Netherlands Journal of Agricultural Science 45 (1997) 81-96

Computational Fluid Dynamics (CFD) as a tool for the analysis of ventilation and indoor microclimate in agricultural buildings

A. MISTRIOTIS [1,2,*], T. DE JONG [1], M.J.M. WAGEMANS [1], AND G.P.A. BOT [1,2]

[1] Institute of Agricultural and Environmental Engineering (IMAG-DLO) P.O. Box 43, NL-6700 AA Wageningen, The Netherlands.
[2] Department of Agricultural Engineering and Physics, Wageningen Agricultural University, Bomenweg 4, NL-6703 HD Wageningen, The Netherlands.
[*] corresponding author (fax: +31-317-4425670; e-mail: mistriot@sun2.aenf.wau.nl)

Received 19 September 1996; accepted 19 April 1997

Abstract

The basic concepts of Computational Fluid Dynamics (CFD) are presented in relationship with an application in modeling the ventilation process and the resulting indoor climate of agricultural buildings. The validity and the advantages of this numerical technique are presented by three examples. First the pressure coefficients along the roof of a 7-span Venlo-type greenhouse were calculated and compared with the corresponding experimental values. Then the ventilation process in a single-span greenhouse was investigated and the results were compared to laboratory experiments. Finally, the use of CFD as a design tool for more efficient ventilation systems was demonstrated in the case of a broiler house.

Keywords: ventilation, greenhouse, animal housing, computational fluid dynamics (CFD)

Introduction

The development of environmentally and economically sustainable agriculture has become one of the main objectives in agricultural research. In this context, research efforts focus on the optimisation of methods, systems and primary production chains. The promotion of animal and crop production methods and systems which permit a better protection of the environment or improvements of the product quality are important examples. In order to improve the crop and animal production systems, more knowledge about the spatial distribution of climatic variables within the growing or housing system is required.

For instance, in horticulture the quality of the product and the development of diseases is closely related to the local temperature and humidity within the greenhouse. A uniform climate in the greenhouse could significantly reduce the use of pesticides by avoiding local conditions which are favourable for diseases. The uniform indoor climate conditions lead to simultaneous growing and production of the crop as well as a uniform product quality, which facilitates the organisation of the harvest. In the future, refined control of the climatic variables could make possible the steering of important plant processes such as transpiration or photosynthesis (Stanghellini & van Meurs, 1992). Direct manipulation of crop growth via controlling the local environmental conditions, could eventually lead to higher standards of quality and more efficient production (Cockshull, 1988).

Livestock confinement housing raises concern with respect to the air quality in the animal and worker environment. Air flow direction and intensity in the animal occupied zone directly influence the effective room temperature and the well being of the occupants and their performance. Dust and other contaminants also affect animal health and production, as well as the health of the workers (Carpenter, 1986; Xin *et al.*, 1996). The concentration of airborne particles is strongly correlated with environmental factors such as ventilation parameters and local air velocity.

A systematic investigation of the ventilation, cooling and/or heating schemes is required for improving building design and climate systems towards higher efficiency and lower energy consumption. Since field testing is both difficult and expensive, the use of accurate simulations would greatly assist the design of improved systems. Advances in digital computing speed and capacity have made possible the numerical solution of non-linear differential equations for fluid flows including wind-induced flows. It involves the numerical solution of the continuity equation and the conservation equations for energy and momentum. It can also include modelling of particle transport by numerically solving the corresponding conservation equation of particles concentration.

The main attractive feature of Computational Fluid Dynamics (CFD) is its potential to assist investigations of large scale structures of 3D flows, allowing incorporation of realistic boundary conditions and obstructions. The present article demonstrates by examples the advantages offered by CFD simulations in analysing the ventilation process in greenhouses and animal houses. CFD allows the explicit calculation of the average air velocity field. Therefore, not only the ventilation rate due to the average air flow can be determined but also the details of the ventilation mechanism and its consequences on the microclimate can be understood.

In the present study the emphasis is given to examples which show the power of CFD as a design tool for buildings. CFD, however, can also be used for other problems related to the spatial distribution of heat and mass transfer in agricultural engineering.

Materials and methods

The Computational Fluid Dynamics (CFD) method allows the explicit calculation of the average velocity vector field of a flow by numerically solving the corresponding transport equations. The balance equations describing these transport phenomena are of the general form:

$$\frac{\partial \varphi}{\partial t} + \vec{\nabla} \cdot \varphi \vec{v} = \vec{\nabla} \cdot \Gamma_\varphi \vec{\nabla} \varphi + S_\varphi \tag{1}$$

where $\vec{v}$ is the velocity vector, Γ_φ is the diffusion coefficient and S_φ is the source term. The symbol φ represents the concentration of the transported quantity. In the total mass balance (continuity) equation, φ corresponds to the density ρ. In a flow field momentum is transported and corresponds to $\rho\vec{v}$. Equation (1) then represents three equations, namely the three momentum conservation equations corresponding to the three components of $\rho\vec{v}$. If energy is also transported in the flow field, an extra equation where φ corresponds to $\rho c_p T$, describes the temperature field.

In a flow field, the continuity equation and the three momentum equations describe the velocity components and the pressure as functions of time and space. When energy is transported, an extra equation describes the temperature field. Solving analytically the set of continuity, momentum and energy equations is only possible for very simple problems. The use of numerical techniques is indispensable for problems with more complex nature.

The most common technique for numerically integrating these partial differential equations is the finite differences method. It consists of discretising the space and time (for transient cases) and solving the transport equations on the spatial grid as difference equations. The solution is obtained for a finite domain which is spanned by the grid. The influence of external factors such as the wind, the solar radiation etc., on the flow is simulated by boundary conditions. In this way, not only the ventilation rate of a building can be obtained but also the details of the average internal flow can be examined.

Both wind driven and thermally driven ventilation can be analysed by CFD simulations and the influence of the building design on the indoor air flow can be systematically investigated. The success of CFD models in simulating the air flow in buildings, greatly depends on the correct modelling of the turbulent transport and the realistic description of the boundary conditions. Several software packages are currently available for solving Fluid Dynamics problems. In the present work the CFD code PHOENICS v2.1 was used (Anonymous, 1994b).

Modelling turbulent air flows.

All the turbulence models implemented in the CFD method avoid the complexity of turbulent dynamics by decomposing the turbulent flow conservation into a slowly varying and a fluctuating component. Therefore, the velocities in the equations (1) are replaced by the sum of a mean component and a fluctuating component. The fluctuating component of the velocity is called the turbulent velocity. If the transport equations are averaged over time, all the terms corresponding to the fluctuating part of the flow are eliminated except for the turbulent diffusion term. Therefore, an extra term describing the turbulent transport is added to the laminar diffusion term, so we can call that turbulent diffusion term. Hence it is possible to replace the viscosity by an effective value, which is the sum of the laminar viscosity and a phenomenological parameter called 'turbulent viscosity'. Consequently, the problem of modelling the turbulent flow is reduced in calculating the turbulent viscosity. (e.g. Patankar, 1980; Awbi, 1991).

During the recent years, the most popular model describing turbulent viscosity is the k-ε model (Harlow & Nakayama, 1968; Launder & Spalding, 1974). In this model, it is expressed in terms of two phenomenological variables, the turbulence kinetic energy k, and its dissipation rate ε. The model adds two extra transport equations corresponding to the two new variables k and ε, to the usual transport equations describing the flow. Therefore, in this two-equation model no empirically defined spatially dependent parameters are required. Hence, complex flows governed by elliptic equations, such as circulating flows can be solved. The k-ε model has been applied to numerous indoor air flow problems with good predictive accuracy (e.g. Awbi, 1991).

Despite its success, the k-ε model has some limitations. An important weakness of the k-ε model is that it assumes spectral equilibrium. In other words, once turbulent energy is generated at the small wave number end of the spectrum (large eddies), it is equally distributed to the whole spectrum. Generally, this is not the case, because the transfer of energy from the large eddies where turbulence is produced to the small eddies where turbulence dissipation occurs, is not automatic.

The above described problem was treated by the introduction of the two-scale k-ε models. One of the two-scale k-ε models is the Chen and Kim (CK) modification of the standard k-ε model (Chen & Kim, 1987). This improves the dynamic response of the equation for k by introducing a second time scale: k/p, where p is the volumetric production rate of k. In addition, several of the standard model coefficients are adjusted so that the models maintain good agreement with experimental data on classical turbulent shear layer flows. This model is particularly successful in describing separating and reattaching flows. This is exactly the case when the wind forces its way through the open windows of a building. Recent results show that the CK model is quite successful in simulating ventilation flows in greenhouses (Mistriotis *et al.*, 1996). The model parameters used in the CK model are given in (Anonymous, 1994b).

Boundary conditions

CFD calculations always simulate a flow in a finite domain. Therefore, the definition of realistic boundary conditions is crucial for correctly reproducing the flow characteristics. In the CFD simulations of the ventilation in agricultural buildings, the boundary conditions describe the incoming wind. Hence, the wind velocity profile, namely the height dependence of the wind velocity, and the turbulent characteristics of the wind should be correctly simulated.

The wind characteristics depend on the terrain type around an agricultural building. It has been shown (Richards & Hoxey, 1993) that the wind velocity profile is described by a logarithmic function of the form:

$$v(z) = \frac{v_f}{K} \ln\left(\frac{z + z_o}{z_o}\right) \qquad (2)$$

where v(z) is the wind velocity as a function of the height z, v_f is the friction velocity, K is the von Karman constant and z_o is the surface roughness length. The constant K has been determined to be equal to 0.42±0.02 (Richards & Hoxey, 1993). The surface roughness length depends on the terrain type. A few typical values given by the Eurocode (Anonymous, 1995) on wind actions on structures are presented in Table

Table 1. Terrain categories and related surface friction length values (Anonymous, 1995).

Terrain type	z_o
Rough open area without obstacles	0.01
Farmland	0.05
Suburban or industrial areas	0.30
Urban areas	1.00

1. The friction velocity v_f can be determined by the measurement of the wind velocity at a reference height.

The turbulent characteristics of the wind at the upstream boundary are described by the values of the k and ε variables. It has been shown (Richards & Hoxey, 1993) that if

$$k = \frac{v_f^2}{\sqrt{C_\mu}} \tag{3}$$

and

$$\varepsilon = \frac{v_f^3}{K(z + z_o)} \tag{4}$$

then k and ε conservation equations of the k-ε model are satisfied together with equation (1). Therefore equations (2-4) describe the wind characteristics and their dependence on the surface friction length z_o.

A comparison between experimental and numerical (CFD) results is possible only if the surface roughness length of the terrain around the studied agricultural building is known. However, this requires complicated and sensitive measurements (McAneny *et al.*, 1987; Richards & Hoxey, 1993) and for this reason such data rarely accompany ventilation measurements.

In the case of low wind speeds (wind velocity below 1 m/s) buoyancy forces play an important role in the ventilation process. Pressure differences due to temperature differences are comparable to the pressure gradient induced by the wind. Therefore, the transport equation for enthalpy must be solved coupled with the mass and momentum transport equations. In this case boundary conditions for the temperature or the heat flux must be defined.

There are several heat sources in an agricultural building, both natural and artificial such as the absorbed solar radiation, a heating system or the body heat of animals. A part of this energy is transformed into latent heat, while the remaining is released to the air as sensible heat. Usually, the sensible heat flux intensity of these heat sources can be measured and the value can be used for defining the corresponding boundary conditions of a CFD simulation for calculating the ventilation flow. However, living organisms such as animals or crops, exhibit a dynamic response to climate changes. Therefore, the sensible heat exchange between living bodies and the air is hard to determine. Nevertheless typical values of the heat fluxes can be assigned in order to simplify the simulation. In cases where the temperature of a heat source is constant (e.g. heating pipes), a fixed temperature boundary condition can be defined.

Results

Analysis of the greenhouse ventilation using CFD
Efficient greenhouse ventilation is crucial both for northern humid winter climates and for Mediterranean hot summer conditions. Under hot weather conditions, ventilation can moderate high indoor temperatures and regulate relative humidity. Similarly, under cool conditions, ventilation reduces excessive humidity levels in order to prevent crop mineral depletion and fungal diseases. Therefore, ventilation is essential for maintaining optimal plant photosynthetic and transpiration activity, and healthy crops. CFD simulations are a powerful tool for investigating greenhouse ventilation efficiency. In this section we present a few typical results demonstrating the accuracy of the CFD method and its limitations.

Pressure coefficients
The wind generated pressure distribution around a building is the driving force of the ventilation when the buoyancy effect is negligible. Therefore, correct predictions of pressure differences by CFD simulations are an indication that the ventilation process is also modelled accurately.

The pressure distribution around a building is usually described by the dimensionless pressure coefficient c_p, which is defined as:

$$P_i - P_o = \frac{1}{2} c_p \rho v_i^2 \tag{5}$$

where $P_i - P_o$ is the difference between internal and external pressure at a point i on the greenhouse cover, v_i is the wind velocity the height of point i and ρ is the air density. Several full scale measurements of pressure coefficients on Venlo-type greenhouses have been performed at Silsoe Institute of Agricultural Engineering, Great Britain. In particular, the pressure coefficients due to a transverse wind along the roof of a 7-span standard Venlo-type greenhouse were extensively measured (Wells & Hoxey, 1980). The span width of the studied greenhouse was 3.2 m, its ridge height was 3.1 m and the height of its gutters was 2.35 m. The length of the greenhouse was 63 m. The wind velocity profile of the greenhouse location has been studied (Richards & Hoxey, 1993) and the surface roughness length has been determined to be 0.01 m. This information enables a comparison between numerical results and experimental data.

The 7-span Venlo-type greenhouse was modelled and studied by CFD simulations. The CK model was used. Since the case of a transverse wind (normal to the ridge) was examined in detail during the experiments, the same case was also investigated numerically. The pressure coefficients at the middle transverse section of the greenhouse is weakly influenced by 'end effects' because its length is large. Therefore, a two-dimensional CFD simulation is sufficient for computing the pressure coefficients. A variable orthogonal grid was used for modelling the greenhouse. The grid was selected fine enough to allow the geometry of the roof to be modelled. The height from gutter to ridge is spanned by 14 grid cells, while the height from ground to gutter by 10 cells. The width of every greenhouse span is spanned by 28 cells. The total grid has 257×59 cells. In this way, the wind flow above the roof is simulated with the accuracy of the available experimental data. Further refinement of the grid shows detectable differences in the value of pressure coefficients only at the ridges. The wind speed at the reference height (10m) was taken equal to 2 m/s. A logarithmic wind velocity profile was considered where the surface friction length is 0.01.

Figure 1 presents both the expèrimental and the numerical values of the pressure coefficient along the roof of the 7-span Venlo-type greenhouse when a transverse wind is considered. A good agreement is observed between numerical and experimental data along the first three spans of the greenhouse roof, where experimental data are available. However, the experiment shows that the suction force has a minimum at the windward side of every span roof near the ridge, contrary to what is expected. This disagrees with the numerical results which show the minimum of the suction force almost at the middle of the windward slopes of the roof. The origin of this behaviour is not yet understood.

Ventilation in greenhouses

CFD simulations have already been used for predicting greenhouse ventilation with success (Mistriotis *et al.*, 1996). However further investigation is necessary for vali-

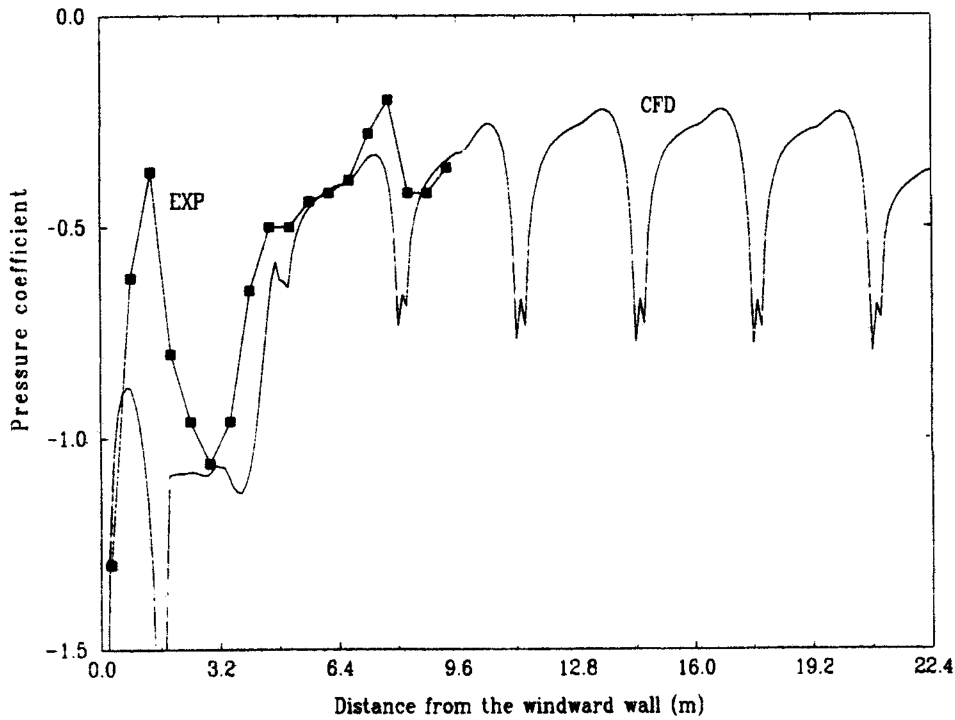


Figure 1. Pressure coefficients along the roof of a 7-span Venlo-type greenhouse. The continuous line corresponds to the numerical results and the filled squares (dashed line) represent the experimental data given by Wells & Hoxey (1980).

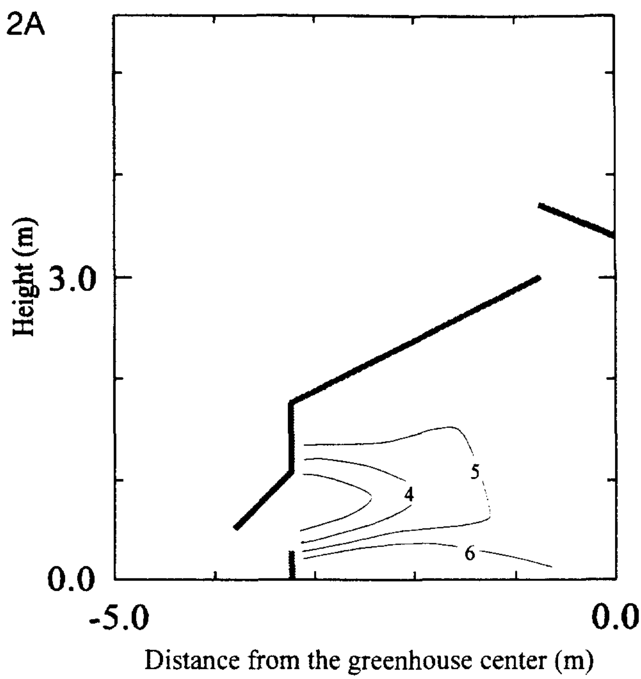


Figure 2A. Thermally driven ventilation in a single-span greenhouse with roof and side ventilators when windspeed is 0 m/s and the energy flux at the greenhouse ground is 500 W/m^2. The measured temperature contours showing the temperature difference to the outside air (°C).

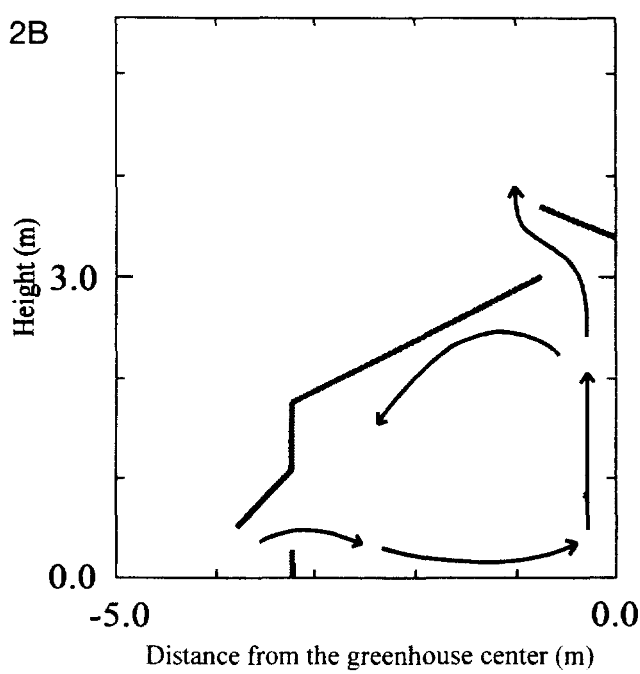


Figure 2B. Thermally driven ventilation in a single-span greenhouse with roof and side ventilators when windspeed is 0 m/s and the energy flux at the greenhouse ground is 500 W/m^2. Experimentally detected air flow pattern (Sase *et al.*, 1984).

2C

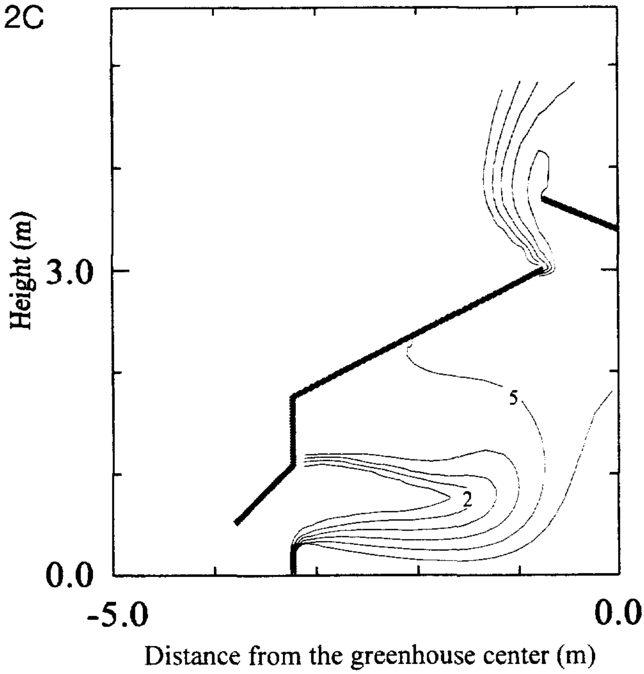


Figure 2C. Thermally driven ventilation in a single-span greenhouse with roof and side ventilators when windspeed is 0 m/s and the energy flux at the greenhouse ground is 500 W/m^2. Numerically calculated temperature contours showing the difference to the outside air (°C).

2D

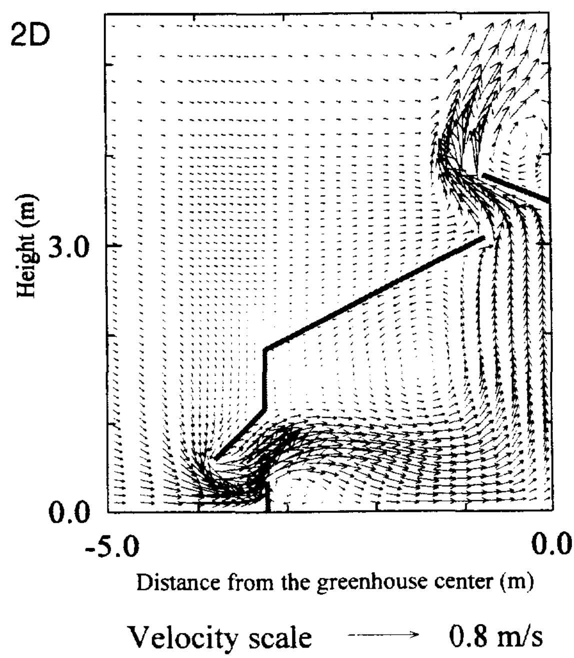


Figure 2D. Thermally driven ventilation in a single-span greenhouse with roof and side ventilators when the wind speed is 0 m/s and the energy flux at the greenhouse ground is 500 W/m^2. Numerically calculated air flow.

dating the method and clarifying its advantages and weaknesses. In particular, limited progress has been made in analysing situations where the external wind speed is low and the ventilation is mainly thermally induced. This is a consequence of the lack of sufficient experimental data which can be used for validating the numerical results.

One of the few studies of thermally driven ventilation in greenhouses is reported by Sase *et al.* (1984). They studied the air flow in a 1/10 scale model of a single-span greenhouse when no external wind exists. The height of the greenhouse at the ridge was 3.5 m, while its height at the gutter was 1.67 m. Its width was 6.5 m and its length was 18 m. This greenhouse was equipped with roof and side wall ventilators. The windows at the roof and the side walls had 0.7 m width. The side windows opened by pivoting around their higher side, which was located at 1.17 m height. They are opened at 45°, while the roof windows were opened at 60°. The floor was heated by an electric heating sheet. The temperature of the floor was kept 20 °C hotter than the outside ambient air.

In Figures 2a and 2b the temperature distribution and the internal air flow pattern are presented respectively when the wind speed is 0 m/s. The contours of Figure 2a showing the temperature difference between the inside and the outside air, were measured by thermocouples. The air flow shown in Figure 2b was detected by introducing smoke streams.

The above described greenhouse was studied numerically by a CFD simulation. The CK model was used. Since the greenhouse length is 2.8 times longer than its width and the ventilators are continuous, the air flow can be studied on a transverse (normal to the ridge) section by a 2-D CFD calculation. Moreover, the location and the opening of the ventilators are symmetric with respect to a vertical plane along the ridge. Therefore, the flow can be assumed symmetric with respect to this plane. Hence, it is sufficient to solve the transport equations only for half of the examined space. The shape of the cover is modelled with the help of a variable orthogonal grid. The half span width is spanned by 18 cells. The height from ground to gutter is spanned by 20 cells, while the height of the side ventilator is spanned by 10 cells. The width of the roof ventilator is spanned by 4 cells. The whole grid has 58×80 cells.

Similarly to the experimental set-up, the ground of the greenhouse is heated. The heat flux intensity H, corresponding to the temperature difference set during the experiment, can be estimated by using the formula for turbulent natural convection (Mills, 1992):

$$H = \alpha\, \Delta T^{4/3} \qquad (6)$$

where ΔT is the temperature difference between the ground and the external air and α is a coefficient determined experimentally. Values of α found in the literature vary between 1.7 and 10 W m^{-2} $K^{-4/3}$ depending on the material forming the ground (Stoffers, 1997; Silva, 1988; Mills, 1992). A typical value of α for plastic materials similar to the one covering the heating resistor used in the experiment described in (Sase *et al.*, 1984) is 3.0 W m^{-2} $K^{-4/3}$ (Stoffers, 1997). Considering that ΔT is 20 K and after applying the corresponding similarity condition, we estimate the convective sensible heat transfer from the ground to the air to be equal to 500 W/m^2 for the full scale case. An open boundary is assumed far from the greenhouse.

Figures 2c and 2d show the numerically calculated temperature distribution and the internal air flow respectively. A good qualitative and quantitative agreement is observed in Figure 2 between the numerical and experimental results. This result justifies the use of CFD in calculating thermally driven ventilation in agricultural building and validates the designing hints obtained in the example which follows.

CFD-study of natural ventilation systems in broiler houses
Natural ventilation in broiler houses instead of mechanical ventilation systems could lead to a significant reduction of energy for the farmer (Leijen *et al.*, 1993). In The Netherlands, however, farmers are reluctant to replace the traditional mechanical ventilation systems with natural ventilation. The main concern is the excessive heat stress of the animals during the summer, especially in situations with high solar radiation levels and almost no wind.

In this example, the question is studied whether in broiler houses natural ventilation can be applied to achieve acceptable climatic conditions inside the building, even under very unfavourable climatic conditions. The results are compared with the recommendations given to farmers (Anonymous, 1994a).

In the simulations the outside climatic conditions are chosen to be an air temperature of 28 °C, no wind and a solar radiation of 800W/m^2. Hence, the ventilation is solely due to the buoyancy effect. With the help of a static model of the livestock house (Van Ouwerkerk, 1996) the boundary conditions for the CFD-simulation (standard k-ε model) of the house under these specific outside conditions were calculated: a heat transmission of the roof of 17 W/m^2 and a heat loss through the floor of 10 W/m^2. The sensible heat production by the animals in the house was derived on the basis of the relation given by Strøm & Feenstra (1980) and was found to be 85 W/m^2 for about 20 animals of 1.75 kg each, per square meter.

In the simulations, three standard broiler houses (width = 15 m, gutter height = 2.30m, roof slope = 22°) were modelled in 2-D, each with a different ventilation scheme (Figure 3). The height of the side wall openings and the width of the ridge openings (both equal to 0.5 m) were chosen following the standard (Anonymous, 1994a). In the first broiler house (A), the side wall inlets were located under the gutter, while in the broiler house (B) they were moved to the floor level.

In the third broiler house (C), a solar chimney for enhanced stack ventilation is modelled. It is essentially divided into two parts: the solar air heater and the chimney. This type of ventilation opening is designed to enhance the effect of thermally induced ventilation in buildings. In practice it has the ability of self balance; the hotter the day is, the hotter the solar air heat collector becomes stimulating faster air movement (Bansal *et al.*, 1993). In the simulations the heat transfer from the absorber plate to the flowing air is 60 W/m^2. The other side of the air cavity is well insulated.

Figures 4a, 4b and 4c picture the results of the CFD simulations for the three configurations. The flow was calculated on a body-fitted variable grid. For the cases (A) and (B), the flow is solved on a 20×15 body-fitted grid. In the case (C), extra cells were introduced for describing the details of the solar chimney (Figure 4c). The region of the solar heater is spanned by a body-fitted grid with 25×4 cells. The chimney is described by a 8×14 cell grid. The results show that the successive ven-

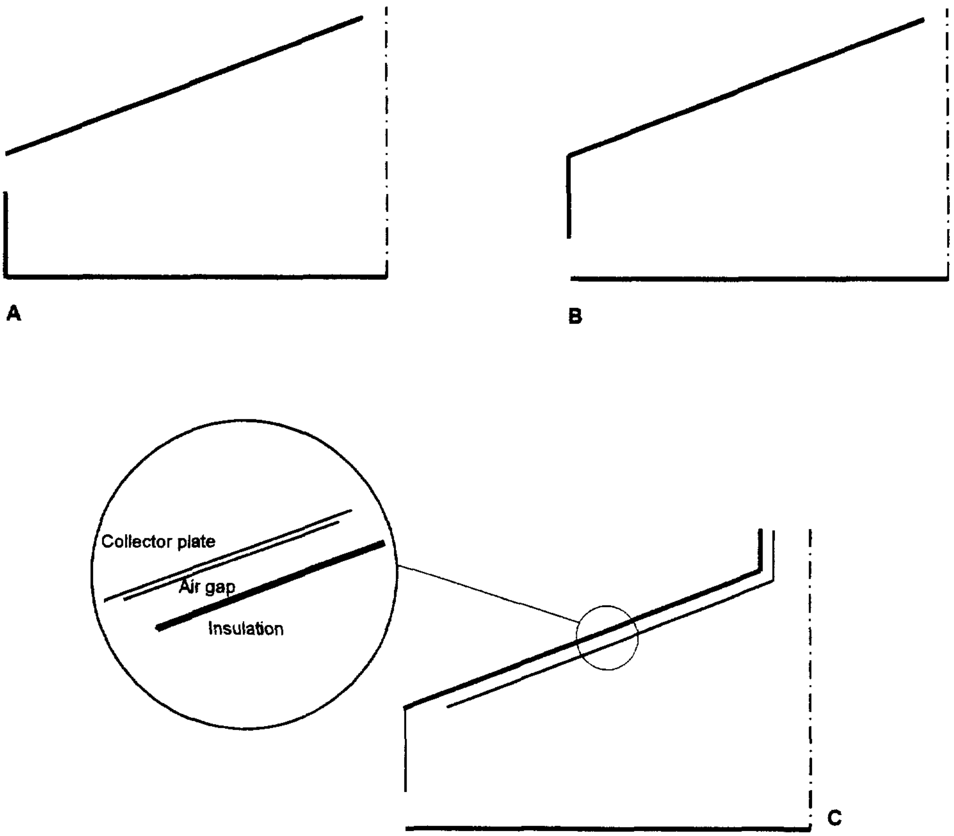


Figure 3. Schematic representation of the left section of the (symmetric) broiler houses. A: ridge vents and high side wall opening; B: ridge vents and low side wall opening; C: as B plus solar chimney (width of the air gap = 0.3m).

tilation schemes improve the calculated indoor climate at animal level, either due to an increased stack height or the solar induced ventilation. In the broiler house with side and ridge vents only (Figures 3A and 3B) the calculated ventilation rate per unit length of the house (whole building) is 1600 m^3/h and 1730 m^3/h respectively.

In the broiler house with the solar chimney the calculated ventilation rate is 1896 m^3/h per unit length of the house (whole building). High temperatures at the animal level are avoided and never exceed the outside temperature more than 3 °C. In this broiler house, both ventilation rates and resulting temperatures meet the requirements set in practice (Anonymous, 1994a). Also the calculated wind velocities at animal level are acceptable, except those very near the ventilation opening. Special attention to the aerodynamic design of the inlet could adjust the direction of the incoming air in such a way that too high velocities near the opening are avoided.

4A

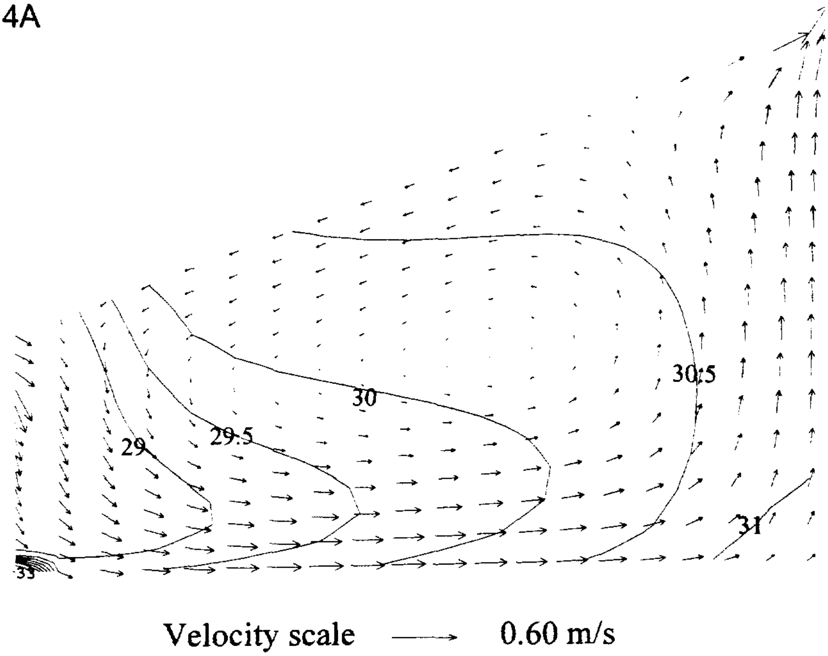


Figure 4A. Calculated temperature and velocity field of the left section of the house (broiler house A).

4B

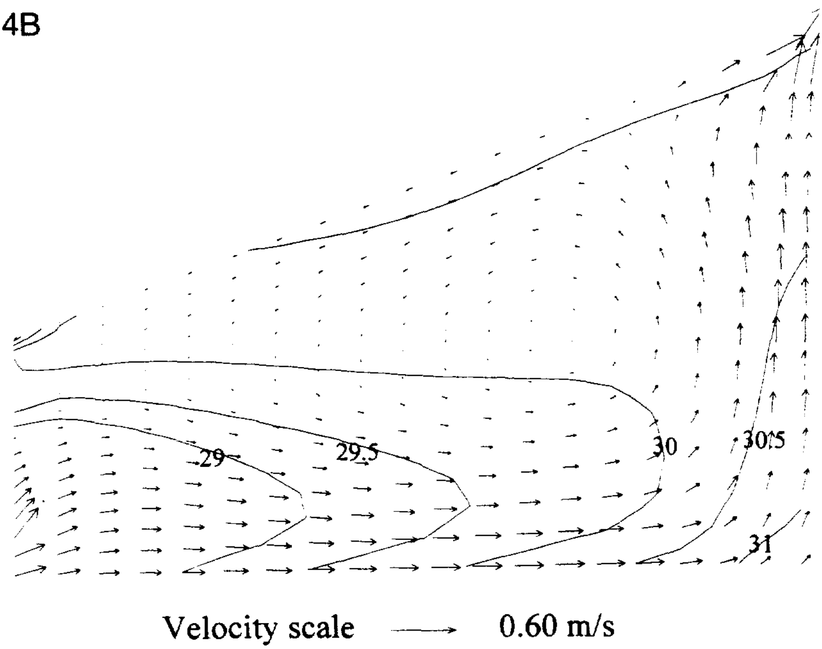
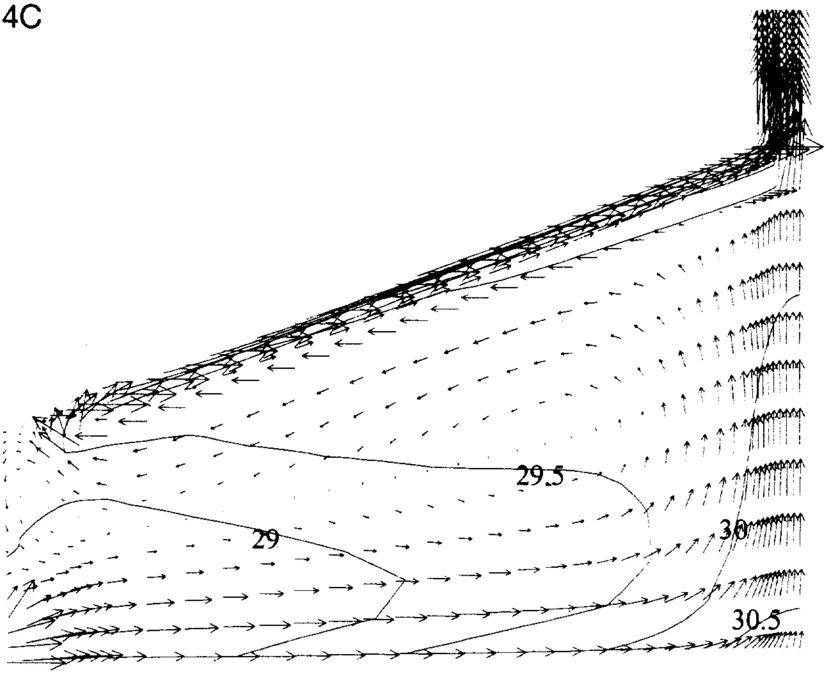


Figure 4B. Calculated temperature and velocity field of the left section of the house (broiler house B).

4C



Velocity scale ⟶ 0.60 m/s

Figure 4C. Calculated temperature and velocity field of the left section of the house (broiler house C).

Discussion

In this work some examples of the use of CFD in agricultural research are presented. The examples particularly focus on the analysis of the ventilation and the resulting indoor climate in agricultural buildings. Since CFD solves the complete momentum, energy and mass transport equations a large amount of information concerning the corresponding air flow can be obtained, something experiments could hardly provide. In the first example the pressure distribution along the roof of a greenhouse was calculated. In the other two examples the temperature distribution and the velocity field were determined.

For many types of flow, the accuracy of the numerical results depends on the correct modelling of the turbulent component of the flow (Mistriotis *et al.*, 1996). In the computations presented here, turbulence is taken into account by implementing the k-ε model or the Chen-Kim model.

In two cases the numerical model calculations were compared with experimental data. They show a reasonably good correlation. Therefore it can be concluded that in these cases CFD can be used to provide a good representation of reality. It should be stressed that model validation is an essential part of the research. The success of

CFD obviously depends on the right physical modelling of the problem, that is the definition of the right boundary conditions. Determining boundary conditions and sources is generally not an easy task. In many cases detailed experimental measurements are required for estimating them. Moreover the modelling of obstructions inside agricultural buildings is also a complicated task due to the usually coarse grid used in CFD calculations. Therefore internal structural elements, equipment, crops or animals have to be roughly described as blocking objects or be neglected when it is possible. Nevertheless, CFD offers greater flexibility in dealing with these complex problems than other methods such as wind tunnel experiments.

Further full scale measurements are required validating the CFD results for a representative system under realistic conditions. These experiments will contribute to further improvements and adjustments in CFD modelling. After validation, CFD can be used to calculate the efficiency of various designs of the system under study. It is expected that soon CFD simulations will develop into a reliable tool for investigating the ventilation efficiency of new agricultural structures.

References

Anonymous, 1994a. Poultry Handbook (in Dutch). Publication 42, Information and Knowledge Center, Ede, 267pp.

Anonymous, 1994b. PHOENICS Manual v.2.1, CHAM, London, UK.

Anonymous, 1995. Eurocode ENV 1991-2-4: Basis of design and actions on structures: Wind actions. NNI, Delft, 153 pp.

Awbi, H.B., 1991. Ventilation in Buildings. E & FN Spon, 313 pp

Bansal, N.K., Rajesh Mathur, & M.S. Bhandari, 1993. Solar chimney for enhanced stack ventilation. *Buildings and Environment*, 28:373–377.

Carpenter, G.A., 1986. Dust in livestock buildings – Review of some aspects. J. Agric. Eng. Res. 33: 227–241.

Chen, Y.S. & S.W. Kim, 1987. Computation of turbulent flows using an extended k-ε model. NASA CR-179204.

Cockshull, K.E., 1988. Integration of plant physiology and physical aspects of the greenhouse climate. *Acta Horticulturae* 229:113–123.

Harlow, F.H. & P.I. Nakayama, 1968. Transport of turbulence energy decay rate. LA-3854, Los Alamos Science Lab., Un. of California, USA

Launder, B.E. & D.B. Spalding, 1974. The numerical computation of turbulent flows. *Comp. Meth. in Appl. Mech. & Eng.* 3:269.

Leijen, C., J. Beukeboom, H. Jansen Venneboer, & J. Voet, 1993. Energy in intensive livestock (in Dutch). Publication 39, Information and Knowledge Center, Ede, 99 pp.

McAneny K.J., A. Baille, & G. Sappe, 1987. Turbulence measurements during Mistral winds with a 1-dimensional sonic anemometer. *Boundary Layer Meteorology* 42:153–166.

Mills A.F., 1992. Heat transfer. Irwin, Boston, 385pp.

Mistriotis, A., G.P.A. Bot, P. Picuno, & G. Scarascia-Mugnozza, 1996. Analysis of the efficiency of greenhouse ventilation using computational fluid dynamics. Accepted *J. of Agric. and Forest Meteorology*.

Patankar, S.V., 1980. Numerical heat transfer and fluid flow. Hemisphere, New York, 197 pp.

Richards, P.J. & R.P. Hoxey, 1993. Appropriate boundary conditions for computational wind engineering models using the k-ε turbulence model. *J. of Wind Eng. and Industrial Aerod.* 46&47:145–153.

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

Silva, A.M., 1988. A contribution to the study of the greenhouse climate: the analysis of some processes of energy transfer (in Portugese). Ph.D. dissertation, University of Evora, Portugal, 352 pp.

Stanghellini, C. & W.T.M. van Meurs, 1992. Environmental control of greenhouse crop transpiration. *J. Agric. Eng. Res.* 51: 297–311.
Stoffers, J.A., 1997. Personal communication. IMAG-DLO, Wageningen.
Strøm, J.S. & A. Feenstra, 1980. Heat loss from cattle, swine and poultry. ASAE paper 80–4021, St. Joseph, Michigan, 39 pp.
Van Ouwerkerk, E.N.J., 1996. User manual for computerprogramme STALKL – climate and energy balance of livestock building (in Dutch). IMAG-DLO Nota P96–11, IMAG-DLO, Wageningen, 62 pp.
Wells, D.A. & R.P. Hoxey, 1980. Measurements of wind loads on full scale glasshouses. *J. Wind. Eng. Ind. Aerod.* 6: 139–167.
Xin, H., T.L. Berry, & G.T. Tabler, 1996. Minimum ventilation requirement and associated energy cost for aerial ammonia control in broiler houses. *Transactions of the ASAE* 39: 645–648