Computational Modelling and Scale Model Validation of Airflow Patterns in Naturally Ventilated Barns

Merlijn De Paepe[1], Jan Pieters[2], Bart Merci[3], Wim Cornelis[4], Donald Gabriels[4] and Peter Demeyer[1]

[1] Technology and Food Science Unit - Agricultural Engineering, Institute for Agricultural and Fisheries Research (ILVO), Merelbeke, Belgium
[2] Department of Biosystems Engineering, Faculty of Bioscience Engineering, Ghent University, Gent , Belgium
[3] Department of Heat, Flow and Combustion Mechanics, Faculty of Engineering and Architecture, Ghent University, Gent, Belgium
[4] Department of Soil Management, International Centre for Eremology, Faculty of Bioscience Engineering, Ghent University, Gent, Belgium

Abstract

The application of natural ventilation in livestock farming still requires better understanding. Key issues are the configuration and dimensions of the ventilation openings throughout the barn. In this paper we tested the effect of six different ventilation opening configurations upon indoor air velocities, by means of Computational Fluid Dynamics (CFD). The 2D CFD results were verified with data from scale model experiments in a wind tunnel. Generally, both methods showed good agreement and indicated that larger ventilation openings resulted in lower air velocities near the inlet opening, but higher velocities at the outlet. Comparison with wind tunnel experiments also showed there is still room for improvement in the used CFD models. The obvious limitations of a 2D approach can be part of the explanation. Nevertheless, the main advantage of the computational approach was the ability to visualize the indoor airflows in each of the six geometries. Further research will involve 3D modelling.

Keywords: CFD, wind tunnel, animal housing, ventilation openings, air velocity

INTRODUCTION

Barns require an efficient ventilation system in order to maintain good indoor air quality, for both animals' and farmers' wellbeing. Ventilation capacities are also directly linked with emissions of particulate matter (PM) and gasses with an important environmental impact, i.e. ammonia, nitrous oxide and methane.

According to the EU, natural ventilation is the most preferable BAT (Best Available Technique) for animal housing (EPA, 2008). Using naturally available resources, like wind and buoyancy, this technique is energy efficient and also relatively low cost in comparison with mechanical ventilation. Still, it remains largely underused in livestock farming . This is partly due to some important challenges, related to complicated time-dependent effects, that must be overcome (Linden, 1999). In practice, these can lead to more complex ventilation management strategies, especially in treating acute changes in the indoor microclimate, e.g. high temperature and air contaminant levels.

The objective of our research is to gain more insight in the basics of the complex natural ventilation process in and around barns, with a focus on airflow patterns and velocities, using CFD and verification with wind tunnel experiments. In this paper we discuss 2D CFD simulations of airflows inside a barn, using six different ventilation opening configurations.

MATERIALS AND METHODS

Computational modelling

RANS modelling with Ansys Fluent 14.0 software was used to perform 2D simulations of natural ventilation with different scale models of a barn (Fig. 1) placed in a wind tunnel. The computational domain (Fig. 2) in fact consisted of the second part of the wind tunnel work section. The first –windward– part was simulated beforehand. The exiting wind velocity profile of the developed flow in the first part of the wind tunnel was used as the inlet velocity profile for the computational domain (second part of the wind tunnel).

The barn geometry is a 1:60 scale model of a typical Belgian barn (Fig. 1), featuring two side openings with variable height and a 0.5 cm wide ridge opening. The six scale model (SM) designs were: ('SM1') standard barn with equal inlet and outlet openings 1.8 cm in height; ('SM2') standard barn but with closed ridge opening; ('SM3') standard barn but with closed outlet opening; ('SM4') low-front barn, with a 2-cm high front wall and 5.6 cm inlet opening height; ('SM5') open-front barn, i.e. without front wall, thus a 7.6 cm high inlet opening; ('SM6') open housing type barn, i.e. without front or back wall. The SM2 and SM3 models are not really applicable for livestock farming, but were taken into account in order to test the power of the CFD model.

Each CFD case featured one of the six scale model designs. The meshes contained approx. 140 000 triangular cells. More details are given in Fig. 3. The boundaries and respective conditions are described hereafter (see 'Experimental validation'). A steady-state calculation using the pressure-based solver and standard k-ω turbulence model was chosen. Computations were carried out for isothermal conditions and with double precision. Convergence of the solutions was assumed at residuals between 10^{-3} and 10^{-6}.

Post-processing included the visualisation of air velocity contours and vectors. Inspection of these vectors enabled the distinct characterisation of flow paths in each of the six scale model designs.

Experimental validation

Norton et al. (2007) found CFD apt for use in agricultural situations, albeit stating that it should always be accompanied by thorough validation. Therefore, experimental air velocity measurements were performed using the scale models (Fig. 1) in the Ghent University I.C.E. wind tunnel, featuring a 12.00 m long, 1.20 m wide and 2.90 m high work section. The wind tunnel is further described in Gabriels et al. (1997) and Cornelis et al. (2004). The experiments were performed under isothermal conditions and a constant, developed airflow. The airflow is always fully turbulent in this setup, with a Reynolds number Re of approximately 419 000.

A transversal flow of 3.5 m s^{-1} impacted on each of the six scale model designs. The hydraulic diameter of the wind tunnel was determined at 2.0 m, while the turbulence intensity was 3%, according to the equation $I = 0.16 * Re^{-0.125}$.

Three calibrated hot-wire anemometers (type 8465, TSI Inc., Shoreview, MN, USA) were placed in the scale model. The measurement locations were at the centre of each inlet and outlet opening (see Fig. 3), as well as centrally indoors. This yielded 18 measuring points, three for each scale model design. All measurements took place simultaneously and at a height of 6 cm, in line with the standard ventilation openings. The measurement frequency was 1 Hz. Velocity values were averaged over 120 s.

RESULTS AND DISCUSSION

The 2D CFD simulations clearly showed that ventilation opening height and configuration affects the indoor air velocities. For instance, larger inlet openings led to lower air velocities near the inlet, but higher velocities at the outlet. Table 1 presents the CFD results, as well as the mean air velocity values for the wind tunnel experiments, plus and minus two standard deviations. Generally, the wind tunnel experiments show a good agreement with the CFD simulations. In six cases the CFD results showed significantly lower air velocities, in one case it was higher. Sørensen and Nielsen (2003) supposed that a two-dimensional CFD approach may be sufficient in the case of a full-width opening in a ventilated room, as is the case here. Still, the measured deviations in our research are probably due to the limitations of a 2D treatment of the flow. A 3D model can capture the flow more qualitatively, including possible lateral movements of flow vortices.

Fig. 4 shows the major flow paths which could be identified visually in the CFD results. Each ventilation configuration gave rise to noticeably different airflow patterns; e.g., closing the ridge opening (as in SM2), led to a ceiling-attached jet flow with a backflow at lower heights. A closed outlet (SM3) induced strong recirculation and forced all air through the ridge opening. Larger inlet openings (SM4-5) also resulted in ceiling-attached flows. Finally, the open-type barn SM6 posed little wind obstruction, hence mainly cross-ventilation occurred. From these findings it can be concluded that in real-life situations, indoor ventilation should always be carefully assessed in order not to jeopardize indoor air quality and the animals' and worker's health.

CONCLUSIONS

The 2D CFD models showed that larger ventilation openings gave rise to lower air velocities near the inlet opening, but higher velocities at the outlet. Comparison with wind tunnel experiments gave reasonable results, but there is still room for improvement in the computational models. Nevertheless, the main advantage of the computational approach was the airflow visualisation for each of the six ventilation opening configurations.

In future research 3D modelling should be performed. This approach also allows to study the effect of different wind incidence angles on the barn's indoor air velocities. Larger scale (real life) experimental setups are also a point of future interest.

ACKNOWLEDGEMENTS

Thanks to our colleague ing. Philippe Van Overbeke for the CAD drawing.

Literature Cited

Cornelis, W.M., Erpul, G. and Gabriels, D. 2004. The I.C.E. wind tunnel for water and wind interaction research. In: S. Visser, W.M. Cornelis (Eds.). Wind and rain interaction in erosion. ESW publications, Wageningen, Netherlands. p. 195-224.

EPA (Environmental Protection Agency, Ireland). 2008. Draft BAT Guidance Note on Best Available Techniques for the Intensive Agriculture Sector.

Gabriels, D., Cornelis, W., Pollet, I., Van Coillie and T., Ouessar, M. 1997. The I.C.E. wind tunnel for wind and water erosion studies. Soil Technology, 10, 1-8.

Linden, P.F. 1999. The fluid mechanics of natural ventilation. Annu. Rev. Fluid Mech., 31, 201-238.

Norton, T., Sun, D.W., Grant, J., Fallon, R. and Dodd, V. 2007. Applications of computational fluid dynamics (CFD) in the modelling and design of ventilation systems in the agricultural industry: a review. Bioresource Technology, 98, 2386-2414.
Sørensen, D.N. and Nielsen, P.V. 2003. Quality control of computational fluid dynamics in indoor environments. Indoor Air, 13, 2-17.

Tables

Table 1. Air velocity magnitudes (in m s^{-1}) obtained through the 2D CFD models as well as experimental values.

Scale model	Measurement location	Computational air velocity (m s^{-1})	Experimental air velocity (m s^{-1}), avg. ± 2 st.dev.
SM1	inlet	3.4	3.7 ± 0.4
	indoors	0.5	0.7 ± 0.2
	outlet	1.9	1.5 ± 0.4
SM2	inlet	3.2	3.0 ± 0.4
	indoors	0.2	0.6 ± 0.2
	outlet	2.1	1.5 ± 0.4
SM3	inlet	1.3	1.3 ± 0.4
	indoors	0.1	0.4 ± 0.2
	outlet	0.1	0.2 ± 0.2
SM4	inlet	1.5	2.0 ± 0.6
	indoors	0.2	0.6 ± 0.2
	outlet	2.6	2.5 ± 0.4
SM5	inlet	1.7	2.3 ± 0.8
	indoors	0.9	0.7 ± 0.2
	outlet	2.1	2.7 ± 0.4
SM6	inlet	2.7	3.5 ± 0.4
	indoors	1.8	2.5 ± 0.2
	outlet	3.3	3.2 ± 0.4

Figures

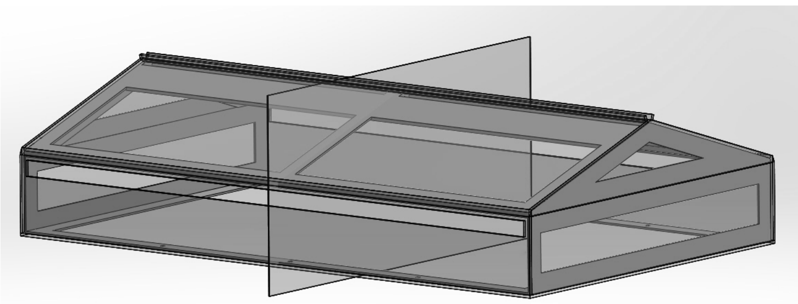

Fig. 1. CAD design of the scale model barn (type 'SM1') used in the wind tunnel experiments, consisting of an aluminium frame (dark) and polycarbonate sheets as walls (transparent). The central section plane shows the two-dimensional geometry used in the CFD model.

Fig. 2. Computational domain, i.e. the second half of a wind tunnel's test section (6 m x 2.9 m), with a scale model positioned at 1.67 m from the velocity inlet boundary.

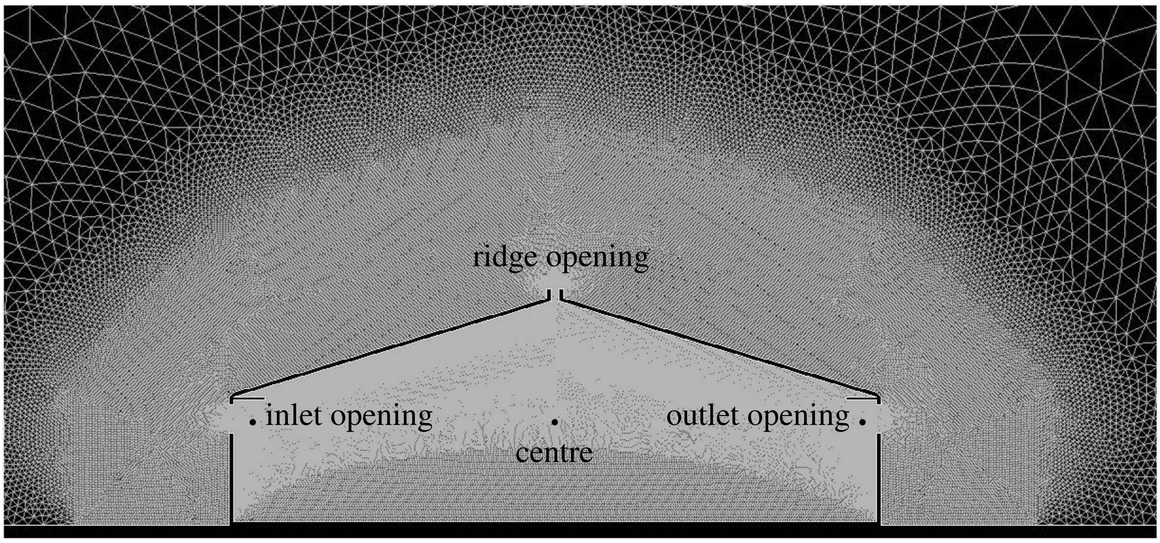

Fig. 3. Detail of the 2D computational mesh in and around a model barn (design 'SM1'). A virtual 'dome' surrounded the barn, in which the cell dimension was 0.1 cm. Outside the dome, the cell dimensions gradually rose to 1.0 cm. The three black dots (indoors) indicate the positions of air velocity readings.

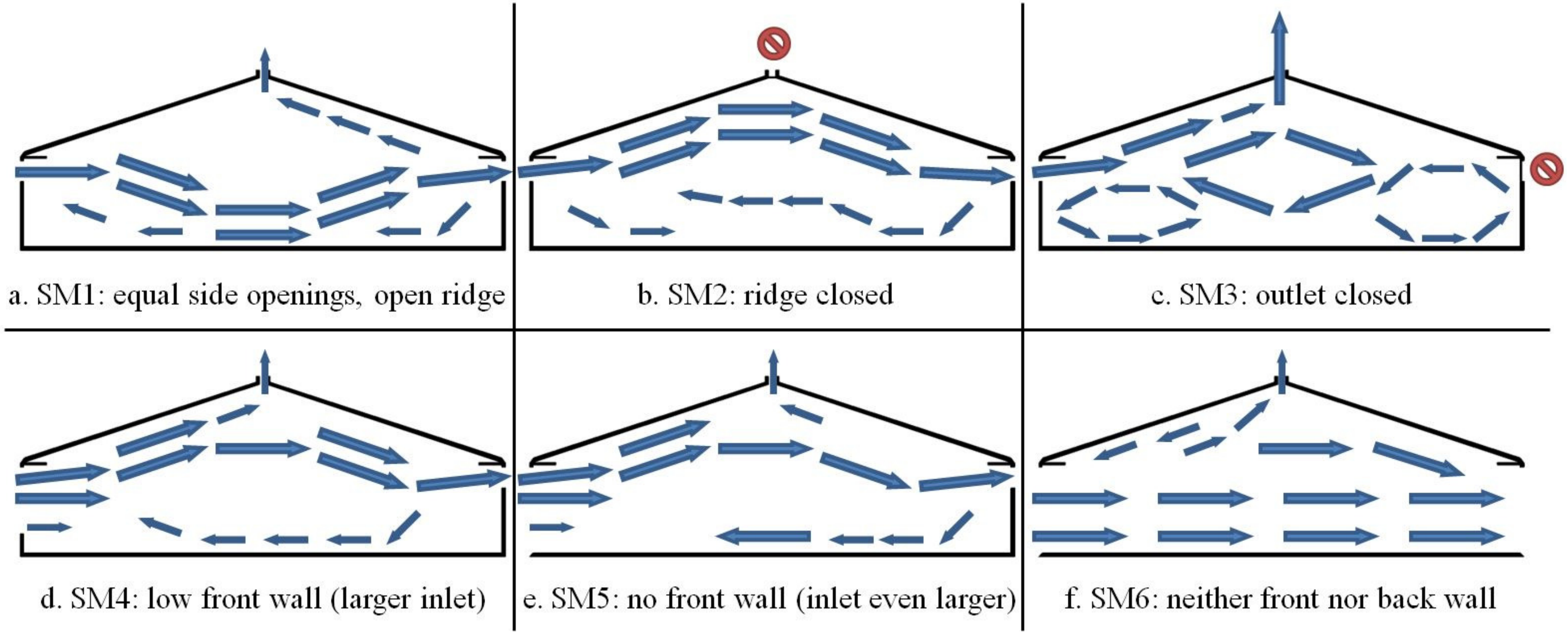

Fig. 4. Main airflow paths as visually observed in the two-dimensional CFD models of the six scaled cattle barn designs. The larger vectors indicate the paths with (relatively) higher velocities, while the small vectors show secondary, lower-speed motions.