Energy and Buildings 86 (2015) 25–34

Contents lists available at ScienceDirect

Energy and Buildings

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

The effect of wind speed and direction and surrounding maize on hybrid ventilation in a dairy cow building in Denmark

L. Rong[a,*], D. Liu[a], E.F. Pedersen[b], G. Zhang[a]

[a] *Department of Engineering, Aarhus University, Inge Lehmanns Gade 10, 8000 Aarhus, Denmark*
[b] *AgriFarm A/S, Agro Business Park, Niels Pedersens Allé 2, 8830 Tjele, Denmark*

ARTICLE INFO

Article history:
Received 19 March 2014
Received in revised form 2 October 2014
Accepted 7 October 2014
Available online 16 October 2014

Keywords:
ACH
CFD
Hybrid ventilation
Wind speed and direction
Surrounding plant canopy

ABSTRACT

This study evaluated the effect of wind speed and direction and surrounding maize field on the air exchange rate (ACH) and indoor air velocity in a dairy cow building with hybrid ventilation, which combined auto-controlled natural and partial mechanical pit ventilation. The standard $k-\varepsilon$ turbulence model and standard wall function were applied in CFD modeling with extension of capability to account for the aerodynamics effect of surrounding maize plant canopy in the wind domain by using user defined functions (UDF). This extended model was validated by on-site measured velocities and temperatures. A reasonably good agreement was found between simulated and measured results. The wind speed influenced ACH greatly while modeling the maize field had little effect on ACH with low wind speed. With wind speed of 3.86 m s^{-1} in validation case, modeling the maize field reduced total ACH by 24%, ACH via bottom openings on the sidewall by 89.7% and air speed measured upwind by 71%. The results revealed that the plant canopy had the most significant effect on ACH through the opening on the sidewall. With the variation of wind direction from 0° to 90°, the difference of ACH could be 60%.

1. Introduction

In temperate climate countries such as Denmark, dairy cow buildings are usually ventilated by natural ventilation. In naturally ventilated buildings (NVB), it is difficult to clean the exhaust air, which leads to the ammonia and other contaminant gases exhaust to the atmosphere directly. In addition, it is a challenge to maintain appropriate thermal conditions in NVB under cold weather conditions owing to the difficulty in controlling the momentum of ventilation air. To overcome the drawback of NVB, a hybrid ventilation to combine natural and partial pit mechanical ventilation has been applied in a dairy cow building in Denmark. The partial pit ventilation is to collect highly concentrated contaminants which can be removed by a filter installed at the exhaust. In addition, the partial pit mechanical ventilation will play the main role under cold weather.

Since natural ventilation (NV) is an important part of the system, design of hybrid ventilation is a challenge due to the change of wind speed and wind direction over time as well as significant impacts of the geometry of the building itself, the geometry of surrounding buildings and the surrounding topography on air exchange rate (ACH). Apart from wind speed [1–3], wind direction and surrounding topography are two significant factors influencing the ACH of NV. Horan and Finn [1] examined ACH of a free-standing two story NVB for four wind directions using Computational Fluid Dynamics (CFD) and found large variations in ACH (from 3.5 to 15 h^{-1}) caused by the wind directions. Teitel et al. [4] studied the air flow in one of the four clustered greenhouses for four wind directions with an interval 30° and found significant differences in indoor airflow patterns and ACH (up to 56%). Norton et al. [5] investigated the ventilation effectiveness in a calf building and found differences in ACH up to 100% depending on the wind direction. Paepe et al. [6] performed measurements in a wind tunnel to study the effect of wind incidence angle and found the difference of ACH depending on the wind incidence angle could be up to 60%. Saha et al. [2] analyzed experimental data to study the effect of wind speed and direction on ACH and emission. They found that the wind direction had an important effect on ACH ($P < 0.05$). There are also studies which found that the wind direction hardly had effects on ACH [7,8] in NVB. However, none of these studies through either measurements or CFD simulations had considered the effect of surrounding topography, e.g. plant canopy, on ACH.

Although NV has been applied for centuries, it is still an intensive area of research owing to both its complexity and potential for sustainable building design and operation [9]. A large number of researchers have been conducting studies on NV, based on

---

[*] Corresponding author. Tel.: +45 50248562.
E-mail addresses: Li.Rong@eng.au.dk, rongli1980@gmail.com (L. Rong).

http://dx.doi.org/10.1016/j.enbuild.2014.10.016
0378-7788/

theory, analytical methods, experiments and numerical simulation [1,3,5,6,10–19]. Usually, theoretical and analytical approaches are very valuable to provide general insights but less suitable when complex geometrical configurations are involved. With full-scale (on-site) experimental measurements, usually only limited sampling positions are feasible due to the complexity and cost. It is also extremely difficult to design the on-site experiments under well-controlled conditions. Wind tunnel experiments, consequently, have been widely used in study of NV, in which the influencing factors such as wind profiles, wind speed and direction can be well controlled. But test of scaled models in the wind tunnel can be hampered by similarity requirements and the results are hard to be used in full-scale buildings. The main advantages of CFD are that it allows full control over influencing factors and it provides detailed information on all relevant parameters distribution simultaneously at all points of the computational domain of full scale geometry. CFD modeling has become one of the most important tools to study ventilation performance [12] including natural ventilation [20]. However, the accuracy and reliability are the main concerns and therefore CFD verification and validation studies are imperative [21].

This paper presents a study on the effect of wind speed and direction and surrounding plant on the hybrid ventilation in the dairy cow building. The objectives are to validate the CFD modeling with capability to simulate the aerodynamic effects of vegetation canopy on wind flow by on-site experiments and to investigate the effect of wind speed and direction and surrounding plant canopy on ACH.

2. Method and materials

2.1. The experimental building

The measurements were performed in a dairy cow building with hybrid ventilation, located in Skjern, Jutland, Denmark (altitude: 55°59′36.6″, Longitude: 8°29′53.52″). The sketch of the building and the locations of the measurement points were illustrated in Fig. 1. The *X* direction was pointing to West and *Z* direction is pointing to North. There were four rows of adjustable openings at each side of the building, two rows located on the sidewall, one row located on the roof and one row on the ridge. All the openings could be adjusted by an auto-control system. There were two big gates on the front wall for supplying feed to the feeding alley. The gates were typically kept closed and were only opened when it was necessary. The walking alley area between cows' cubicles and feeding alley was slatted floor. The feeding alley and cubicles were slightly higher than the slatted floor. Below the slatted floor, the manure was scraped to the deeper slurry channel near the back wall, above which there was another walking alley in slatted floor connected all other walking alleys to the milking building on the East. The milking building was not plotted in Fig. 1. Between the cattle building and milking building, there was also a gate which was only opened during milking time. Outside of the building on the West, there was maize field (112 m × 2.2 m × 300 m), which was 1.0 m away from the sidewall.

In the partial mechanical pit ventilation, there were eight exhaust channels with four named as EA and four named as WA below the cows' cubicles to transport the exhaust air to the central air channel as shown in Fig. 1. There were also four air supply channels to add fresh air to the slurry channel, named as SA. The pit exhaust airflow rate was approximately 25% of the designed maximum ventilation rate (450 $\mathrm{m}^3\ \mathrm{cow}^{-1}\ \mathrm{h}^{-1}$) and supplying airflow rate was 50% of the exhaust airflow rate.

Air velocities were measured by three-Axis ultrasonic anemometers (Wind-Master, Gill instruments Ltd.) at seven positions in the cattle building space, marked as A–F and H in green, Fig. 1. Four were near the side wall openings (B, E on the East; C, D on the West) in the height of 3.07 m. Two were placed near the roof openings (H on the East and F on the West) in a height of 7.6 m and one, A, was located in the center of the building near the ridge opening in a height of 9.9 m. The same type anemometer was used on the West, 10 m high and 30 m away from the side wall to measure the outdoor wind speed and direction. All eight ultrasonic anemometers were connected to a multi-point adapter supplying electricity power to the anemometers and collecting three-Axis velocity data. The data sampling frequency was 20 Hz. The indoor and outdoor air temperature was measured by Type T thermocouples with an accuracy of ±1.0 °C. One was placed outdoor and 27 were installed inside the cattle building, marked with small red squares in Fig. 1. The three lines to measure indoor temperature were located at $Z = -12.5$ m, 2.5 m and 17.5 m, respectively. Four temperature sensors were also placed on the sidewall and roof in order to obtain information for setting boundary conditions on the building walls in CFD simulations.

2.2. The CFD model

The fundamental problem of CFD simulations lies in the prediction of the effects of turbulence. In this section, the theory to model the flow in surrounding maize canopy was briefly introduced as well as the porous media model of slatted floor and animal occupied zone (AOZ). In addition to the physical difficulties of modeling the turbulence, there were many other sources influencing the accuracy and quality of CFD simulations [22–25] such as numerical discretization, grid, boundary conditions, which were also described.

2.2.1. CFD representation of the building geometry

The dimensions of the building are: length (*Z* direction) 45.0 m, width (*X* direction) 74.0 m, eave height (*Y* direction) 3.41 m, roof height 8.61 m and ridge height 11.3 m. The origin of the coordinate was in the center of the floor, see Fig. 1. The corridor to the milk building and the milk building itself (including offices) was also included in the CFD geometry. The domain size of geometry model for CFD simulation was 550 m (*X* direction from −300 to 250), 151.34 m (*Y* direction from −1.34 to 150) and 500.0 m (*Z* direction from −300 to 200). The distances from the building to the sides, to the inlet and to the top of the domain are at least five times of the height of the building and the distance from the building to the outlet is fifteen times of the height, as recommended by Tominaga et al. [26]. The opening ratio of the openings (defined as the area at a certain opening position divided the area at the full open position) were 92% and 92.2% at bottom windows, 90.5% and 89.8% at the top windows on the sidewall, 91.0% and 92.6% at the roof, and 93.0% and 28.5% at the ridge openings on the East and West, respectively. These data were obtained from the data log of a control system (SmartFarm). The 360 cows housed in the building were divided into three AOZ domains, as shown in Fig. 2. The AOZ was modeled as porous media. As mentioned previously, there was a maize canopy on the West (positive × direction) of the building, which was 1.0 m away from the sidewall. The width of the maize field was 112 m, the height was 2.2 m and the length was 300 m (*Z* direction from −200 to 100). The maize canopy was modeled by including additional source terms to the momentum, turbulent kinetic energy and turbulent kinetic energy dissipation equations via UDF in Fluent. This will be described in the following section.

2.2.2. Theoretical considerations of CFD modeling

The standard $k-\varepsilon$ turbulence model has been used in CFD simulations of livestock buildings [3,5,18,27] and was adopted in Ansys Fluent 14.0 (commercial CFD software) to describe turbulent