings in order to predict the perceived micro-climate.

Because the thermal environment of a building is intricately linked with ambient conditions, optimising and controlling ventilation systems are not elementary problems and may require extensive experimentation (Boulard et al., 2002; Zhang et al., 2000). Sophisticated techniques that allow efficient and accurate description of the dominant phenomena are often required. The mathematical modelling of the interaction between the indoor production system, the ventilation system and the outdoor environment is thus a necessary requisite. The clear advantages of developing mathematical models for agricultural buildings are as follows (adapted from Schauberger et al., 2000):

1. They can quantify the essential needs of the occupant in all parts of ventilated buildings and consequently contribute to the development of optimisation strategies aimed at increasing production performance.
2. They can be incorporated as part of indoor climate control, as well as to check the design values of the ventilation system.
3. They can combine the spatial and temporal dependant parameters that affect release of all pollutants within the building to efficiently calculate time-varying concentrations within, and emissions from production systems.
4. They can quantify different housing systems with alternative manure management techniques to achieve a fully integrated control of pollution.

Therefore, agricultural production systems, which are amenable to description through the physical laws of fluid motion, heat and mass transfer processes, can be effectively modelled so that the benefits of pre-design evaluation, design and optimisation strategies can be realised.

3.2. The macro-model versus micro-model description of ventilation

The Navier–Stokes equations, outlined below, represent the dynamic behaviour of a flow regime, both spatially and temporally. These equations constitute a micro-model of fluid motion, and require time consuming iterative techniques to solve them. As discussed by Roy et al. (2002), the efficient quantification of ventilation in agricultural buildings can be carried with a macro-model based on Bernoulli's equation. This requires some qualifying assumptions so that the model matches the spatial domain of interest, e.g. the flow is inviscid, steady, irrotational, hydro-static and gravity is the only acting external force. Consequently, much information on the flow is lost, including details on the spatial distribution of the airflow patterns, which can lead to inaccuracies in the calculated airflow variables (Boulard et al., 2002).

The tacit use of the above assumptions is the main cause for concern in macro-modelling, as they amount to a description of airflow that violates one of the fundamental laws of fluid mechanics; the conservation of kinetic energy. The result is that kinetic energy is falsely created or destroyed in calculations, depending on the specific application. This has been termed the disposable kinetic energy and is a function of the geometry characteristics of an opening, and depends on whether or not the flow is turbulent (Roy et al., 2002). The macro-model deals with this as a pressure head loss coefficient, often referred to as ventilation coefficient (Verlinde et al., 1998), which is not a physical attribute of the flow regime but instead acts to obviate the detail required had the full nature of the flow been considered in the model. The value of this coefficient is dependant on the flow regime, and must be systematically determined by fitting the model to experimental data. An approximation for the value of this coefficient obviously limits the accuracy of the developed solution (Axley and Chung, 2005). Therefore, macro-modelling techniques require a comprehensive experimental approach in their validation (Kittas et al., 1997).

To overcome the inherent limitation of the classic macro-model, a recent modelling technique has been derived from a theoretical approach that explicitly conserves kinetic energy (Axley and Chung, 2005; Kato, 2004). The resulting equation system is tied to the mathematical relations used in the determination of frictional loss factors for a large number of flow components and devices, thereby precluding the need for semi-empirical pressure drop coefficients. Initial studies have indicated that these models may perform better than the classical macro-model in some situations, and that secondary flows induced by primary airflows can be properly predicted (Axley and Chung, 2005). Notwithstanding this, a comprehensive spatial analysis of airflow cannot be carried out with this method.

In contrast to the macro-model, the micro-model of fluid motion does not need any experimental determination or tuning of parameters. The micro-model establishes the changes in momentum of fluid particles as a product of changes in pressure and dissipative viscous forces acting inside the fluid. Thus it can determine all the interactions between fluid and obstruction boundaries. Provided the flow regime and boundary conditions of the problem are correctly described, there should be few limiting physical assumptions that can impinge on the accuracy of the micro-model. Micro-modelling forms the basis upon which CFD is based; the application of CFD in ventilation system design is described below.

4. Principles of CFD

Computational fluid dynamics is a sophisticated design and analysis tool that uses computers to simulate fluid flow, heat and mass transfer, phase change, chemical reaction, mechanical movement, and solid and fluid interaction. The technique enables a computational model of a physical system to be studied under many different design constraints. The quality of a CFD study is a function of not only the physics available in the software to model the system, but also the understanding that the CFD modeller has of both the numerics and physics contained in the software package. If used correctly CFD can provide an understanding of the physics of a flow system in detail, and does so through non-intrusive flow, thermal and concentration field predictions. The following discusses the main physical models upon which CFD software is based and outlines the general methodology used to carry out a comprehensive study with commercial software.

4.1. Governing equations

The governing equations of fluid flow and heat transfer can be considered as mathematical formulations of the conservation laws that govern all fluid flow, heat transfer and associated phenomena. These conservation laws describe the rate of change of a desired fluid property as a function of external forces and can be written as:

Continuity equation: the mass flows entering a fluid element must balance exactly with those leaving.

$$\frac{\partial \rho}{\partial t} + \frac{\partial}{\partial x_i}(\rho u_j) = 0 \tag{1}$$

Conservation of momentum (Newtons second law): the sum of the external forces acting on the fluid particle is equal to its rate of change of linear momentum.

$$\frac{\partial}{\partial t}(\rho u_i) + \frac{\partial}{\partial x_j}(\rho u_i u_j) = \frac{\partial}{\partial x_j}\left[-p\delta_{ij} + \mu\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i}\right)\right] + \rho g_i \tag{2}$$

Conservation of energy (the first law of thermodynamics): the rate of change of energy of a fluid particle is equal to the heat addition and the work done on the particle.

$$\frac{\partial}{\partial t}(\rho C_{\mathrm{a}} T) + \frac{\partial}{\partial x_j}(\rho u_j C_{\mathrm{a}} T) - \frac{\partial}{\partial x_j}\left(\lambda \frac{\partial T}{\partial x_j}\right) = s_{\mathrm{T}} \tag{3}$$

CFD enforces these conservation laws over a discretised flow domain in order to compute the systematic changes in mass, momentum and energy as fluid crosses the boundaries of each discrete region (Versteeg and Malalsekeera, 1995).

4.2. Numerical analysis

CFD codes developers have a choice of many different numerical techniques to discretise the modelled fluid continuum. The most important of these include finite difference, finite elements and finite volumes.

Due to difficulties in the handling of complex geometry, finite difference techniques are rarely used in engineering flows. Finite elements have been used to successfully model the wind and building interaction in agricultural building design (Mistriotis and Briassoulis, 2002). However, because of the difficulties involved in the programming and implementation of this technique not many commercial finite element packages exist.

The ease in the understanding, programming and versatility of finite volumes has meant that they are now the most commonly used numerical techniques in CFD code development, and as a consequence are the most commonly used to form design solutions for ventilated agricultural structures (Fatnassi et al., 2006).

4.3. Commercial CFD codes

Over the last 2 decades there has been enormous development of commercial CFD codes to meet the sophisticated modelling requirements of many research fields. The typical features of a CFD code required by a ventilation engineer include the ability to model flow dependant properties, implement user defined functions and model flow through porous media (Boulard and Wang, 2002). Various meshing options are also beneficial for modelling flow within and around buildings of complex geometry. The performance of commercial CFD codes at different applications has been notably varied, as many have emerged from highly specialised research fields (Kopyt and Gwarek, 2004). Therefore, functional considerations of a code should be taken into account before selection. Among the most routinely used commercial codes include ANSYS CFX, ANSYS FLUENT, PHOENICS and CFD2000. These codes incorporate at least all the aforementioned functionalities, employ graphical user interfaces, and support Windows, UNIX and Linux platforms.

4.4. Performing a CFD analysis with commercial software

Undertaking a CFD study demands the use of three predefined environments within the software, with each environment representing an equally important section of the modelling process. The following provides an introduction to the different modules comprising a CFD software package.

4.4.1. Pre-processing

The pre-processor of CFD software holds all the mathematical statements attributable to the potential success of a modelling exercise and therefore embodies the most important phase of model definition. The main tasks facing a user in the pre-processing environment include problem consideration, geometry creation or import, mesh development, physical property set-up, and the implementation of solving techniques and parameters.

4.4.2. Solving

The solver environment, within all commercial CFD software packages, organises the mathematical input from the pre-processor into numerical arrays and solves them by iterative methods. Iterative methods are commonly used to solve a whole set of discretised equations so that they may be applied to a single dependent variable. Segregated solving techniques such as Semi-Implicit Method for Pressure-Linked Equations (SIMPLE), used by most commercial CFD software packages, determine the pressure field indirectly by closing the discretised momentum equations with the continuity equations in a sequential manner (Patankar and Spalding, 1972). However, as the number of cells increase the elliptic nature of the pressure field becomes more profound and the convergence rate is inhibited. This has led to the development of multigrid techniques that compute velocity and pressure corrections in a simultaneous fashion, thus enhancing convergence rates (Ferry, 2002).

4.4.3. Post-processing

The post-processing environment allows the user to visualise and scrutinise the resulting field solution. Contour, vector and line plots enhance interpretation of results and are being progressively fortified in commercial software packages. Some packages also allow the export of field data to external modelling programs so that it can be processed further. Fig. 2 illustrates the visualisation techniques that can provide sufficient information to move forward in the design process.

4.5. Representing the building geometry in CFD models

4.5.1. Model dimensionality in mechanical ventilation

When CFD was first used to model airflow in mechanically ventilated rooms, it was assumed that symmetric rooms with two-dimensional boundary conditions had a two-dimensional airflow pattern (Gosman et al., 1980; Timmons, 1984). However, Bjerg et al. (1999) found that this assumption was dependant on the ratio of room width to height and room length to height, with ratios greater than one and three respectively exhibiting three-dimensional effects. Although in some cases, even when complying with these conditions, two-dimensional airflows in ventilated rooms were hard to achieve without the use of guiding plates (Zhang et al., 2000).

The studies of Bjerg et al. (1999) and Zhang et al. (2000) have confirmed that many two-dimensional CFD investigations of air motion in rooms should be viewed with caution unless there is solid proof that these patterns would occur in a physical situation. This also means that results from much more recent two-dimensional investigations

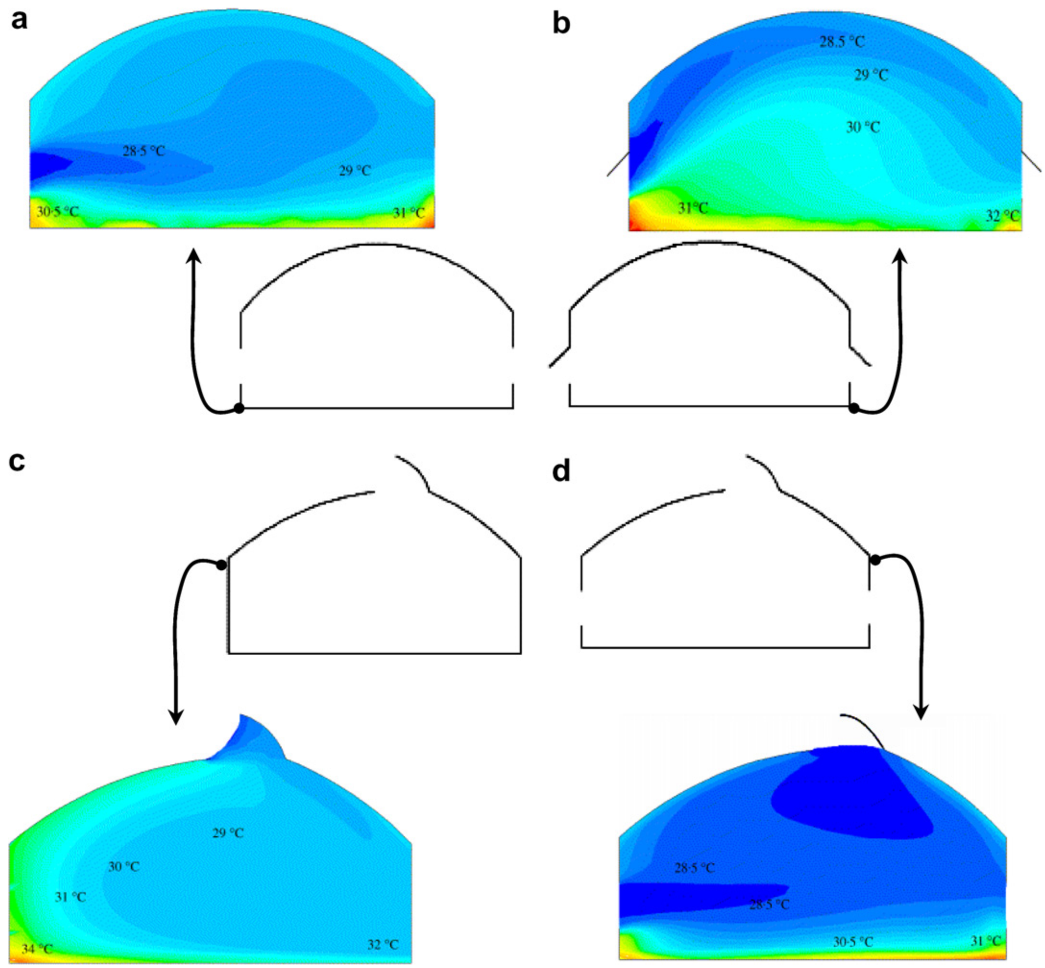

Fig. 2. Contour plots exhibiting the enhanced visualization abilities of computational fluid dynamics. Configuration (d) provides a more uniform temperature distribution throughout the building (Bartzanas et al., 2004).

can be considered as, at best indicative and at worst highly speculative (Mirade, 2003; Bartzanas et al., 2004).

4.5.2. Computational domain in natural ventilation

The representation of the model of a naturally ventilated building in a computational domain of a CFD study gives a number of possible approaches. For example a model may consist of:

1. Both indoor and outdoor regions of the naturally ventilated building.
2. Indoor and outdoor regions divided into sub-domains and solved for independently, with the solutions being interpolated and matched at the region interfaces.
3. Just the indoor environment.

The first approach offers the advantage of directly coupling both internal and external environments. Yet, there is a large amount of computational overhead due to the required meshing of flow impingement and development areas at the windward and leeward of the building respectively, which is heightened by the detailed meshing of the building interior. As a result computations may take a large amount of time to complete (Boulard et al., 2002). However, as discussed below, advances in unstructured meshing has made this process more efficient in recent years (Shklyar and Arbel, 2004).

The second method is not commonly encountered, although some large scale natural ventilation investigations have used this technique when the direct coupling of indoor and outdoor environment was unattainable due to the lack of sufficient computer resources (Jiang and Chen, 2002). The final method has been shown to be flawed in some cases with solutions often bearing no resemblance to the physical flow regime (Fracastoro and Perino, 1999).

5. Governing equations for describing the interaction between building, outdoor climate and indoor occupants

Alongside the Navier–Stokes equations account must also be taken of the additional processes that may influence the dynamics of a ventilation system. The governing equations may need to be fortified with additional physical models or assumptions to fully represent the physical situation. Important physical models commonly used in ventilation applications, which include turbulence and porous media models, and models that describe animal and crop responses to the environment are discussed below.

5.1. Turbulent air motion

Ventilating flows are usually associated with turbulent motion; primarily due to high flow rates and heat transfer interactions involved in the flow field. Whilst the Navier–Stokes equations can be solved directly for laminar flows, the current state of computational capability is unable to resolve the fluid motion in the Kolmogorov microscales associated with turbulent flow regimes (Friedrich et al., 2001). For room air motion this microscale is about 1 mm, which implies a turbulent motion with a broad spectrum of length scales. The simulation of all these motions would be very expensive (Etheridge and Sandberg, 1996). However, ventilation engineers are generally content with a statistical probability that environmental variables (such as velocity and temperature) will exhibit a particular value in order to undertake suitable design strategies. Such information is afforded by the Reynolds averaged Navier–Stokes equations (RANS), which determine the effect of turbulence on the mean flow field through time averaging. With this averaging, the stochastic properties of turbulent flow are essentially disregarded and six additional stresses (Reynolds stresses) result, which need to be modelled by a physically well-posed equation system to obtain closure that is consistent with the flow regime.

At the present time, there are many turbulence models available, and their prediction capability has undergone great improvement over the last number of years. However, it should be noted that none of the existing turbulence models are complete, i.e. their prediction performance is highly reliant on turbulent flow conditions and geometry. Without a complete turbulence model capable of predicting the average field of all turbulent flows, the present understanding of turbulence phenomena will hamper comprehensive quantification of indoor air motion. Some of the best performing turbulence models are discussed below.

5.1.1. Eddy viscosity models

The eddy viscosity hypothesis (Boussinesq relationship) states that an increase in turbulence can be represented by an increase in effective fluid viscosity, and that the Reynolds stresses are proportional to the mean velocity gradients via this viscosity (Ferziger and Peric, 2002). This hypothesis forms the foundation for many of today's most widely used turbulence models, ranging from simple models based on empirical relationships to variants of the sophisticated two-equation k–ε model, which describe eddy viscosity through turbulence production and destruction (Boulard et al., 2002). All these models have relative merits with respect to simulating ventilating flows.

The standard k–ε model remains the standard used in the modelling of agricultural buildings and applications of this model are found in recent literature (Bartzanas et al., 2004). Some studies have suggested that the model performed just as well as a Reynolds stress closure model (Bjerg et al., 1999; Gebremedhin and Wu, 2003). However, the studies did not provide sufficient substantiation of these suggestions. Other studies, as illustrated in Fig. 3, have commended complex turbulence models over the standard k–ε model even though the published comparisons did not exhibit any considerable disparity between predictions (Reichrath and Davies, 2002a; Shklyar and Arbel, 2004; Lee et al., 2005).

Certainly, there have been many situations where the standard k–ε model has failed to sufficiently represent the turbulent regime, with predictions often proving to be totally inaccurate (Moureh and Flick, 2005). The most pronounced and well documented inaccuracies occur when predicting impinging flows, flows under adverse pressure gradients and flows with strong streamline curvature. This is in part due to the Boussinesq theory being invalid for flows in which non-isotropic turbulence exists (Wright and Easom, 2003). In addition, the standard k–ε model contains many empirical constants that have long been known to have an adverse effect on prediction performance (Versteeg and Malalsekeera, 1995). Because the aforementioned flow regimes are commonly encountered in ventilation engineering it would seem intuitive that the standard k–ε model should not be used at all. Yet, due to its favourable convergence behaviour and reasonable accuracy the standard k–ε turbulence model is routinely employed.

In recent times engineers have turned to more complex eddy viscosity turbulence models like the two-scale k–ε turbulence models such as the Chen–Kim model (Chen and Kim, 1987) and the renormalisation group (RNG) k–ε model (Yakhot and Orszag, 1986) to model air movement within and around buildings. These models are not so reliant on empiricism and can account for anisotropy of strained flows. Wind engineering applications have highlighted the superiority of two-scale k–ε models in the prediction of impinging flows (Murakami, 1998; Richards and Hoxey, 2006). In a two-dimensional wind induced ventilation study Mistriotis et al. (1997a) showed that better qualitative agreement with experimentally observed flow patterns could be achieved with a two-scale k–ε turbulence model than with the standard k–ε model. Quantitative differences between these turbulence models were later found in three-dimensional investigation of wind induced ventilation by Roy and Boulard (2005). Brugger et al. (2005) also found notable differences in ventilation rates predicted by a two-scale k–ε model and the standard k–ε model. Unfortunately as evident from Fig. 4, the k–ε turbulence modelling of confined flows with adverse pressure gradients has not yielded positive results, and the two-scale k–ε models have also failed to predict the coanda effect in mechanically ventilated rooms.

All k–ε models variants seem to perform similarly when heat transfer is coupled with the flow field. Although the veracity of this statement has not been directly scrutinised by the agricultural engineering community, simulations of other indoor environments have shown similar agreement

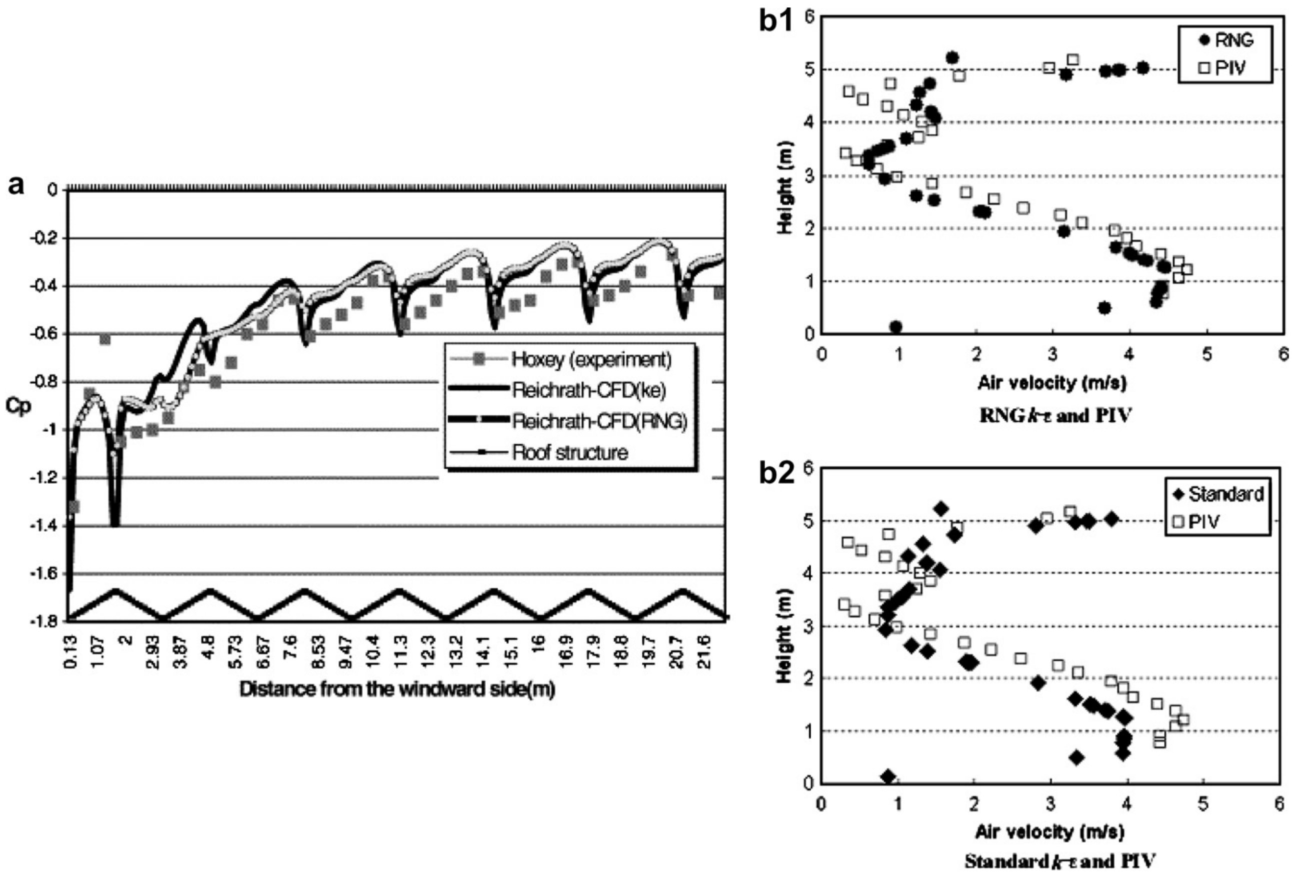

Fig. 3. Two plots illustrating the justification used, in two different studies, for choosing a specific turbulence model. In (a) pressure coefficients were used to validate the CFD model and k–ε RNG was chosen (Reichrath and Davies, 2002a). In (b1) and (b2) indoor velocities, determined by particle image velocitometry (PIV), were used to validate the CFD model and the k–ε RNG was chosen (Lee et al., 2005).

of all k–ε models with experimental data (Posner et al., 2003; Hoang et al., 2000; Rouaud and Havet, 2002). Recently, the SST (shear stress transport) model, which combines the k–ε and the k–ω models using a blending function, has found to predict airflow in good agreement with experimental data not only in the near wall region of the flow (Kuznik et al., 2006) but also in the free stream (Stamou and Katsiris, 2006). Thus, this model should be considered when buoyancy forces play a major role in driving the flow within a ventilated space.

5.1.2. Reynolds stress closure models

As stated above the Reynolds stress closure models (RSM) have exhibited far superior predictions for flows in confined rooms where adverse pressure gradients occur. These models have recently been shown to enhance the CFD predictions of primary and secondary flow patterns in empty-isothermal rooms (Moureh and Flick, 2005; Schalin and Nielsen, 2004), and in loaded rooms with heat transfer (Moureh et al., 2002). The added benefits of RSM models are offset by the extra computational time and memory required in solving the flow regime, alongside the difficulties in attaining good convergence behaviour (Gebremedhin and Wu, 2003). However with computer power increasing rapidly, RSM models may become utilised more frequently for steady-state confined room flows in the future (Schalin and Nielsen, 2004).

5.1.3. Large eddy simulation

Simulation methodologies such as large eddy simulation (LES) have also been addressed to enhance the prediction of turbulent flow and ventilation transfers. LES forms a solution given the fact that large turbulent eddies are highly anisotropic and dependant on both the mean velocity gradients and geometry of the flow domain. With the advent of more powerful computers LES now offers a way of alleviating the errors caused by the use of RANS turbulence models. LES is, as of yet, not suitable for providing efficient three-dimensional design solutions owing to the time involved in developing flow solutions (Sorensen and Nielsen, 2003). However, with the uncertainty currently surrounding the veracity of unsteady RANS solutions, LES computations offer a more accurate technique for solving transient flows. Recent ventilation applications have used LES successfully to predict the detailed two-dimensional flow-field around ventilated rooms (Sun et al., 2004a). LES has also been applied successfully in wind engineering applications (Murakami, 1998).

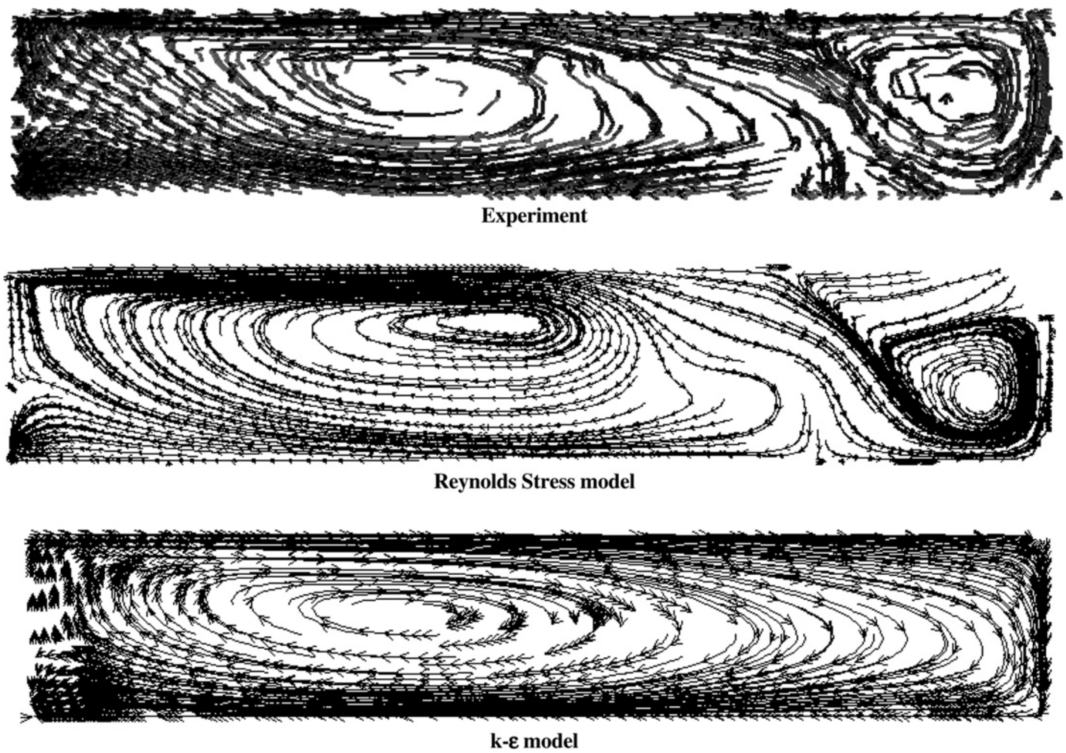

Fig. 4. Influence of the turbulence model on the flow pattern at the symmetry plane of a confined flow (Moureh and Flick, 2005).

5.2. Air flow coupled with heat and mass transfer

5.2.1. Modelling buoyancy

The equation of state relates the density of the air to thermodynamic state, i.e. its temperature and pressure. Climatic variables in housing facilities are a function of varying flow properties caused by the heating and cooling of air. There are two main methods of modelling the density variations that occur due to buoyancy. The first is the well known Boussinesq approximation (Ferziger and Peric, 2002). This has been used successfully in many housing applications (Boulard et al., 2002):

$$\rho = \rho_{\text{ref}}[1 - \beta(T - T_{\text{ref}})] \tag{4}$$

The assumptions involved in this approximation include (Etheridge and Sandberg, 1996):

(a) Density differentials in the flow are only required in the buoyancy term of the momentum equations.
(b) There is a linear relationship between temperature and density, with all other extensive fluid properties being constant.
(c) The temperature difference in the flow field is less than about 30 °C.

This relationship only considers dry air as the fluid medium, whereas it is conceivable in most climatic flows a mixture of dry air and moisture will be involved. Gan (1994) derived an extended version of the Boussinesq approximation, which admits a description of the density of moist air as a function of temperature and moisture concentration. With a reference density, temperature, and water concentration and by using Taylor's expansion theorem the density variation can be expressed as:

$$\rho = \rho_{\text{ref}}[\beta(T - T_{\text{ref}}) + \beta_{\text{c}}(C - C_{\text{ref}})] \tag{5}$$

where

$$\beta = -\frac{1}{\rho}\left(\frac{\partial\rho}{\partial T}\right)_{P_{\text{at}}}; \quad \beta_C = -\frac{1}{\rho}\left(\frac{\partial\rho}{\partial C}\right)_{P_{\text{at}}}$$

Despite the enhanced relation, Eq. (5) is not commonly used in the literature, even though some CFD software packages allow the front-end incorporation of this extension (Anon, 2006). The Boussinesq approximation is not sufficiently accurate at large temperature differentials (Ferziger and Peric, 2002). Therefore, another method of achieving the coupling of the temperature and velocity fields is necessary when the Boussinesq approximation is invalid. This is done by treating the air as an ideal gas and expressing the density difference by means of the ideal gas equation:

$$\rho = \frac{p_{\text{ref}} W_{\text{a}}}{RT} \tag{6}$$

This method can be considered as a weakly compressible formulation, which means that the density of the fluid is dependent on temperature and composition but not pressure. This relation is a more numerically complex method of representing buoyancy.

Whilst Eq. (6) may provide an accurate description of the density variations within the flow regime, it has been found to impact on convergence behaviour of CFD solutions (Foster et al., 2002).

5.2.2. Modelling crop heat and mass transfer

Heat and mass transfer are primary functions of all biological entities. Sources and sinks of heat, contaminants and water vapour are prevalent in the housed environment and level of detail to which they are modelled has been developed over recent years. Modelling of greenhouses microclimate has evolved from the determination of global crop canopy transfers prior to the CFD calculation (Mistriotis et al., 1997b) to the incorporation of user defined routines that calculate the effects of solar radiation and crop transpiration as a function of the local conditions (Boulard and Wang, 2002; Fatnassi et al., 2006). Boulard and Wang (2002) described an example of a routine used to compute crop transpiration.

The user routine was divided into two steps. The first step allowed for the computation of the indoor climate based on the net solar energy, which was assimilated as a heat source. The second step partitioned the solar energy into convective and latent fluxes depending on local conditions. This partition depended on the heat and water vapour exchanges (stomatal and aerodynamic) between the solid matrix of the porous medium and the air within each mesh of the crop canopy (represented by a porous medium). Such modelling techniques pave the way for more detailed predictions of the indoor environment as a function of crop biological response.

5.2.3. Modelling animal heat and mass transfer

Gebremedhin and Wu (2001, 2003) used CFD predictions to aid in the modelling of the thermal environment of housed cows. In their studies an external FORTRAN program was used to compute all the models that were needed to determine heat and mass transfer from a cows' body. The local velocity and temperature required for the FORTRAN program were computed in a CFD software package. More recently they have enhanced this model by combining a CFD computed flow field with a finite difference program to calculate the heat (including radiation) and mass transfer between cows' in a multi-occupant environment (Gebremedhin and Wu, 2005). However, the validity of these models has not been upheld by experimental results.

Unlike greenhouse engineering, where the crop canopy can be accurately modelled as porous media, the porous media assumption, as discussed below, is not necessarily an accurate representation of the housed entity in animal housing studies. This is because an animals body is basically a solid obstruction, i.e. no explicit air exchange occurs, as with a crop canopy. Thus, calculating the convective and latent heat fluxes using a porous media assumption similar to that employed by Boulard and Wang (2002) may result in incorrect solutions.

5.3. Flow resistance: porous media

Windbreaks have long been used in agriculture to prevent wind damage and improve microclimates by modifying wind momentum and turbulence distribution. Fig. 5 shows the effect of windbreaks on wind induced ventilation. They can also be used to channel flows in sheltered areas to enhance the ventilation performance of buildings.

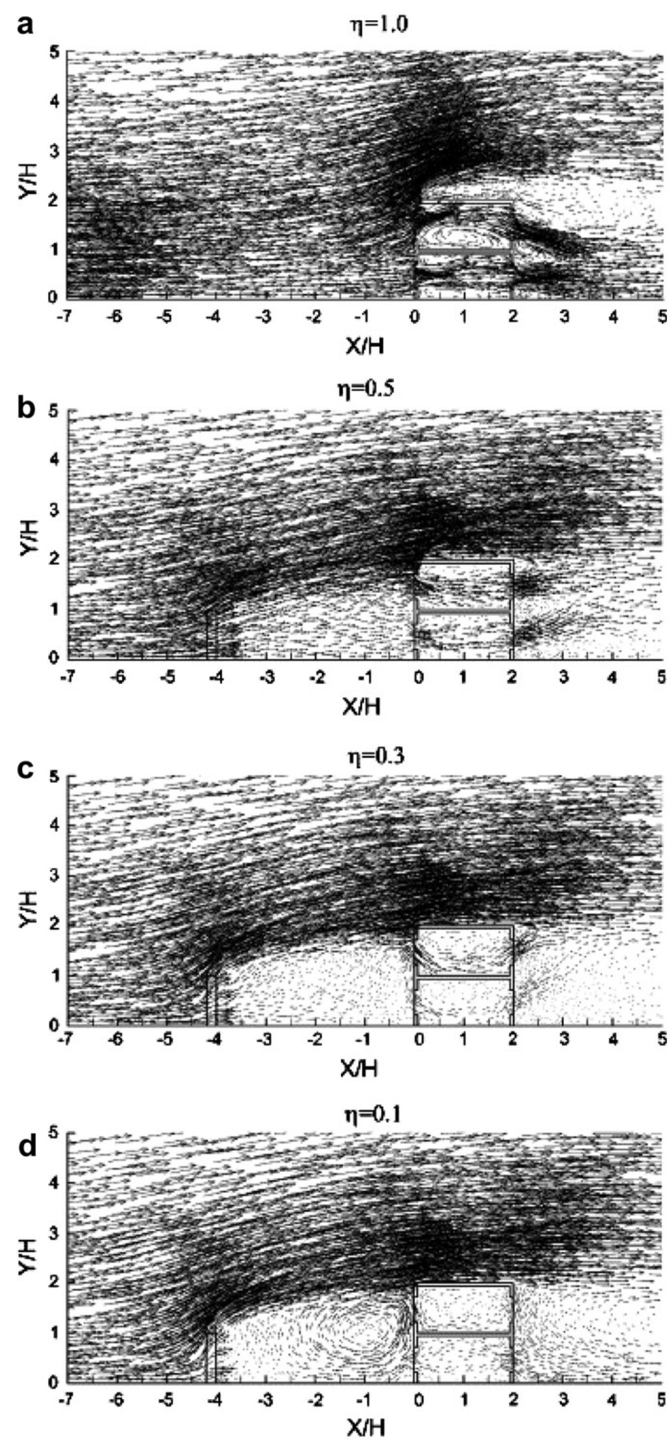

Fig. 5. Effect of windbreak porosity on wind induced natural ventilation, from most porous (a), to least porous (d) (Chang, 2006).

In greenhouses nets may be used to protect buildings from the entrance of strong winds and dust and insects, and their effective use has been classified both numerically and experimentally (Bartzanas et al., 2002; Katsoulas et al., 2006).

5.3.1. Porous media convention

The porous media convention, which relates air velocity, alignment with flow, and permeability to the pressure drop over an object, has often been used to represent the aerodynamic effect of windbreaks in numerical simulations (Boulard et al., 2002). As highlighted by Miguel et al. (1998) the linear relationship between pressure drop and volume averaged velocity caused by viscous drag (Darcy's law) is violated in high velocity flows, which are often encountered in the natural environment. A modification of this law has been provided by the Darcy–Forcheimer equation, which relates the drag force through a porous medium to a linear combination of flow velocity, i.e. the viscous resistance due to the obstacle boundaries, and the square of the flow velocity, i.e. the resistance due to inertial effects. This can be described as follows (Miguel et al., 1998):

$$\frac{\partial p}{\partial x} = -\frac{\mu}{K}\bar{u}_i + \rho\left(\frac{C_F}{\sqrt{K}}\right)\bar{u}_i|\bar{u}_i| \tag{7}$$

Eq. (7) represents a commonly used method to characterise the macroscopic airflow through a volume of material occupied by a solid matrix and air, also known as the porous media convention. Porous insect screens used in greenhouses are routinely represented by Eq. (7) in CFD studies. As highlighted by Verboven et al. (2006), Eq. (7) is strictly only valid for infinite barriers, whereas in most engineering applications, including ventilation system design, a finite barrier is under investigation. To represent this confinement effect sufficiently an additional viscous term should be added to Eq. (7). The pressure drop over the barrier can then be determined by:

$$\frac{\partial p}{\partial x_i} = -\frac{\mu}{K}\bar{u}_i + \rho\left(\frac{C_F}{\sqrt{K}}\right)\bar{u}_i|\bar{u}_i| + \mu_{\text{eff}}\nabla^2 u_i \tag{8}$$

Verboven et al. (2006) also showed that the importance of this term is exhibited in confinement flows $\leqslant 10$, where the term confinement flow means the ratio of the hydraulic length of the confined region to the effective obstruction diameter of the barrier. Therefore, judgement should be made prior to CFD computations as to whether Eq. (7) or Eq. (8) is appropriate.

5.3.2. The meteorological convention

The meteorological convention has also been used in CFD studies to represent the momentum deficit of airflow through objects with a large porosity matrix. In this case the inertial effects are parameterised via the drag over the area density of a porous entity such as a stacked objects or crop (Boulard et al., 2002; Mirade et al., 2006). This is carried out by letting $\bar{u}_i = 0$ in Eq. (7), i.e. by making the assumption that viscous drag (μ/K) is negligible when compared to the inertial effects posed by the windbreak. In addition, the term $C_F/\sqrt{K}$ in Eq. (7) must be replaced with $C_D\widehat{A}$ where C_D is the drag coefficient and $\widehat{A}$ is the frontal area density. This results in the following equation (Wilson, 1985):

$$\frac{\partial p}{\partial x_i} = -C_D\widehat{A}\bar{u}_i|\bar{u}_i| \tag{9}$$

Takle and Wang (1997) extended Eq. (9) for representing a barrier of finite thickness:

$$\frac{\partial p}{\partial x_i} = -C_D\widehat{A}\bar{u}_i\sqrt{\bar{u}_i^2 + \bar{u}_j^2 + \bar{u}_k^2} \tag{10}$$

As pointed out by Wilson and Mooney (1997) this representation of the mean drag force is logical for barriers with finite thickness, and is in principle superior to Wilson's (1985) model, i.e. Eq. (9).

5.3.3. Using a resistance coefficient

Wilson (1985) followed the meteorological convention to determine a momentum sink for ideal shelter flows where obtaining C_D and $\widehat{A}$ is not possible, i.e. by using a resistance coefficient (k_r):

$$\frac{\partial p}{\partial x_i} = -k_r\bar{u}_i|\bar{u}_i|\delta(x,0) \tag{11}$$

where δ (m^{-1}) localises the momentum movement to $x = 0$ (Wilson and Mooney, 1997). This model is applicable to the description of a neutrally stratified flow over an infinitesimally thin porous fence and the relationship between k_r and $C_D\widehat{A}$ is represented by:

$$k_r = \int_{\infty}^{\infty} C_D\widehat{A}\,dx = \sum_i (C_D\widehat{A})_i\Delta x_i \tag{12}$$

where k_r, can be determined from the porosity of the fence by an empirical relationship determined by Hoerner (1965):

$$k_r = \frac{1}{2}\left[\frac{3}{2K} - 1\right]^2 \tag{13}$$

The value of $C_D\widehat{A}$ can be determined by k_r/W, where W is the width of the barrier. These momentum sinks have performed reasonably well in numerical simulations with good agreement found between predictions and experiments (Takle and Wang, 1997).

5.3.4. Porous media assumption for solid obstructions

In livestock buildings the porous media assumption has been used to model the drag effect produced by superficial air velocity through internal partitions, slatted floors, porous ceilings and pen covers with parameters such as permeability and resistance coefficients being determined from experiments (Lee et al., 2004; Sun et al., 2002; van Wagenberg et al., 2004). Animals have also been simulated as a volume resistance causing a pressure drop described by the following equation (Svidt et al., 1998):

$$\frac{\partial p}{\partial x_i} = \frac{f}{2}\rho u_i^2 \tag{14}$$

where the loss coefficient f was defined based on the recommendations of Nielsen et al. (1997, 1998). It is worth noting that best agreement with experimental measurements was found when the volume was in its most solid state, i.e. f being about unity. Other studies have also used the porous media assumption to model hanging beef carcasses in a large chiller and stacked cheeses in a ripening room (Mirade et al., 2006). In these studies the meteorological convention was adopted. Similar to the greenhouse studies of Boulard and Wang (2002) and Bartzanas et al. (2004), the coefficients in the default porous media assumption of CFD software package were altered in order to represent the coefficients of Eq. (7). The resulting equation was applied as a momentum sink over the region of interest.

It is important to note that when computing the flow through a porous object the resistance coefficient will change depending on whether one solves for superficial or interstitial velocity. Superficial velocity is the ratio of volumetric flow rate to cross sectional area. Thus the volume porosity and area porosity vector do not need to be specified as their values are unity. Eqs. (7)–(14) are examples when the superficial velocity must be used. Interstitial velocity, which is namely a reduced area velocity, i.e. the ratio of superficial velocity to porosity, requires that both these parameters and thus the resistance coefficient must be changed accordingly.

5.4. The atmospheric boundary layer

5.4.1. Simulation with CFD

The rural terrain regularly contains a complex aggregate of hills, buildings, hedgerows, forests and crop canopies (i.e. roughness elements) that disturb the variables within the atmosphere. The dynamical effect of this complex rough surface arises via the transfer of momentum from the wind to the ground, which causes the mean atmospheric flow field to adjust according to the local roughness. Although it would be naïve to imagine that the full range of small scale disturbances afforded by these roughness elements can be comprehensively accounted for, however, it is necessary to represent the mean effects of these interactions in cases where the external atmosphere has bearing on the quality of a buildings microclimate.

Naturally ventilated buildings, which are commonly used in agriculture, rely heavily on the interaction between the atmosphere and building envelope in order to modify the indoor environment. Thus, it is of paramount importance that the profile of the atmospheric flow field used in CFD simulations should correspond with the conditions experienced at the modelled site as closely as possible. This means that a CFD model must specify a fully developed atmospheric boundary layer (ABL) under a constant shear stress considering variables such as the mean stream-wise velocity, kinetic energy and frictional velocity. In early CFD studies the ABL was modelled in the form of the Blasius solution for a laminar boundary layer over a flat plate, which meant that the appropriate structure of the turbulent regime in the flow field was not described (Bottcher and Willits, 1987). Since then, two mathematical expressions routinely used to represent the ABL in CFD models are the *power-law* and the *log-law*. The *power-law* has no theoretical justification other than it is known to fit the profile of the mean velocity well when the correct parameters are defined. The widely accepted *log-law* has been developed on sound physical principles, and is capable of accurately representing the lower regions of the ABL, i.e. the region of interest in natural ventilation studies. It can be represented in the following form:

$$U_z = \frac{u_*}{K_v} \ln\left[\frac{z+z_0}{z_0}\right] \tag{15}$$

Richards and Hoxey (1993) have provided a detailed explanation of the homogenous turbulent flow over rural terrain and defined the mean turbulent kinetic energy and dissipation rate inlet profiles for use in CFD simulations employing k–ε turbulence model. The variables k, ε and frictional velocity (u_*) can be represented by the following equations:

$$k = \frac{u_*^2}{\sqrt{C_\mu}} \tag{16}$$

$$\varepsilon = \frac{u_*^3}{K_v(z+z_0)} \tag{17}$$

$$u_* = \frac{K_v U_{ref}}{\ln((h+z_0)/z_0)} \tag{18}$$

In recent studies Eq. (15) has been fitted to the mean flow field measured at the experimental site (Fatnassi et al., 2003, 2006). By doing this, both the frictional velocity and roughness representative of the site can be accurately determined. Otherwise assumptions need to be made based on landscape classification tables (Simiu and Scanlan, 1996).

5.4.2. The fluctuating of nature wind

The atmospheric boundary layer varies in both speed and direction, which in turn alters a buildings ventilation rate over time. In many CFD studies of natural ventilation the fluctuating properties of wind have not been reproduced despite their obvious effect on the internal flow field. Wind-engineering studies have successfully employed LES to simulate the effects of natural wind conditions on a naturally ventilated building (Jiang and Chen, 2002). RANS models have shown to be limited in this regard, and whether the quasi-steady solution, obtained by a RANS model at each time step, can represent a real transient condition is still in doubt (Swaddiwudhipong and Khan, 2002). Moreover, because the solution must be developed with relatively small time steps, the computing time may not be much less than that required by LES (Jiang and Chen, 2002).

6. Validating the CFD predictions

Computational fluid dynamics, despite its versatility in modelling all types of ventilating flows, has a major fundamental limitation in that its accuracy is essentially limited by the uncertainties in specifying boundary conditions (Etheridge and Sandberg, 1996). Due to computational limitations CFD models cannot generally contain all the microscopic details of the physical system and some form of simplification is needed to reduce the number of calculations involved in developing a solution. These simplifications range from the indirect calculation of turbulence field to the assumptions made in the numerical representation of the physical boundary conditions. For example, empirical coefficients which form an integral part of many turbulence models and wall treatment models make assumptions about the flow that can have adverse effects on the accuracy of the field solution, especially when heat transfer calculations are made (Loomans, 1998). Thus experimental validation is a necessary part of the modelling process and the yardstick of success is the level of agreement that can be attained between numerical predictions and experiments (Xia and Sun, 2002). This section looks at the various techniques that have been used in the validation of CFD models.

6.1. Laboratory studies

Physical models have traditionally been used to study indoor airflows, and in more recent times to validate numerical predictions. With the use of an environmental wind tunnel a scaled version of the atmospheric boundary layer can be simulated. This makes it possible to determine wind and structure interactions on a scaled level, thereby offering a valuable means of quantifying the indoor climate during the design stage of agricultural buildings (Choiniere and Munroe, 1994; Sase and Takakura, 1984). Other techniques such as salt bath modelling have also been used to physically simulate flows caused by thermal buoyancy. CFD models have proven to be consistent with the results of physical modelling in many cases (Bartzanas et al., 2004; Mistriotis et al., 1997a). And even in cases where discrepancies existed, limiting factors were readily identifiable (Okushima et al., 1989). Techniques, which have been employed to measure flow at scaled level are discussed below.

6.1.1. Scaling for salt bath modelling

Physical models using working fluids other than air have been used over the years to simulate the movement of airflow patterns. Salt bath modelling is a technique that employs water infused with salt solutions of various concentrations to act the same way as thermal buoyancy in a full scale ventilated structure. Of course, dimensional analysis is needed in order to maintain consistency between the model and prototype. This comes about through the Buckingham Pi theorem, which typically requires three independent and dimensionless terms in ventilation studies. The scaling laws for studying ventilation in salt baths have been presented elsewhere in the literature and will not be discussed here (Montero et al., 2001a; Oca et al., 1998). With the Buckingham Pi theorem only knowledge of variables related to the problem of interest are required. However if one or more important variable is neglected, serious mistakes of model design could result. The potential of salt bath modelling as a validation technique for CFD has also been noted in the literature (Montero et al., 2001a).

6.1.2. Scaling for wind tunnel modelling

When air is used as the working fluid, it is essential to maintain geometric, kinematic and dynamic similarity. Geometric similarity is fulfilled once the solid boundaries are equal for the two systems. Kinematic similarity is achieved by accurately scaling flow boundaries, including air supply opening, outlet, and roughness of all surfaces (Yu and Hoff, 1999). Dynamic similarity requires equality of the dimensionless parameters that are involved in driving the airflow in both systems. The importance of each dimensionless parameter is conditioned by the ventilation mechanism that is being studied. In many model studies at least one or more of the following dimensionless parameters have been equalised between both the model and prototype: Reynolds number, Froude number, Euler number, Grashof number, Archimedes number and Rayleigh number. Accurately maintaining dynamic similarity has proven to be essential when validating CFD simulations (Boulard et al., 1999; Mistriotis et al., 1997a). A more detailed analysis on the use of dimensionless numbers in scaled flows can be found in the literature (Etheridge and Sandberg, 1996; Yu and Hoff, 1999).

6.1.3. Particle image velocitometry

Alongside the developments made in CFD in the past number of years, advancements have also been made in airflow measurement technology. Particle image velocitometry (PIV) is an example of a recently developed measurement technology. PIV measures a 2D velocity vector map of a flow field at any time instant by acquiring and processing images of particles seeded in the flow. PIV is based on the principle that speed is equal to the displacement divided by the time interval. The particle speed then represents air flow velocity.

There have been successful investigations of airflow patterns in greenhouses with PIV in recent years. However, most have been done on isothermal flow regimes. Lee et al. (2005) used a two-dimensional PIV technique to investigate natural ventilation characteristics of an isothermal multi-span greenhouse (Venlo-type and fully open roof type). The airflow distributions measured with PIV were used to study CFD accuracy and good agreement was found between both techniques. A PIV technique has recently been used to quantify the airflow patterns seeded with smoke in a slot ventilated enclosure (van Brecht et al., 2004, Fig. 6). This study was conducted on a larger scale than a wind tunnel model, thus making it possible to visualise the effects of buoyancy on the incoming flow. Although CFD simulations were not validated in this study, measurement techniques similar to this would be amenable to future CFD validation experiments.

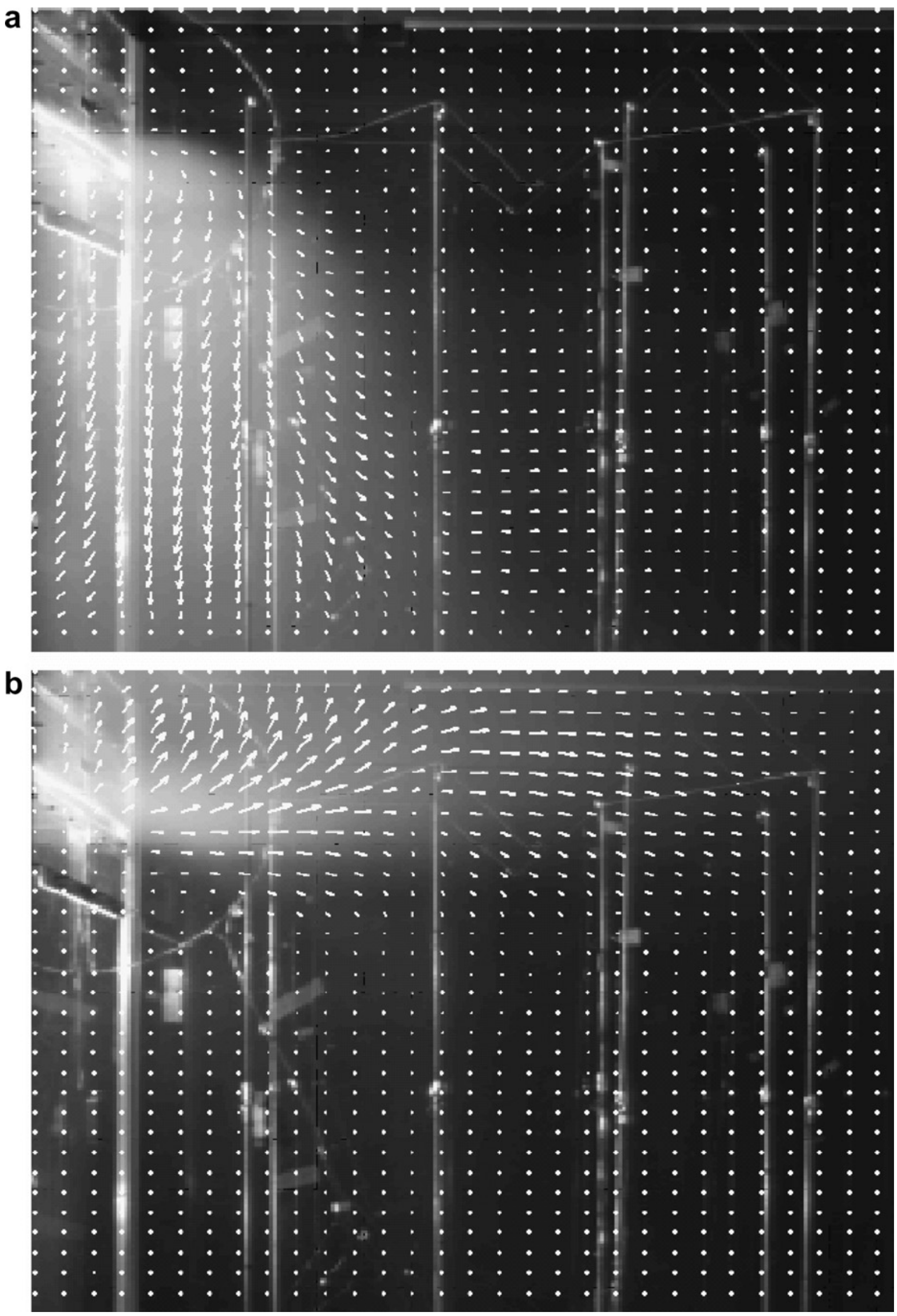

Fig. 6. Optical flow-field plotted over smoke patterns at ventilation rates of (a) 140 m^3/h and (b) 290 m^3/h (van Brecht et al., 2004).

6.2. Full scale studies

Over recent years, the measurement of the environment within full scale agricultural structures has proven to have two main purposes. Firstly, detailed measurements of the spatial and temporal distribution of climatic variables are the only means of accurately quantifying the dynamism of the indoor environment. Secondly, full scale measurements are often used to validate numerical models. Therefore, full scale experimental techniques including tracer gas and sonic anemometry, and they're amenability to CFD validation studies are reviewed below.

6.2.1. Tracer gas techniques

Tracer gas techniques are widely used to measure the ventilation rate and mixing characteristics of indoor airflows. The technique has been found to produce reasonably accurate measurements of ventilation rate when used in a continuous tracer injection mode (Baptista et al., 1999), and for that reason tracer gas is sometimes used to validate numerical models (Ould Khaoua et al., 2006). In recent years, a numerical tracer gas model has been incorporated in CFD simulations and has shown reasonable agreement with experiments (Bartzanas et al., 2004).

However, other than giving an indication of the level mixing within the building, the tracer gas technique has found to be lacking when a complete description of the internal indoor environment is required (Demmers et al., 2000). Boulard et al. (1996) noted that in tracer gas experiments, owing to the imperfect homogenisation of gas, the effective ventilation rate rather than the true ventilation rate is determined. This reason may explain the inconsistency between measurements and predictions in some studies (Demmers et al., 2000).

6.2.2. Sonic anemometry

Sonic anemometers are standard instruments used today in many micrometeorological studies. Alongside the capabilities for measuring three-dimensional wind speed and direction, they are also commonly used to accurately determine turbulence and heat fluxes. For this reason, sonic anemometers been adopted by the ventilation community to quantify the indoor climate and also to validate numerical predictions (Boulard et al., 1997). Sonic anemometers derive wind velocity components and sonic temperature from the differences between the transit times of forward and backward sound pulses between aligned transducers. Because sonic anemometers have the ability to determine the fluctuating properties of the flow, they can resolve the fast eddies that contribute to turbulent flux, and consequently most of this flux can be recorded.

The transport of heat, moisture and momentum in the agricultural structures are governed almost entirely by turbulence. Analysis techniques such as eddy covariance make it is possible to quantify turbulent transport given the high sampling rate of a sonic anemometer. As long as the data from sonic anemometers is interpreted correctly, they can provide a comprehensive account of a buildings indoor environment. The sound physical principles underlying eddy covariance systems are described by Boulard et al. (1996, 1997). Full characterisation of indoor environment of greenhouses has been completed by various authors (Boulard et al., 1997; Wang et al., 1999; Wang and Deltour, 1999). Wang and Deltour (1999) successfully used sonic anemometry to show the reverse airflow in a leeward ventilated greenhouse. The data from this study was later used as a basis for validation for the CFD model of Reichrath and Davies (2002a), who predicted the same phenomenon.

Sonic anemometry is highly amenable to the validation of CFD simulations. Good agreement has been found between CFD and sonic anemometer measurements for flows within ventilated enclosures, and in naturally ventilated structures (Bartzanas et al., 2004; Boulard et al., 2002). The high resolution of sonic anemometers can also throw light on the areas of the flow regime where it may be difficult to get agreement between CFD predictions and experiments. For example, Heber and Boon (1993) and Heber et al. (1996) noted that non-isotropic turbulence exists in the shear boundary layer of the air jet and near heat sources. Since most turbulence models assume isotropy in all regions of the flow, attempting to validate the CFD model at these regions would not make sense.

7. Computing the environment of greenhouses

Greenhouses are essentially plant growth conditioners, where the microclimate parameters are maintained at levels consistent with the biological demands of the crop. A CFD analysis of the greenhouse environment expresses the interaction between the outdoor climate and the internal crop canopy. With the power of computers rapidly increasing, the ability to provide elaborate spatial and temporal analyses of the greenhouse microclimate with a reduced amount of expensive and time consuming experimentation is now possible. CFD application studies used in the advancement of greenhouse technology before 2002 have been comprehensively reviewed by Boulard et al. (2002) and Reichrath and Davies (2002b). However, as noted by Bartzanas et al. (2004) many studies up to this time period suffered from a lack of realism due to a substandard modelling of heat and mass transfers between the crop and local environment. Therefore because of greater experience with CFD software packages and progression in CFD technology, more recent studies should reflect reality to a greater degree. Table 1 summarises the CFD modelling of greenhouses over the last twenty years. The developments since 2002 are discussed below.

7.1. Mono-span greenhouses

7.1.1. Tunnel greenhouses

The spatial distribution of velocity, temperature and humidity is known to vary strongly inside tunnel greenhouses. To investigate this, Boulard and Wang (2002) modelled a wind induced ventilation of a tunnel greenhouse with a 3 m s^{-1} wind of normal incidence to the structure. As outlined above, they incorporated subroutines into the CFD code which computed the crop dynamical response as a function of local conditions. Their model found a strong heterogeneity in the indoor environment, which was characterised by a main air stream crossing from windward to leeward openings, while the air along the floor and walls remained stagnant. Furthermore, the model computed the level of crop transpiration on the north side to be 30% smaller than other locations because of lower solar energy and air speed. However, no design solutions to create more uniform conditions were given.

Bartzanas et al. (2004) studied different vent arrangements in the same greenhouse with a wind of 3 m s^{-1} blowing both parallel and normal to the building axis. Solar energy models were not incorporated. Grid independence was checked and differences between predicted and measured ventilation rates varied from 12% to 15%. The authors highlighted the fact that the singular use of ventilation rate as a criterion was not the best option for evaluating the ventilation performance of greenhouses. Consequently, uniformity of temperature and humidity

Table 1
CFD modelling of ventilation in greenhouses from the earliest to the most latest studies

Authors	App.	Dim.	Code	Turb.	Grid	Val.	Occ.	Real.	S-o-t-a	Comments
Okushima et al. (1989)	Greenhse. flow-field	3D	–	Std. k–ε	NGCS	Poor	No	1	5	First use of std. k–ε in natural ventilation flows. Insufficient computational capability to test grid dependency
Mistriotis et al. (1997a)	Greenhse. flow-field	2D	PHOENICS	RNG k–ε	NGCS	Qual.	No	3	5	First to use a numerical tracer gas technique. Good analysis of various turbulence models
Mistriotis et al. (1997b)	Greenhse. flow-field	2D	PHOENICS	RNG k–ε	NGCS	Reas.	Yes	3	5	Plants represented by a heated floor. Various empirical models used to determine the sensible heat partition
Bartzanas et al. (2002)	Greenhse. design	3D	CFD 2000	Std. k–ε	NGCS	Reas.	Yes	4	4	The drag effect of the crop canopy on the flow-field was computed using porous media. Good analysis of insect screens versus ventilation rates
Boulard and Wang (2002)	Greenhse. flow-field	3D	CFD 2000	Std. k–ε	NGCS	Reas.	Yes	5	5	Crop transpiration model and drag models were involved. Most realistic study up to that point
Campen and Bot (2003)	Greenhse. flow-field	3D	FLUENT	Std. k–ε	NGCS	Reas.	No	5	4	Used an empty building to observe the effect of wind incidence on ventilation rate. Noted that this effect must be taken into account when validating CFD flow-field predictions
Fatnassi et al. (2006)	Greenhse. design	3D	CFD 2000	Std. k–ε	NGCS	Good	Yes	5	5	Comprehensive modelling exercise applying the models of Boulard and Wang (2002) to greenhouse design

App., application; dim., dimensionality; turb., turbulence model; val., quality of experimental validation; Occ., were the occupants of the building simulated?; real., indication between 1 and 5 of how realistic the study was, with 1 being least realistic and 5 being most realistic; S-o-t-a, an indication between 1 and 5 of where the study was in terms of the state-of-the-art at that period, with 1 being furthest and 5 being closest; Greenhse., Greenhouse; NGCS, no grid convergence study; reas., reasonable; hse, house; std. k–ε, standard k–ε turbulence model; RNG k–ε, RNG k–ε turbulence model.

over the crop were also incorporated as criteria in the study. Bartzanas et al. (2004) concluded that a combination of side and roof openings offered a design solution that achieved well ventilated and uniform climatic conditions. However, due to modelling constraints, the effect of vent configuration on crop transpiration could not be investigated from north to south of the greenhouse.

A more recent CFD study of tunnel greenhouse predicted the indoor environment with various RANS turbulence models (Roy and Boulard, 2005). The predictions were not validated by experiments. Although similar profiles were predicted by all models, the RNG k–ε model predicted velocities three times greater than those predicted by the standard k–ε model. The authors noted that their next step was the fine experimental determination of velocity, temperature and humidity profiles in order to validate the turbulence model. Since the applied boundary conditions were very similar to those used by Bartzanas et al. (2004) and Boulard and Wang (2002) (all of whom used the standard k–ε model), the solutions of Bartzanas et al. (2004) and Boulard and Wang (2002) could yet be rendered little more than indicative.

7.1.2. Pitched roof greenhouses

Since the first CFD studies carried out by Mistriotis et al. (1997a,b) there have been a number of investigations into the climate distribution within pitched roof greenhouses, with nearly all being conducted using a two-dimensional assumption. The extension of these models to three-dimensions has only taken place in more recent years. These consist of both isothermal and non-isothermal studies. As of yet no explicit incorporation of crop transpiration models, similar to those noted above, have been published (Campen, 2005; Lee et al., 2005; Shklyar and Arbel, 2004).

Shklyar and Arbel (2004) computed the flow in an isothermal, pitched roof, single span greenhouse in order to study the effect of vent angles and wind incidence on wind induced ventilation. The standard k–ε turbulence model was used and overall grid dependence was examined. They found that by changing the vent angle from 20° to 40° the ventilation rate almost doubled. In addition, this ventilation rate was less sensitive to a change in wind direction from 0° to 45° than from 45° to 90°, i.e. ventilation rate was induced by a perpendicular wind was almost five times greater than that when wind was parallel to the structure. Shklyar and Arbel (2004) also noted how the longitudinal component of the air velocity reduced to almost zero in the centre of the building, and used this observation to uphold the validity of all past two-dimensional CFD greenhouse studies. However, it would seem that this observation was highly dependant both on the turbulence model (standard k–ε model was employed) and the mesh density used in the simulation (mesh was coarse along longitudinal length of the building).

More recently Campen (2005) used CFD to investigate the climate distribution within a pitched roof greenhouse under different design constraints, including different wind speeds and direction, various porous screens, and different structural configurations. The standard k–ε turbulence model was used and no details were given on mesh density. The crop was represented by a porous medium releasing a sensible heat flux. This investigation yielded some interesting results. Firstly, it was found that the resistance afforded by a screen net was more influential on the ventilation rate than wind direction. At low wind speeds, $0.5\ \mathrm{m\,s^{-1}}$, a greenhouse with roof and sidewall ventilation gave a large temperature increase inside the building. This was caused by both buoyancy and wind forces counteracting one another. This temperature increase was also induced when the greenhouse of various design configurations was lengthened from 12 m to 36 m. The effect was more pronounced when obstructions, i.e. adjacent buildings, were in close proximity.

7.1.3. Specialised single span greenhouses

CFD has been used to provide design solutions for the indoor environment of many different style single span greenhouses. These greenhouses have been specialised according to local environmental conditions and range from Moroccan type greenhouses to solar-distiller greenhouses. Table 2 summarises the CFD studies carried out for specialised greenhouses.

7.2. Multi-span greenhouses

The "parral" greenhouse is the most commonly used multi-span structure in Mediterranean countries including Europe and North Africa. In its present form the "parral" greenhouse is an empirically developed low cost structure, which provides shelter from rain and wind and acts as a solar collector in winter (Baeza et al., 2005). Since its conception, many two-dimensional and three-dimensional CFD studies have investigated the distributed climate inside parral greenhouses, with the most recent ones carried out by Brugger et al. (2005), Baeza et al. (2005), Molina-Aiz et al. (2005), Campen and Bot (2003), Kacira et al. (2004) and Fatnassi et al. (2006).

7.2.1. Two-dimensional studies

Many of the existing two-dimensional CFD studies have undertaken various parametric investigations of the greenhouse structure in order to recommend growers and greenhouse manufacturers the optimal ventilation configuration (Brugger et al., 2005; Molina-Aiz et al., 2005). However, other than acting as an indication of the possible flow and temperature patterns with a greenhouse, two-dimensional CFD predictions of such a large-scale structure have little additional value and cannot be generalised (Molina-Aiz et al., 2005). In a 2D study Molina-Aiz et al. (2005) observed a reduction in ventilation rate of around 88% when the span of the building was increased from 1 to 5. Large temperature gradients were also observed in the middle of the building under all vent configurations, and insect

Table 2
CFD modelling of ventilation of alternative styled greenhouses

Authors	GH style	Design challenge	Code	S-o-t-a	Realism	Results and comments
Fatnassi et al. (2003)	Parral	To prevent the influx of insects into the building whilst maintaining a good indoor climate	CFD 2000	5	5	Authors suggested that increase in vent opening area is necessary when grower uses insect nets. Indoor climate was highly dependant on type of insect net
Fath and Abdelrahman (2005)	Solar distilled	To provide an optimum indoor environment whilst distilling water	FLUENT	5	4	Inlet velocity should not decrease below 0.5 m/s in order to maintain suitable levels of relative humidity
Davies and Paton (2005)	Solar distilled	To reduce the amount of complex airflow patterns in the indoor environment	FLOVENT	4	5	Introducing a pipe array that shaded the plants provided best control over the flow patterns

GH, greenhouse; S-o-t-a, an indication between 1 and 5 of where the study was in terms of the state-of-the-art at that period; with 1 being furthest and 5 being closest; Realism, an indication between 1 and 5 of how realistic the simulation was; with 1 being the least and 5 being most realistic.

screens were seen to greatly affect the ambient temperature and velocity difference between indoor and outdoor environments. Baeza et al. (2005) showed that by increasing the windward flaps on the roof of a 2D parral greenhouse model, temperature uniformity and ventilation rates were enhanced. Unfortunately, the fact that no crop drag effect was simulated took from the realism of the simulation. This was also the case with Brugger et al. (2005), who developed a 2D model to show that increasing the roof slope has a positive effect on ventilation rates. The study concluded that for an improved ventilation rate, light transmission and easiness of construction, a roof slope of 27° provided a good compromise for the model greenhouse.

Radiative transfers have also been modelled in 2D in order to investigate night time climate and energy fluxes of an unheated greenhouse (Montero et al., 2005). In contrast to the other 2D CFD models, this study provided a good explanation of the thermal behaviour of the greenhouse, as all predictions were in close agreement with experimental observations, and because night time radiative fluxes are reasonably spatially and temporally invariant. The solution fields showed that internal curtains may reduce the risk of condensation. In addition, Montero et al. (2005) noted similar CFD simulations have shown that there is a potential for using aluminised materials on greenhouse roofs to passively increase the temperature of the incoming air.

7.2.2. Three-dimensional studies

The first three-dimensional study of a multi-span greenhouse was carried out by Campen and Bot (2003), who investigated the effects of wind speed and direction on the wind induced ventilation rates of a parral greenhouse. The study showed that a change in wind direction by only 10° could increase the ventilation by up to 50%. Overall ventilation exchange was only slightly lower for flap windows, which was due to the greater opening area of the ventilators. Interestingly, in the case of the roll-up ventilators, the predicted ventilation rates were much higher at the leeward of the structure than those of the flap ventilated structure. However, more comprehensive account of the physical processes must be taken into account in future investigations in order to lessen the variance between predictions and measurements, which reached 60% in this study.

Kacira et al. (2004) showed that roll-up side ventilators in saw-tooth multi-span greenhouses enhanced the ventilation flow rate per unit floor space over the crop canopy when compared to butterfly side ventilators. No temperature or humidity effects on the indoor climate were simulated; thus the criterion used by Kacira et al. (2004) for choosing the ventilation configuration was inadequate, and more comprehensive solutions must be developed before the conclusions can be upheld. Nevertheless, the observation that the majority of the incoming flow never reached the crop canopy when butterfly side-openings were used is of importance to both growers and greenhouse manufacturers.

The most realistic modelling study of a multi-span greenhouse has been carried out by Fatnassi et al. (2006) and is illustrated in Fig. 7. Their model was similar to Boulard and Wang (2002) and Roy and Boulard (2005) in that the transpiration phenomena of the indoor crop canopy were simulated as close to the physical situation as possible. The simulations showed only slight differences between singular use of windward ventilators and the combined use of windward and leeward roof ventilators. Thus the general consensus that a combination of ventilators is required was disproved. The predictions exhibited a 2–3 times increase in humidity and temperature with respect to outside conditions when insect screens were used. When such screens are necessary, Fatnassi et al. (2006) suggest that some form of climatic modification such as a fogging system should be employed so that humidity and temperature can be maintained at a level favourable for plant growth.

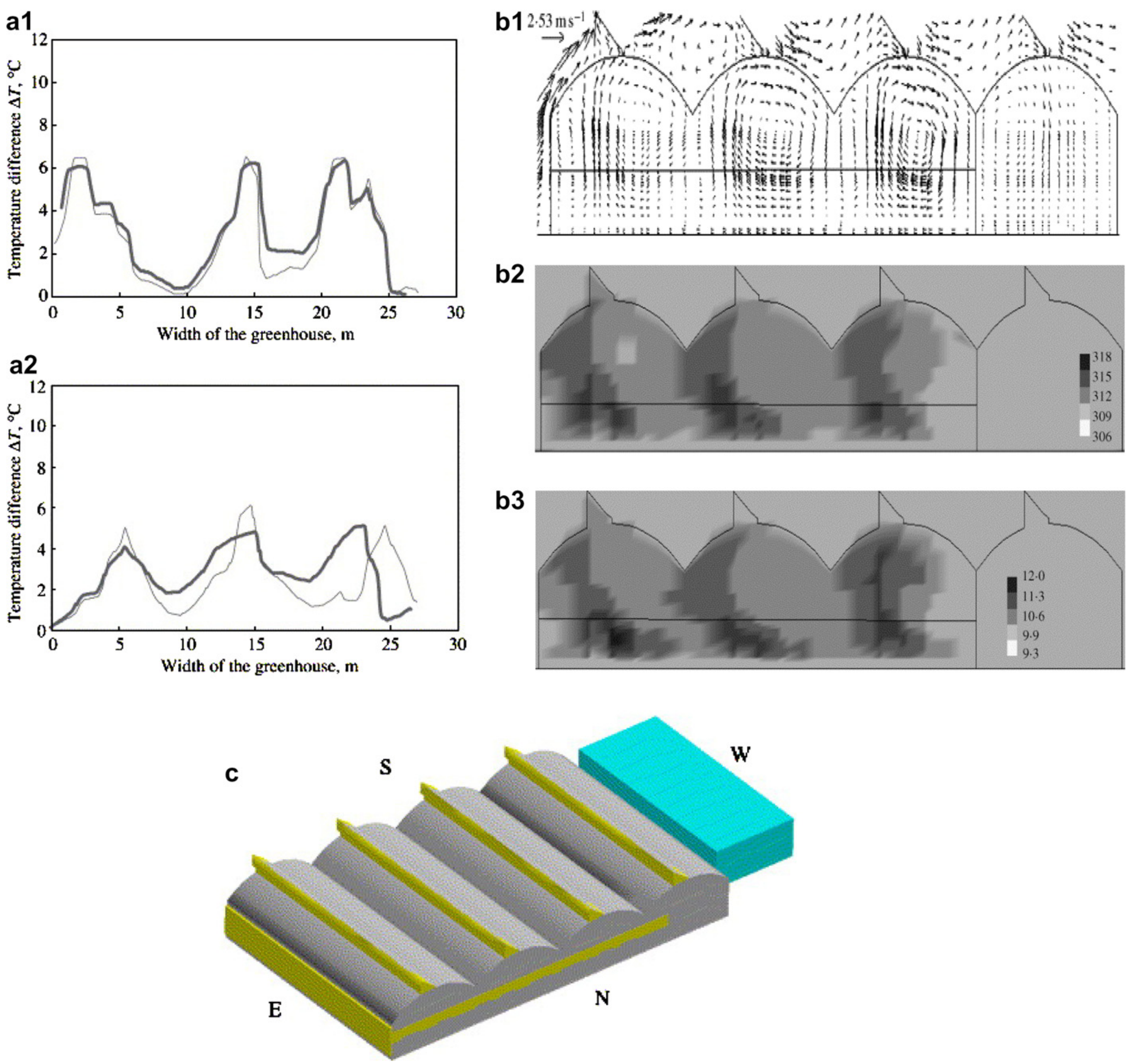

Fig. 7. The three-dimensional modelling of a parral style greenhouse (Fatnassi et al., 2006). Plots (a1) and (a2) show that using a combination roof and side openings do not increase the ventilation efficiency, plot (a1): temperature difference between inside and outside, where — windward openings only, — windward and leeward openings, plot (a2) temperature difference between inside and outside, where — windward and side openings, — windward and leeward and side openings. Plots (b1)–(b3) show the indoor flow-field (m/s), temperature (K) and humidity (g/kg) in the building from top to bottom respectively. Plot (c) shows a three-dimensional view of the parral greenhouse.

8. Computing the environment of animal buildings

Similar to the application of CFD in greenhouse design, many of the past animal housing studies lacked realism through the employment of broad assumptions and/or substandard modelling of the housing conditions. Indeed, the first CFD studies crudely approximated the geometry of livestock buildings as a two-dimensional rectangle (Choi et al., 1988, 1990). Small advances towards the development of realistic solutions were made in the following years. Hoff et al. (1992) used CFD to model the buoyant flow in a slot ventilated rectangular facility, in which heat transfer was simulated via a heated floor. Harral and Boon (1997) simulated the isothermal flow patterns in a three-dimensional geometrical representation of a livestock building. In later years more work was done in developing validation facilities with focus to creating more accurate CFD models (Zhang et al., 2000). Only in recent times has CFD modelled heat and mass transfer in a geometrical representation of an animal production facility (van Wagenberg et al., 2004).

The need for a dynamic representation of heat and mass transfer in livestock buildings in CFD studies has lately been acknowledged (Aerts and Berckmans, 2004; Gebremedhin and Wu, 2003). When dynamic models were used in conjunction with CFD predictions, they provided a more detailed picture of all the occurring phenomena (Gebremedhin and Wu, 2005). Because dynamic interactions have only recently been investigated, the accuracy afforded by CFD simulations has yet to be brought to the level at which greenhouse technology is currently situated. Table 3 summarises the CFD modelling studies of the indoor environments of animal houses, and the present status of CFD research in animal housing is discussed below.

8.1. Poultry houses

The indoor environment of poultry houses plays a decisive role in the ultimate success of a production facility. Air temperature, relative humidity, pollution concentration, thermal radiation and air movement directly affect the birds' ability to maintain homeothermy. If subjected to conditions outside the comfort zone a bird needs to make physiological adjustments to maintain core body temperature (Tinôco et al., 2001). This reduces production performance because dietary energy is used to produce or dissipate heat instead of being used for growth and development. Therefore, the requirement to keep the thermal environment within an optimal production range is evident. Yet, only three CFD studies exist, in which the indoor environment of a poultry house has been investigated.

Due to numerical modelling constraints and small computational capacity, van Ouwerkerk et al. (1994) were only able to model two-dimensional air motion and heat transfer and, considering the associated limitations, aptly used the simulation results to determine locations within the building where environmental control sensors may act effectively. Mistriotis et al. (1997c) studied the natural ventilation in a broiler house with a two-dimensional CFD model. The study focused on addressing the problem of meeting adequate levels of ventilation on days of high solar radiation coupled with low wind speeds. The animals were modelled as a constant heat flux, which was represented by a heated floor. Mistriotis et al. (1997c) found that by employing a solar chimney instead of conventional housing techniques, the indoor temperature and air velocity were maintained within the animals comfort zone. Both of the above studies highlighted the scope for the numerical simulation to aid in the better designing of housing systems for poultry.

The level of indoor pollution also influences production performance and it has been found that, in many poultry production facilities, the hygienic threshold values for indoor pollution are often exceeded (von Wachenfelt, 1994). As deviations from this threshold predispose poultry to respiratory infections and development of lesions in the eyes (Hauser and Folsch, 1988), the ability to analyse and control the dispersion of pollutants is of paramount importance. Worley and Manbeck (1995) used CFD to compute the dispersion of contaminants in a highly turbulent flow regime within a poultry house. Again, the lack of computer power precluded the development of a comprehensive three-dimensional model. Computations were validated with measurements taken from a 1/5 scale slot-inlet ventilated poultry building. The objective was to evaluate the performance of several ceiling inlet configurations by computing the ventilation effectiveness with a simple random flight model. Predictions showed that multiple slotted ceiling inlet system and porous ceiling inlet system exhibited good airflow, particulate removal, and particle concentration characteristics.

8.2. Pig houses

It has long been known that microclimate control, which focuses on regulating indoor environment according to the demands of the animals, is important in intensive swine production (Panagakis and Axaopoulos, 2004). With respect to Irish pig production, based on the correlation between lung lesions and food conversion ratio, it has been estimated that complicated respiratory diseases cost the Irish pig industry €3 million annually (Kavanagh, 2003).

Many of the CFD applications in animal housing have been directed towards optimising environmental conditions in pig production facilities (Bjerg et al., 1999, 2000; Sun et al., 2002, 2004b). These studies can be effectively grouped into the modelling of pollutant dispersion and the thermal environment.

8.2.1. Pollutant distribution

Sun et al. (2002) developed an isothermal, two-dimensional CFD model of an alternative pig housing system, a high rise hog building (HRHB), to predict the distribution

Table 3
CFD modelling of ventilation in animal housing from the earliest to the most latest studies

Authors	App.	Dim.	Code	Turb.	Grid	Val.	Occ.	Real.	S-o-t-a	Comments
Bottcher and Willits (1987)	Animal hse. flow-field	2D	In-hse	Lam.	GCS	Reas.	No	1	5	Used vorticity and stream-function technique. One of the first papers on the numerical prediction of natural ventilation
Choi et al. (1990)	Animal hse flow-field	2D	In-hse	Std. k–ε	NGCS	Good	No	1	4	First to use the std. k–ε model in mechanically ventilated rectangular animal room, suggested modification to std. k–ε model for similar type flows
Hoff et al. (1992)	Animal hse. flow-field	3D	In-hse	LRN k–ε	NGCS	Reas.	Yes	2	5	Animals represented by a heated floor in a rectangular room. Results cannot be generalised as basis for design
Harral and Boon (1997)	Animal hse. flow-field	3D	PHOENICS	Std. k–ε	GCS	Reas.	No	2	3	Only section of facility modelled. Authors proposed modifications to std. k–ε model and found predictions to be highly dependant on grid density
Sun et al. (2002)	Animal hse. design	2D	FLUENT	RNG k–ε	NGCS	Poor	No	1	2	Used a passive scalar to model ammonia distribution. Results cannot be generalised as basis for design
van Wagenberg et al. (2004)	Animal hse. design	3D	FLUENT	Std. k–ε	NGCS	Poor	Yes	3	4	Used real conditions to validate model. Found large discrepancies between predictions and measurements. Predictions showed large variations in front of building
Gebremedhin and Wu (2005)	Animal hse. flowfield	3D	PHOENICS	RNG k–ε	NGCS	None	Yes	3	4	Used flow field to compute heat and mass transfer from the cows surface, no direct coupling in CFD program, only isothermal flow-field computed

App., application; dim., dimensionality; turb., turbulence model; val., quality of experimental validation; occ., were the occupants of the building simulated?; real., indication between 1 and 5 of how realistic the study was; with 1 being least realistic and 5 being most realistic; S-o-t-a, an indication between 1 and 5 of where the study was in terms of the state-of-the-art at that period; with 1 being furthest and 5 being closest; in-hse, in-house code; lam., laminar model; GCS, grid convergence study; NGCS, no grid convergence study; reas., reasonable; hse, house; std. k–ε, standard k–ε turbulence model; LRN k–ε, low Reynolds Number k–ε turbulence model; RNG k–ε, RNG k–ε turbulence model.

of ammonia and found considerable concentrations in localised areas above the slatted floor. Unfortunately, this model was inaccurate as the fan position was misrepresented in two-dimensions. In a later study Sun et al. (2004b) modelled the ammonia distribution in the same isothermal building in three dimensions. Predictions showed that this housing system was similar to conventional deep pit buildings at evacuating indoor airborne ammonia. However, the isothermal assumption hampered the accuracy of the model. Furthermore, the passive scalar approach employed by Sun et al. (2002, 2004b) used concentration values as input for the CFD calculations of ammonia distribution. Therefore this simulation cannot be viewed as accurately modelling the ammonia emissions from the surface of the stored slurry. Predicala and Maghirang (2003) used dynamic modelling techniques of CFD to compute the stochastic transport of particulate matter in a mechanically ventilated swine barn. The study determined a simple method of calculating the emission rates of particulate matter from buildings equipped with multiple fans.

8.2.2. The thermal environment

8.2.2.1. Phenomenological studies. Early three-dimensional predictions of airflow in a livestock room found reasonable agreement with experimental results even though the housed animals were modelled as a uniformly heated floor (Hoff et al., 1992). Hoff et al. (1992) also predicted a number of flow phenomena, yet none of which could be generalised to provide design solutions for related structures. Bjerg et al. (2000) found that their CFD model inadequately predicted recirculation zones in locations where these occurred experimentally. Nevertheless, Bjerg et al. (2000) were able to form a number of recommendations for the CFD modelling of similar animal housing airflows. Zhang et al. (1999) studied the buoyant flow generated by a single simulated pig. Transient simulations were observed to agree well with measured data. Zhang et al. (1999) recognised that although the simulations represented a gross simplification of the actual dynamics inside the building, similar type modelling techniques could be used to develop accurate solutions when more complex interactions occur.

8.2.2.2. Design studies. Some design studies, which have employed CFD to predict the environment in occupied functional livestock buildings, have been conducted in recent times. Khankari et al. (1997) conducted 2D simulations of animals in a swine building by using obstructions with a constant temperature source and found that the animal's microclimate may be conserved by keeping one ceiling inlet open. Lee et al. (2004) also conducted a CFD analysis of a fully stocked swine house. Their results showed errors of 10–18%, and as boundary conditions were not emphasised and a number of broad assumptions were made, it is difficult to assess the quality of such a study.

More recently, van Wagenberg et al. (2004) tested the possibilities for using experimental boundary conditions, taken from environmental measurements with live animals, as input to CFD simulations, in order to investigate the viability of integrating simulation and experimentation in the development of swine housing ventilation systems. The standard k–ε model was employed but a grid independence study was not carried out. Their model showed considerable discrepancies between measurements and predictions in some building locations, whereas good agreement was found in other locations. The disparities between simulated and measured temperature distribution were thought to arise from a combination of inadequate grid density, simplified inlet boundary conditions and the exclusion of radiative effects. Another possible reason for these discrepancies could be due to the inadequate modelling of the thermal response of the pigs. Bruce and Clark (1979) noted that by physiological and behavioural means the pig is able to adjust its sensible and latent heat production to maintain a constant total. Thus simulating the pig as a static and constant source of heat is inadequate if a detailed solution, validated by true conditions, is required. Overall, van Wagenberg et al. (2004) confirmed that the air distribution was inhomogeneous at high ventilation rates, and predicted large CO_2 concentrations and high temperature differentials in the pens closest to the inlet, which were consistent with experimental measurements. The results also indicated that with further advancements in computer power and modelling expertise, modelling environmentally dependant boundary conditions may bring about a higher degree of accuracy in future CFD studies.

8.3. Cow houses

Fig. 8 illustrates a CFD predicted flow-field around cows in a ventilated room. Only a few cases of CFD applications in cow housing are evident in the literature. Nonetheless, because the CFD simulations were used in conjunction with animal biological responses to determine the thermal environment, these studies have proved highly valuable and have raised the bar for future simulations. Gebremedhin and Wu (2003, 2005) solved the flow around cows to investigate, with an external program, the heat and mass transfer phenomena in a forced ventilated enclosure of simple geometry. They found that the total heat loss from an animal is highly dependant on both the animals' position and orientation to the flow field. For example, a cow positioned very close to the inlet had a total heat loss of 710 W, whereas an animal positioned to the back of the room had a heat loss of 214 W. Furthermore, a wall inlet producing a ceiling air jet seemed to create the most uniform environmental conditions in the building and minimised the large differences in animal heat loss (Gebremedhin and Wu, 2005).

Even though Gebremedhin and Wu (2003, 2005) calculated heat and mass transfer from the simulated cows, only isothermal conditions were simulated in the CFD model. Consequently, this study was suggested to be the first step towards the development of a comprehensive non-isother-

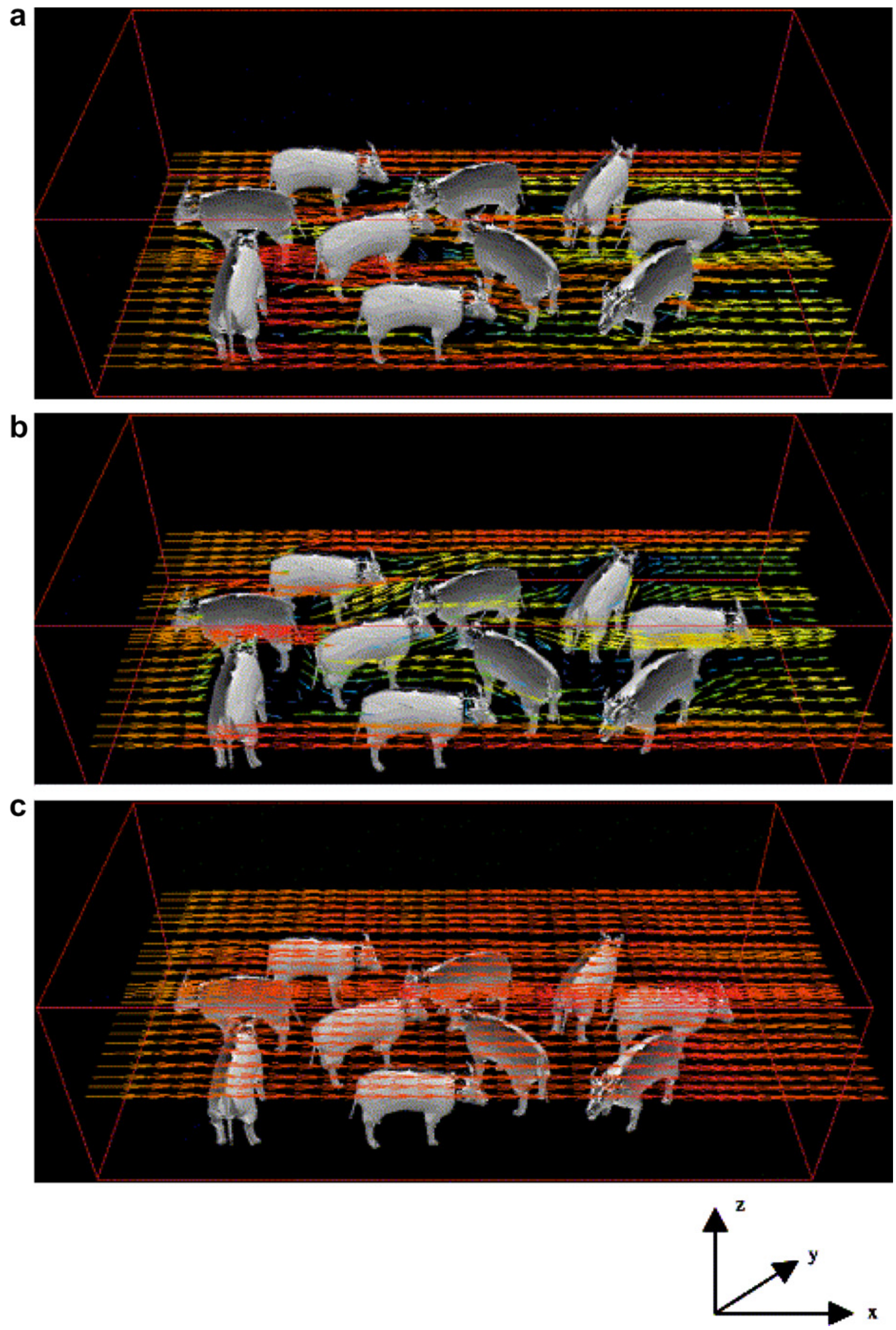

Fig. 8. Flow field around a ventilated room of cows at three different horizontal planes: (a) x–y plane at $z = 0.6$ m, (b) x–y plane at $z = 1.2$ m, and (c) x–y plane at $z = 1.8$ m. (Gebremedhin and Wu, 2003).

mal CFD model, in which heat production by a biological entity will be included.

9. Challenging issues confronting CFD modellers

9.1. Application issues

9.1.1. Dispersion modelling

The dispersion of contaminants in the highly turbulent flow regimes that dominate livestock buildings has also been a focus of some studies (Hoff et al., 1995; Reynolds, 1997; Sun et al., 2002; Worley and Manbeck, 1995). Hoff et al. (1995) and Sun et al. (2002) studied the distribution of ammonia concentration in a pig house by adding a passive scalar and solving it simultaneously with the velocity equations. This is known as the Eulerian concept and reasonably good agreement with measurements have been found with this technique. However, this concept is invalid when particulate sizes of about 1 μm are present in the flow regime and Lagrangian stochastic models, i.e. random flight models (using a Lagrangian autocorrelation function) are required (Reynolds, 1997). This technique allows particles that are thrown into the near boundary region of the airflow to experience velocities lower than those sufficient to maintain streamline trajectory. Simple random flight models, such as those used in the simulations of Worley and Manbeck (1995) and more recently Predicala and Maghirang (2003) have been reasonably successful but cannot be considered fully accurate in flows where inhomogeneous turbulence occurs (Harral and Burfoot, 2005). Because of this, Reynolds (1997) developed a more comprehensive random flight model for use with the k–ε turbulence modelling of complex ventilating flows. The superiority of this model for prediction of particle trajectories in a food production facility has been recently confirmed (Harral and Burfoot, 2005).

9.1.2. Modelling occupant movement

As noted from the results of van Wagenberg et al. (2004), the stochastic nature of occupant behaviour in buildings, which are generally ignored in modelling studies, can lead to inconsistencies between measurements and predictions. Whilst on a large scale CFD simulations may qualitatively correspond to the flow patterns and pollutant distribution in buildings, on smaller scales (i.e. in confinement pens) the discrepancies between measurements and predictions can be so large that the consequence of ignoring the influence of occupant movement may lead to the development of erroneous design criteria (van Wagenberg et al., 2004). Although the influence of occupant movement is out of reach for all macro-models many commercial CFD packages can be exploited in their present form to compute the effects of occupant movement by either direct or indirect means.

Direct simulation of occupant movement requires the involvement of Lagrangian algorithms, in which individual mesh nodes follow the associated particles during the movement. This usually demands that the computational mesh around a moving object be updated for each timestep of the simulation. As a consequence, such computations can be extremely time consuming and may not even be possible on large scales where more than one individual is moving.

Indirect simulation, which demands more synergism between CFD and experimental work, can be completed in much quicker times and seems to be the suitable technique for understanding the complex flow behaviour arising from occupant movement. Brohus et al. (2006) successfully introduced this technique by investigating the influence of human movement on contaminant distribution in operating theatres. Movement was simulated by means of locally distributed sources of momentum and kinetic energy. The study noted that in order to gain quantitative information at local levels, as a function of typical occupant behaviour, future investigations should comprise the statistical treatment of behavioural data in order to comply with the stochastic nature of the topic. This means that statistical data similar to that developed from the behaviour of housed pigs (Boon, 1982) can be exploited if future CFD applications in animal housing take this technique on board.

9.1.3. CFD in ventilation control

It has been shown above that CFD comprehensively simulates the natural phenomenon, and has demonstrated its usefulness as an analysis, design and system optimisation tool over recent years. Unfortunately due to the enormous complexity involved in these simulations, CFD techniques are not yet amenable to the on-line control of ventilation systems, and reduced order models which use statistical data to effect changes to the environment via controlled inputs may be more appropriate (Desta et al., 2004). Nevertheless, these reduced order models are heavily reliant on good quality data, which is often unattainable in practice.

Recent studies have combined both low order and CFD models to address this problem. The means by which this is done is illustrated in Fig. 9. Simply put, these studies confirmed that in practical situations where good experimentation practices are not generally possible, the use of CFD to generate the time-series data is a viable alternative. More research is needed to investigate into whether similar approaches can be applied to the control of natural ventilation.

9.2. CFD modelling issues

9.2.1. Turbulence modelling

The lack of understanding surrounding the quantification of turbulent air motion and its effect on indoor housing systems has been an issue confronting ventilation engineers over the last 2 decades. In many applications, the simplifying assumptions that have been made by turbulence modellers have proved to be unreasonable. A typical example of this is the Reynolds number assumption, whereby either a high or low Reynolds number flow regime is assumed *apriori* to a simulation. The most outstanding misapplication of this is in studies where turbulent and laminar flow regimes co-exist, such as in the case of animal houses and greenhouses. This fact is evidenced by many studies in which all high-Reynolds number k–ε turbulence models have yielded very similar results of reasonable to poor quality. The low-Reynolds turbulence model has been used to overcome this inadequacy, yet due to mesh demanding requirements it is rarely used. 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.

9.2.2. Time stepping in transient simulations

Airflow is inherently unsteady. Thus in many cases steady-state ventilating flow regimes do not exist and numerical difficulties are often encountered when trying to solve the steady governing equations (Stamou and Katsiris, 2006). Some studies have overcome this by "forcing" a

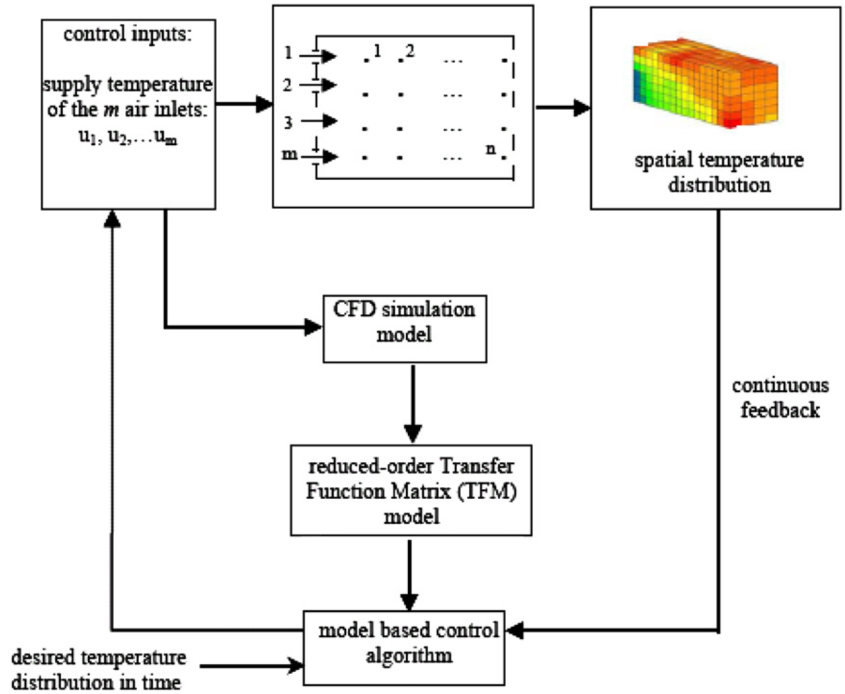

Fig. 9. Schematic of the approach used in the development of a CFD model-based controller (Desta et al., 2004).

steady solution. This means that some constraining condition was imposed to suppress the unsteady features of the flow regime; typically involving the use of adaptive time-stepping, first-order convection schemes, k–ε turbulence models (which are known for their diffusiveness), and two-dimensional or symmetrical boundary conditions. The forcing action does not exist in nature; thus the veracity of such simulations may be dubious and can only form indicative solutions.

Time stepping is an important attribute of CFD that allows a solution to march forward in time. An optimum time-step can be considered as a trade-off between computational efficiency, temporal accuracy, and stability of the numerical scheme. The maximum time-step selection of CFD computations is bounded by the requirements of the simulation, i.e. the time-step must be small enough to resolve the frequencies of importance in the unsteady flow regime. This frequency can be obtained from non-dimensional numbers such as the Stroudal number if forced convection is dominant, from experimental data, or from previous computations (Liu et al., 2003). It has been shown that using a small number of outer-iterations to develop a converged solution at each time step results in the most accurate way of simulating transient flows (Liu et al., 2003).

9.3. Enhancing CFD solutions

As shown above past CFD studies have been often limited by assumptions and poor modelling techniques, i.e. two-dimensional modelling and inadequate turbulence model application. Fortunately CFD technology has developed over recent years so that more comprehensive three-dimensional studies can be conducted efficiently. CFD codes offer a large range of features, which can meet the demands of the ventilation engineer. However, as commercial codes are designed for robustness rather than accuracy, it is necessary to maintain control over the modelling process so that all accuracy requirements are upheld. Some aspects of this are discussed below.

9.3.1. The CFD mesh

One of the major advances to occur in meshing technology over recent years was the ability for unstructured and hybrid meshes to be incorporated into general codes. This allows the CFD mesh to be fit to any arbitrary geometry (Shklyar and Arbel, 2004). This also means that local mesh refinement can be achieved both more effectively and efficiently and a solution can be developed to capture the desired flow features without creating distorted cells, such as those created by structured meshes. The versatility of unstructured meshes has meant that CFD is now finding more efficient solutions in cases where complex geometry plays a large role in developing flow patterns (Mirade, 2003).

9.3.2. Observing the y^+ criterion of turbulence models

An important feature of RANS turbulence models is the near wall treatment of turbulent flow. Low Reynolds number turbulence models solve the governing equations all the way to the wall. Consequently, a high degree of mesh refinement in the boundary layer is required to satisfactorily represent the flow regime, i.e., $y^+ \leqslant 1$. Conversely, high Reynolds number turbulence models use empirical relationships arising from the log-law condition that describe the flow regime in the boundary layer of a wall. This means that the mesh does not have to extend into this region, thus the number of cells involved in a solution is reduced. The use of this method requires $30 < y^{+} < 500$ (Versteeg and Malalsekeera, 1995), although $y^{+} \approx 10$ is also acceptable (Sorensen and Nielsen, 2003). Generally these wall treatment assumptions do not adversely affect solutions and many studies have employed them with relative impunity.

9.3.3. Mesh convergence study

The designing of a good quality CFD model, necessities mesh refinement so that one can obtain as nearest to a grid independent solution as possible. Unfortunately, in some large scale ventilation modelling cases it is still not yet possible to obtain a grid independent solution with today's computer power (Mirade, 2003). A spatial convergence technique proposed by Roache (1998) has been used in CFD ventilation applications. In using this technique the requirements for grid independence are relaxed, and the engineer is given a conservative estimate of the error (GCI) between the fine-grid solution and the unknown exact solution (Sorensen and Nielsen, 2003). To carry out a mesh convergence study both CFD solutions must be on a grid that is within the asymptotic range of convergence. The GCI can then be described as:

$$\mathrm{GCI} = \frac{F_{\mathrm{s}}|\varepsilon|}{(f^{p_{\mathrm{o}}} - 1)} \tag{19}$$

where the relative error ε between fine and coarse grid solutions is defined as:

$$\varepsilon = \frac{\mathrm{fo}_2 - \mathrm{fo}_1}{\mathrm{fo}_1} \tag{20}$$

F_{s} is the factor of safety, which is usually 3 for two grid comparisons (Slater, 2006), fo_1 and fo_2 are the solution functions (i.e. velocity at a location) of the coarse grid and the fine grid respectively, f is the grid refinement ratio, and p_{o} is the formal order of accuracy of the convection scheme (i.e. UPWIND is 1st order therefore $p_{\mathrm{o}} = 1$). Sorensen and Nielsen (2003) illustrated the advantage of using this technique in building ventilation simulations. However, because this technique has been intentionally developed for uniform grids, it may not be suitable for application in cases where grid sizes vary considerably throughout the CFD domain (Brohus et al., 2006). Thus, the application of the GCI technique may be limited to simulations of slot ventilated enclosures or similar where the grid distribution is reasonably uniform. For CFD simulations outside this bracket, it is necessary to monitor solution progress, error field development, residual errors, and influence of grid density to ensure grid independence.

10. Conclusions

The state-of-the-art in CFD modelling and its application in ventilation system design in agriculture have been discussed. It is evident in this review that as the power of computers has increased so too has the level of sophistication of CFD packages. Therefore, it is necessary to exploit the power of CFD as best as possible in each simulation.

An examination of turbulence models used in ventilation system modelling has shown that the quality of a solution is highly dependant on the turbulence model. The standard k–ε model, which still commonly used, at times yields inadequate results and circumspective choice of other turbulence models should be made depending on the phenomena involved in the simulation.

The use of porous media models to simulate the pressure drop over flow restrictors such as insect nets and fences is popular in CFD simulations. These models used can be altered depending on the scale and requirements of the simulation, and the accuracy associated with their implementation depend on the choices made by the modeller.

To ensure CFD simulations are more than just theoretical exercises, experimental validation is necessary. Over recent years this has been carried both in the laboratory, using wind tunnel and salt water baths, and in full scale, using tracer gas or sonic anemometry. New technologies such as particle image velocitometry have also shown to complement CFD predictions. Validation has been successful in many cases and even in cases where discrepancies exist, deficiencies in the model or measurement technique were readily identifiable.

With regards to agricultural buildings, advances in CFD technology has meant that field solutions can now include dynamic biological responses of housed entities, and thereby enhance the realism of the simulations. Greenhouse CFD applications has been of a higher standard than animal housing CFD applications over recent years, owing to the incorporation of crop biological models in computations. Recent studies have adopted similar techniques in animal housing applications. Overall this review underlines the scope which CFD can offer to indoor environmental modelling and highlights the need for more studies to focus on facility design rather than phenomena observation.

References

Aerts, J.M., Berckmans, D., 2004. A virtual chicken for climate control design: static and dynamic simulations of heat losses. Transactions of the ASAE 47 (5), 1765–1772.

Anon., 2006. ANSYS CFX release 10.0 technical specifications. Public notice, ANSYS, Inc., Southpointe, Canonburg, PA, USA.

Axley, J.W., Chung, D.H., 2005. Mechanical power balances for multi-zone building airflow analysis. International Journal of Ventilation 4 (2), 95–112.

Baeza, E.J., Perez-Parra, J., Montero, J.I., 2005. Effect of ventilator size in natural ventilation in parral greenhouse by means of CFD simulations. Acta Horticulturae 691, 465–472.

Baptista, F.J., Bailey, B.J., Randall, J.M., Meneses, J.F., 1999. Greenhouse ventilation rate: theory and measurement with tracer gas techniques. Journal of Agricultural Engineering Research 72 (4), 363–374.

Bartzanas, T., Boulard, T., Kittas, C., 2002. Numerical simulation of the airflow and temperature distribution in a wind tunnel greenhouse equipped with insect-proof screen in openings. Computers and Electronics in Agriculture 34, 207–221.

Bartzanas, T., Boulard, T., Kittas, C., 2004. Effect of vent arrangement on windward ventilation of a tunnel greenhouse. Biosystems Engineering 88, 479–490.

Bjerg, B., Morsing, S., Svidt, K., Zhang, G., 1999. Three-dimensional airflow in a livestock test room with two-dimensional boundary conditions. Journal of Agricultural Engineering Research 74, 267–274.

Bjerg, B., Svidt, K., Zhang, G., Morsing, S., 2000. The effect of pen partitions and thermal pig simulators on airflow in a livestock test room. Journal of Agricultural Engineering Research 77, 317–326.

Boon, C.R., 1982. The effect of airspeed changes on the group postural behavior of pigs. Journal of Agricultural Engineering Research 27, 71–79.

Bottcher, R.W., Willits, D.H., 1987. Numerical computation of two-dimensional flow around and through a peaked-roof building. Transactions of the ASAE 30, 469–475.

Boulard, T., Wang, S., 2002. Experimental and numerical studies on the heterogeneity of crop transpiration in a plastic tunnel. Computers and Electronics in Agriculture 34 (1–3), 173–190.

Boulard, T., Meneses, J.F., Mermier, M., Papadakis, G., 1996. The mechanisms involved in the natural ventilation of greenhouses. Agricultural and Forest Meteorology 79 (1–2), 61–77.

Boulard, T., Papadakis, G., Kittas, C., Mermier, M., 1997. Air flow and associated sensible heat exchanges in a naturally ventilated greenhouse. Agricultural and Forest Meteorology 88 (1–4), 111–119.

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. Journal of Agricultural Engineering Research 74, 135–144.

Boulard, T., Kittas, C., Roy, J.C., Wang, S., 2002. Convective and ventilation transfers in greenhouses, part 2: determination of the distributed greenhouse climate. Biosystems Engineering 83, 129–147.

Brohus, H., Balling, K.D., Jeppesen, D., 2006. Influence of movements on contaminant transport in an operating room. Indoor Air 16, 356–372.

Bruce, J.M., Clark, J.J., 1979. Models of heat production and critical temperature for growing pigs. Animal Production 28 (3), 353–358.

Brugger, M., Montero, J., Baezz, E., Perez-Parra, J., 2005. Computational fluid dynamic modelling to improve to design of the Spanish parral style greenhouse. Acta Horticulturae 691, 425–432.

Campen, J.B., 2005. Greenhouse design applying CFD for Indonesian conditions. Acta Horticulturae 691, 419–424.

Campen, J.B., Bot, G.P.A., 2003. Determination of greenhouse-specific aspects of ventilation using three-dimensional computational fluid dynamics. Biosystems Engineering 84, 69–77.

Chang, W.R., 2006. Effect of porous hedge on cross ventilation of a residential building. Building and Environment 41, 549–556.

Chen, Y.S., Kim, S.W., 1987. Computation of turbulent flows using an extended k–ε model. NASA report, CR-179204, Durbin, PA, USA.

Choi, K., Albright, L.D., Timmons, M.B., 1988. An application of the k–e turbulence model to predict air distribution in a slot ventilated enclosure. Transactions of the ASAE 31, 1804–1814.

Choi, K., Albright, L.D., Timmons, M.B., 1990. An application of the k–e turbulence model to predict how a rectangular obstacle in a slot-ventilated enclosure affects airflow. Transactions of the ASAE 33, 274–2781.

Cook, N.J., 1990. The Designers Guide to Wind Loading of Building Structures, Part 2: Static Structures. Butterworths, London, UK.

Choiniere, Y., Munroe, J.A., 1994. A wind-tunnel study of wind direction effects on air-flow patterns in naturally ventilated swine buildings. Canadian Agricultural Engineering 36 (2), 93–101.

Davies, P.A., Paton, C., 2005. The seawater greenhouse in the United Arab Emirates: thermal modelling and evaluation of design options. Desalination 173, 103–111.

Demmers, T.G.M., Burgess, L.R., Phillips, V.R., Clark, J.A., Wathes, C.M., 2000. Assessment of techniques for measuring the ventilation rate, using an experimental building section. Journal of Agricultural Engineering Research 76 (1), 71–81.

Desta, T.Z., Janssens, K., Van Brecht, A., Meyers, J., Baelmans, M., Berckmans, D., 2004. CFD for model-based controller development. Building and Environment 39 (6), 621–633.

Etheridge, D.W., Sandberg, M., 1996. Building Ventilation Theory and Measurement. John Wiley and Sons, UK.

Fath, H.E.S., Abdelrahman, K., 2005. Micro-climatic environmental conditions inside a greenhouse with a built-in solar distillation system. Desalination 171, 267–287.

Fatnassi, H., Boulard, T., Bouirden, L., 2003. Simulation of climatic conditions in full-scale greenhouse fitted with insect-proof screens. Agricultural and Forest Meteorology 118, 97–111.

Fatnassi, H., Boulard, T., Poncet, C., Chave, M., 2006. Optimisation of greenhouse insect screening with computational fluid dynamics. Biosystems Engineering 93, 301–312.

Ferry, M., 2002. New features of the MIGAL solver. In: Proceedings of the 9th PHOENICS User Conference, Moscow Power Engineering Institute, Moscow, Russia. pp. 1–23.

Ferziger, J.H., Peric, M., 2002. Computational Methods for Fluid Dynamics, third ed. Springer-Verlag, Berlin, Germany.

Foster, A.M., Barrett, R., James, S.J., Swain, M.J., 2002. Measurement and prediction of air movement through doorways in refrigerated rooms. International Journal of Refrigeration 25, 1102–1109.

Fracastoro, G.V., Perino, M., 1999. Numerical Simulation of Transient Effects of Window Openings. In: Proceedings of First International One day Forum on Natural and Hybrid Ventilation, HybVent Forum'99, Sydney, Australia.

Friedrich, R., Hüttl, T.J., Manhart, M., Wagner, C., 2001. Direct numerical simulation of incompressible turbulent flows. Computers and Fluids 30, 555–579.

Gan, G., 1994. Numerical method for a full assessment of indoor thermal comfort. Indoor Air 4, 154–168.

Gebremedhin, K.G., Wu, B.X., 2001. Numerical simulation of flow field around a cow using 3-D body-fitted coordinate system. Journal of Thermal Biology 26, 563–573.

Gebremedhin, K.G., Wu, B.X., 2003. Characterization of flow field in a ventilated space and simulation of heat exchange between cows and their environment. Journal of Thermal Biology 28, 301–319.

Gebremedhin, K.G., Wu, B., 2005. Simulation of flow field of a ventilated and occupied animal space with different inlet and outlet conditions. Journal of Thermal Biology 30, 343–353.

Gosman, A.D., Nielsen, P.V., Restivo, A., Whitelaw, J.H., 1980. The flow properties of rooms with a small ventilation opening. Journal of Fluids Engineering 102, 316–326.

Harral, B.B., Boon, C.R., 1997. Comparison of predicted and measured air flow patterns in a mechanically ventilated livestock building without animals. Journal of Agricultural Engineering Research 66, 221–228.

Harral, B., Burfoot, D., 2005. A comparison of two models for predicting the movements of airborne particles from cleaning operations. Journal of Food Engineering 69, 443–451.

Hauser, R.H., Folsch, D.W., 1988. Measures for measuring ammonia in poultry houses: review and approved practices. Journal of Veterinary Medicine 35, 579–593.

Heber, A.J., Boon, C.R., 1993. Air velocity characteristics in an experimental livestock building with nonisothermal jet ventilation. ASHRAE Transactions 99, 1139–1151.

Heber, A.J., Boon, C.R., Peugh, M.W., 1996. Air patterns and turbulence in an experimental livestock building. Journal of Agricultural Engineering Research 64, 209–226.

Hoang, M.L., Verboven, P., De Baerdemaeker, J., Nicolaï, B.M., 2000. Analysis of the air flow in a cold store by means of computational fluid dynamics. Journal of Refrigeration 23, 127–140.

Hoerner, S.F., 1965. Fluid Dynamic Drag: Practical Information on Aerodynamic Drag and Hydrodynamic Resistance. Hoerner Fluid Dynamic, Bakersfield, CA, USA.

Hoff, S.J., 2004. Automated control logic for naturally ventilated agricultural structures. Transactions of the ASAE 20 (1), 47–56.

Hoff, S.J., Janni, K.A., Jacobson, L.D., 1992. Three-dimensional buoyant turbulent flows in a scaled model, slot-ventilated, livestock confinement facility. Transactions of the ASAE 35, 671–686.

Hoff, S.J., Li, L., Tsao, L., 1995. Simulated and measured effect of rectangular obstructions on carbon dioxide gas dispersion in a scaled swine building. Transactions of the ASAE 38 (5), 1519–1532.

Jiang, Y., Chen, Q., 2002. Effect of fluctuating wind direction on cross natural ventilation in buildings from large eddy simulation. Building and Environment 37, 379–386.

Kacira, M., Sase, S., Okushima, L., 2004. Optimization of vent configuration by evaluating greenhouse canopy ventilation rates under wind-induced ventilation. Transactions of the ASAE 47 (6), 2059–2067.

Kato, S., 2004. Flow network model based on power balance as applied to cross-ventilation. International Journal of Ventilation 2, 395–408.

Katsoulas, N., Bartzanas, T., Boulard, T., Mermier, M., Kittas, C., 2006. Effect of vent openings and insect screens on greenhouse ventilation. Biosystems Engineering 93, 427–436.

Kavanagh, N.T., 2003. Pig herd health and production in Ireland: a study of some major influences. Thesis submitted for a fellowship of the Royal College of Veterinary Surgeons, 62–64 Belgravia House, Horseferry Road, London SW1P 2AF, UK.

Khankari, K., Karki, K., Patankar, S., Janni, K., 1997. Computational analysis of ceiling inlet ventilation systems for livestock buildings. In: Proceedings of the fifth International Symposium of Livestock Environment, Minnesota, USA, pp. 32–39.

Kittas, C., Boulard, T., Papadakis, G., 1997. Natural ventilation of a greenhouse with ridge and side openings: sensitivity to temperature and wind effects. Transactions of the ASAE 40 (2), 415–426.

Kopyt, P., Gwarek, W., 2004. A comparison of commercial CFD software capable of coupling to external electromagnetic software for modeling of microwave heating process. Proceedings of the 6th Seminar in Computer Modelling and Microwave Power Engineering, Austin, Texas, USA. Worcester Polytechnic Institute, Worcester, MA, pp. 33–39.

Kuznik, F., Rusaouen, G., Brau, J., 2006. Experimental and numerical study of a full scale ventilated enclosure: comparison of four two equations closure turbulence models. Building and Environment 39 (8), 871–881.

Lee, I., You, B.K., Kang, C.H., Jeun, J.G., Kim, G.W., Sung, S.H., Sase, S., 2004. Study on forced ventilation of piglet house. Japan Agricultural Research Quarterly 38, 81–90.

Lee, I., Lee, S., Kim, G., Sung, J., Sung, S., Yoon, Y., 2005. PIV verification of greenhouse ventilation air flows to evaluate CFD Accuracy. Transactions of the ASAE 48 (6), 2277–2288.

Liu, Y., Moser, A., Gubler, D., Schaelin, A., 2003. Influence of time step length and sub-iteration number on the convergence behavior and numerical accuracy for transient CFD. In: Proceedings of the 11th Annual Conference of the CFD Society of Canada, pp. 480–485.

Loomans, M., 1998. The Measurement and Simulation of Indoor Air Flow. Ph.D. Thesis, Eindhoven University of Technology, Eindhoven, The Netherlands.

Miguel, A.F., van de Braak, N.J., Silva, A.M., Bot, G.P.A., 1998. Physical modelling of natural ventilation through screens and windows in greenhouses. Journal of Agricultural Engineering Research 70, 165–176.

Mirade, P.S., 2003. Prediction of the air velocity field in modern meat dryers using unsteady computational fluid dynamics (CFD) models. Journal of Food Engineering 60, 41–48.

Mirade, P.S., Rougier, T., Daudin, J.D., Picque, D., Corrieu, G., 2006. Effect of design of blowing duct on ventilation homogeneity around cheeses in a ripening chamber. Journal of Food Engineering 75, 59–70.

Mistriotis, A., Briassoulis, D., 2002. Numerical estimation of the internal and external aerodynamic coefficients of a tunnel greenhouse structure with openings. Computers and Electronics in Agriculture 34 (1–3), 191–205.

Mistriotis, A., Bot, G.P.A., Picuno, P., Scarascia-Mugnozza, G., 1997a. Analysis of the efficiency of greenhouse ventilation using computational fluid dynamics. Agricultural and Forest Meteorology 85, 217–228.

Mistriotis, A., Bot, G.P.A., Picuno, P., Scarascia-Mugnozza, G., 1997b. Computational analysis of ventilation in greenhouses at zero- and low-wind-speeds. Agricultural Forest Meteorology 88, 121–135.

Mistriotis, A., de Jong, T., Wagemans, M.J.M., Bot, G.P.A., 1997c. Computational fluid dynamics as a tool for the analysis of ventilation and indoor microclimate in agricultural buildings. Netherlands Journal of Agricultural Science 45, 81–96.

Molina-Aiz, F.D., Valera, D.L., Pena, A.A., Gil, J.A., 2005. Optimisation of Almeria-type greenhouse ventilation performance with computational fluid dynamics. Acta Horticulturae 691, 433–440.

Montero, J.I., Anton, A., Kamarudddin, R., Bailey, B.J., 2001a. Analysis of thermally driven ventilation in tunnel greenhouse using small scale models. Journal of Agricultural Engineering Research 79 (2), 213–222.

Montero, J.I., Hunt, G.R., Kamaruddin, R., Antón, A., Bailey, B.J., 2001b. Effect of ventilator configuration on wind-driven ventilation in a crop protection structure for the tropics. Journal of Agricultural Engineering Research 80, 99–107.

Montero, J.I., Munoz, P., Anton, A., Iglesias, N., 2005. Computational fluid dynamic modelling of night-time energy fluxes in unheated greenhouses. Acta Horticulturae 691, 403–409.

Moureh, J., Flick, D., 2005. Airflow characteristics within a slot-ventilated enclosure. International Journal of Heat and Fluid Flow 26, 12–24.

Moureh, J., Menia, N., Flick, D., 2002. Numerical and experimental study of airflow in a typical refrigerated truck configuration loaded with pallets. Computers and Electronics in Agriculture 34, 25–42.

Murakami, S., 1998. Overview of turbulence models applied in CWE-1997. Journal of Wind Engineering and Industrial Aerodynamics 74–76, 1–24.

Nielsen, J.R., Nielsen, P.V., Svidt, K., 1997. Air distribution in a furnished room ventilated by mixing ventilation. In: Proceedings of Ventilation‘97, 5th International Symposium on Ventilation for Contaminant Control, Ottawa, Canada.

Nielsen, J.R., Nielsen, P.V., Svidt, K., 1998. The influence of furniture on air velocity in a room – an isothermal case. In: Proceedings of Room Vent’98, Sixth International Conference on Air Distribution in Rooms, Stockholm, Sweden, KTH. pp. 281–286.

Oca, J., Montero, J.I., Anton, A., Crespo, D., 1998. A method for studying natural ventilation by thermal effects in a tunnel greenhouse using laboratory-scale models. Journal of Agricultural Engineering Research 72, 93–104.

Okushima, L., Sase, S., Nara, M., 1989. A support system for natural ventilation design of greenhouses based on computational aerodynamics. Acta Horticulturae 284, 129–136.

Ould Khaoua, S.A., Bournet, P.E., Migeon, C., Boulard, T., Chasseriaux, G., 2006. Analysis of greenhouse ventilation efficiency based on computational fluid dynamics. Biosystems Engineering 95 (1), 83–98.

Panagakis, P., Axaopoulos, P., 2004. Comparison of two modeling methods for the prediction of degree-hours and heat-stress likelihood in a swine building. Transactions of the ASAE 47 (2), 585–590.

Patankar, S.V., Spalding, D.V., 1972. A calculation procedure for heat, mass and momentum transfer in three-dimensional parabolic flows. International Journal for Heat and Mass Transfer 15, 1787–1806.

Posner, J.D., Buchanan, C.R., Dunn-Rankin, D., 2003. Measurement and prediction of indoor air flow in a model room. Energy and Buildings 35, 515–526.

Predicala, B.Z., Maghirang, R.G., 2003. Numerical simulation of particulate matter emissions from mechanically ventilated swine barns. Transactions of the ASAE 46, 1685–1694.

Reichrath, S., Davies, T.W., 2002a. Computational fluid dynamics simulations and validation of the pressure distribution on the roof of a commercial multi-span Venlo-type glasshouse. Journal of Wind Engineering and Industrial Aerodynamics 90, 139–149.

Reichrath, S., Davies, T.W., 2002b. Using CFD to model the internal climate of greenhouses: past, present and future. Agronomie 22, 3–19.

Reynolds, A.M., 1997. A model for predicting dust concentrations within a ventilated airspace. Journal of Agricultural Engineering Research 66, 103–109.

Richards, P.J., Hoxey, R.P., 1993. Appropriate boundary conditions for computational wind engineering models using the k–ε turbulence model. Journal of Wind Engineering and Industrial Aerodynamics 46–47, 145–153.

Richards, P.J., Hoxey, R.P., 2006. Flow reattachment on the roof of a 6 m cube. Journal of Wind Engineering and Industrial Aerodynamics 94, 77–99.

Rouaud, O., Havet, M., 2002. Computation of the airflow in a pilot scale clean room using k–ε turbulence models. International Journal of Refrigeration 25, 351–361.

Roache, P.J., 1998. Verification and Validation in Computational Science and Engineering. Hermosa, Albuquerque, New Mexico.

Roy, J.C., Boulard, T., 2005. CFD predictions of the natural ventilation in a tunnel type greenhouse: influence of wind direction and sensitivity to turbulence models. Acta Horticulturae 691, 457–464.

Roy, J.C., Boulard, T., Kittas, C., Wang, S., 2002. Convective and ventilation transfers in greenhouses, part 1: the greenhouse considered as a perfectly stirred tank. Biosystems Engineering 83, 1–20.

Sase, S., Takakura, T., 1984. Wind tunnel testing on airflow and temperature distribution of a naturally ventilated greenhouse. Acta Horticulturae 148, 329–336.

Schalin, A., Nielsen, P.V., 2004. Impact of turbulence anisotropy near walls in room airflow. Indoor Air 14, 159–168.

Schauberger, G., Piringer, M., Petz, E., 2000. Steady state balance model to calculate the indoor climate of livestock buildings, demonstrated for finishing pigs. International Journal of Biometeorology 43, 154–162.

Shklyar, A., Arbel, A., 2004. Numerical model of the three-dimensional isothermal flow patterns and mass fluxes in a pitched-roof greenhouse. Journal of Wind Engineering and Industrial Aerodynamics 92, 1039–1059.

Slater, J.W., 2006. Examining spatial (Grid) convergence. Public tutorial on CFD verification and validation, NASA Glenn Research Centre, MS 86 and 21000 Brookpark Road, Cleveland, OH 44135.

Swaddiwudhipong, S., Khan, M.S., 2002. Dynamic response of a wind-excited building using CFD. Journal of Sound and Vibration 253, 735–754.

Simiu, E., Scanlan, R.H., 1996. Winds Effects on Structures. Wiley, UK.

Sorensen, D.N., Nielsen, P.V., 2003. Quality control of computational fluid dynamics in indoor environments. Indoor Air 13, 2–17.

Spoolder, H.A.M., Edwards, S.A., Armsby, A.W., Corning, S., 2000. A within farm comparison of three different housing systems for finishing pigs. In: Proceedings of the First International Conference on Swine Housing, Des Moines, Iowa, pp. 40–48.

Stamou, A., Katsiris, I., 2006. Verification of a CFD model for indoor airflow and heat transfer. Building and Environment 41, 1171–1181.

Sun, H., Stowell, R., Keener, H., Elwell, D., Michel, F., 2002. Two-dimensional computational fluid dynamics modelling of air velocity and ammonia distribution in a high rise hog building. Transactions of the ASAE 45, 1559–1568.

Sun, Y., Tan, Z., Zhang, Y., Zhao, L., 2004a. Comparison of six CFD models for room airflow studies with PIV measurement data. Presented at 2004 ASAE/CSAE Annual International Meeting, Paper Number: 044097, American Society of Agricultural and Biological Engineers, St. Joseph.

Sun, H., Keener, H., Deng, W., Michel, Jr. F., 2004b. Development and validation of 3-D CFD models to simulate airflow and ammonia distribution in a high-rise hog building during summer and winter conditions. Agricultural Engineering International: the CIGR Journal of Scientific Research and Development. Manuscript BC 04 004. Vol. VI.

Svidt, K., Zhang, G., Bjerg, B., 1998. CFD simulation of air velocity distribution in occupied livestock buildings. In: Proceedings of Sixth International Conference on Air Distribution in Rooms, Stockholm, Sweden. pp. 491–496.

Takle, E.S., Wang, H., 1997. Reply to comments by J.D. Wilson and C.J. Mooney on a numerical simulation of boundary-layer flows near shelterbelts. Boundary-Layer Meteorology 85, 151–159.

Timmons, M.B., 1984. Use of physical models to predict the fluid motion in slot-ventilated livestock structures. Transactions of ASAE 27 (2), 502–507.

Tinôco, I.F.F., Zanolla, N., Alvarenga e Melo, R.C., Baêtal, F.C., Tinôco, A.L.A., Yanagi, Jr. T., Moraes, S.R.P., Silva, J.N., 2001. Effect of different ventilation systems on hot weather thermal comfort and performance of broiler chickens raised at high placement. In: Livestock Environment VI: Proceedings of the 6th International Symposium, Louisville, Kentucky, USA, pp. 250–255.

van Brecht, A., Vranken, E., Guarino, M., Berckmans, D., 2004. Optical flow algorithm to quantify the two-dimensional velocity components of a visualised air jet. Transactions of the ASAE 47 (3), 847–856.

van Ouwerkerk, E.N.J., Voskamp, J.P., Aliskan, Y., 1994. Climate simulation and validation for an aviary system for laying hens Institute of Agricultural and Environmental Engineering, Report N 94-C-063, AgENG, Milan, Italy.

van Wagenberg, A.V., Bjerg, B., Bot, G.P.A., 2004. Measurements and simulation of climatic conditions in the animal occupied zone in a door ventilated room for piglets. Agricultural Engineering International: The CIGR Journal of Scientific Research and Development, Manuscript BC 03 020, Vol VI.

Verboven, P., Flick, D., Nicolaï, B.M., Alvarez, G., 2006. Modelling transport phenomena in refrigerated food bulks, packages and stacks: basics and advances. International Journal of Refrigeration 29 (6), 985–997.

Verlinde, W., Gabriels, D., Christiaens, J.P.A., 1998. Ventilation coefficient for wind-induced natural ventilation in cattle buildings. Transactions of the ASAE 41 (3), 783–788.

Versteeg, H.K., Malalsekeera, W., 1995. An Introduction to Computational Fluid Dynamics. Longman Group Ltd, Harlow, UK.

von Wachenfelt, E., 1994. Modern techniques give less pollution in broiler houses. In: Proceedings of the XII World Congress on Agricultural Engineering, Milan CIGR, Merelbeke, Belguim, pp. 590–598.

Wang, S., Deltour, J., 1999. Lee-side ventilation-induced air movement in a large-scale multi-span greenhouse. Journal of Agricultural Engineering Research 74 (1), 103–110.

Wang, S., Boulard, T., Haxaire, R., 1999. Air speed profiles in a naturally-ventilated greenhouse with a tomato crop. Agricultural and Forest Meteorology 96 (4), 181–188.

Wilson, J.D., 1985. Numerical studies of flow through a windbreak. Journal of Wind Engineering and Industrial Aerodynamics 21, 119–154.

Wilson, J.D., Mooney, C.J., 1997. Comments on a 'Numerical simulation of boundary-layer flows near shelterbelts' by H. Wang and E. Takle. Boundary-Layer Meteorology 85, 137–149.

Wright, N.G., Easom, G.J., 2003. Non-linear k–ε turbulence model results for flow over a building at full-scale. Journal of Wind Engineering and Industrial Aerodynamics 27, 1013–1033.

Worley, M.S., Manbeck, H.B., 1995. Modelling particle transport and air flow in ceiling inlet ventilation systems. Transactions of the ASAE 38 (1), 231–239.

Xia, B., Sun, D.-W., 2002. Applications of computational fluid dynamics (CFD) in the food industry: a review. Computers and Electronics in Agriculture 34, 5–24.

Yakhot, V., Orszag, S.A., 1986. Renormalization group analysis of turbulence: I. basic theory. Journal of Scientific Computing 1, 1–51.

Yu, H., Hoff, S.J., 1999. Airflow pattern similarity criteria for ceiling slot–ventilated enclosures under isothermal conditions. Transactions of the ASAE 42 (2), 459–469.

Zhang, G., Svidt, K., Bjerg, B., Morsing, S., 1999. Buoyant flow generated by thermal convection of a simulated pig. Transactions of the ASAE 42, 1113–1120.

Zhang, G., Morsing, S., Bjerg, B., Svidt, K., Strøm, J.S., 2000. Test room for validation of airflow patterns estimated by computational fluid dynamics. Journal of Agricultural Engineering Research 76, 141–148.