ABSTRACT Results of a study are presented to understand the 3-dimensional airflow pattern created by a high volume low speed (HVLS) stirring fan with multiple blades operating within a free stall barn. The flow fields are simulated using a finite volume based computational fluid dynamics (CFD) package to solve the discretized Navier-Stokes equations coupled with an appropriate turbulence model. Boundary conditions are selected to match the practical application. The simulation results are presented with velocity contours, vector maps, streamline trajectories, and isovelocity surfaces. The simulated data produced in this study are expected to be useful for further enhancement of existing fan design, for use with hybrid particle tracking velocimetry (HPTV) analyses, and could it be helpful in better understanding ventilation rates from naturally ventilated structures.

Keywords: Ventilation, Barn, Computational fluid dynamics, Thermal comfort.

INTRODUCTION Maintaining proper indoor air quality (IAQ) in livestock buildings is vital to ensure optimal animal productivity and occupant safety. Previous research has found that the structural design and ventilation configuration of livestock buildings can create non-uniform IAQ conditions in the animal occupied zone (e.g., Norton et al., 2010). The objective of ventilation in agricultural buildings is to provide a spatially uniform air temperature, humidity, and velocity distribution at levels that maintain high animal productivity while minimizing energy consumption. At the same time, the concentration levels of particulate and gaseous contaminants should be controlled. Previous research has shown evidence of direct link between poor air quality in the microenvironment around an animal and animal respiratory diseases (Lago et al., 2006).

Two different ventilation methods often used to maintain acceptable IAQ in livestock buildings include natural ventilation and mechanical ventilation. Natural ventilation, as the name implies, occurs naturally when an air flow develops in a space due to pressure differentials across the structure. Buildings experience natural ventilation due to gradients...
in wind speed and temperature across the exterior surfaces. The air exchange rate relies heavily on the natural porosity of the buildings including windows, doors, stacks etc and the outdoor climate. The green-building movement has inspired a renewed interest in natural ventilation in recent years because of its huge potential in energy savings. However, the main drawback of natural ventilation systems is the lack of user control and the difficulty of sustaining high air exchange rates and uniform air temperature and velocity distributions in the occupied zones for larger buildings. Mechanical ventilation overcomes these limitations through the use of powered fans to force air through the building. Mechanical systems are capable of consistently providing the required air exchange rate and can quickly respond to changing animal heat and pollutant loads. Panel fans are frequently used for ventilating livestock buildings. The air exchange rate in livestock buildings can vary over several orders of magnitude between winter and summer (Moulsley & Randall, 1990; Gates et al., 2004). However, the main drawbacks for mechanical ventilation are higher initial costs and higher operating costs for electricity, which can be significant in relation to profit margins.

In an attempt to bridge the gap between the high cost of mechanical ventilation systems and low controllability of natural ventilation, fan manufacturers have begun to introduce low power mixing fans for naturally ventilated livestock buildings. Large diameter high volume low speed (HVLS) stirring fans have the potential to enhance the mixing effect within natural ventilated buildings while adding minimal energy cost. These fans are intended to maintain uniform air temperature and velocity within the occupied zones and provide a comfortable environment for animals. However, little research has been conducted to evaluate the performance of these systems. There are currently no established performance criteria for HVLS fans which are comparable to the criteria used for panel fan systems. For example, panel fans are classified through measurements of flow rate as a function of applied pressure load and rotation velocity (Maghirang et al., 1998; Moulsley & Randall, 1990). Without this type of quantitative measurement standard, it is difficult to provide clear fan selection and installation guidelines to users.

Due to the low cost of numerical simulation and ability to handle complex geometry, more and more environmental phenomena are being studied using computational fluid dynamics (CFD) in recent years (Norton et al., 2007). In fact, the earliest application of CFD on indoor airflow prediction can be traced back to 1978 (Nielsen et al., 1978). From the point view of the required computational resources, the current available CFD models could be classified into (1) Direct numerical simulation (DNS), (2) Large eddy simulation (LES), and (3) Reynolds averaged Navier-Stokes models (RANS) (Stamou & Katsiris, 2006). In DNS models, the target volume has to be divided into a very fine mesh comparable with Kolmogorov scale, which is usually unaffordable for full-scale indoor air quality applications. LES is a kind of simplified version of DNS. It assumes that the small scales are independent of flow geometry, therefore only large scales are solved while the remaining smaller eddies are simulated using sub-grid models. With the rapid development of parallel computing, there are some successful applications of LES on indoor air quality within some simple geometry chambers (Choi & Edwards, 2008; Ezzouhri et al., 2009; Zhang & Chen, 2000; Zhang et al., 2007). Although some impressive results have been obtained, LES is still a very expensive approach for general simulation purpose. RANS could be regarded as a special case of LES. It only solves statistically averaged variables of the transport equations and simulates the turbulence fluctuation effect on the mean airflow. Compared with DNS and LES, RANS can
achieve reasonable accuracy with much less computing cost. The performance of five different k-ε models has been compared in previous studies (Gebre medhin & Wu, 2003; Chen, 1995). According to their reports, the RNG k-ε model, which applies a rigorous renormalization group method, produced simulation results that were more agreeable with experimental data than other models and appears to be the most practical, stable and accurate model for room airflow simulations.

The work to be addressed in the current paper is the initial phase of a project toward understanding the ventilation performance of HVLS systems. Given the energy savings potential of HVLS fans, there is a real need to develop standardized performance criteria and testing procedures, and subsequently to develop guidelines for their implementation in facilities. This work is aimed to serve as a basis for the development of these standards. In this work a numerical study was conducted to identify the performance variation between two different HVLS installations in a typical dairy free-stall barn. Through CFD simulation we are able to gain key insights into the impact of single versus multiple HVLS fans in a given space. The RNG k-ε model was applied to predict the three dimensional steady state airflow pattern created by the HVLS fans. The aim is to draw attention to this important issue, to provide some preliminary data for future experiment design and evaluation criteria development, and to guide the development of a sampling strategy for CFD model validation in full-scale facilities.

**COMPUTATIONAL DOMAIN AND MESH GENERATION** The simulated building is a typical dairy free-stall barn without side walls as shown in Figure 1. To reduce the computational load, only one 3D section located in the center of the barn was numerically modeled. Since the length of the building is much greater than the width, we assume that the airflow through the central section is periodic and not impacted by ends of the building. The eave height is 3 m. The installation level of the HVLS fans is 3.5 m. The slope angle of the roof is close to 30 degrees. A 1 m wide roof ridge opening serves as an inlet/outlet open to enhance natural ventilation. Figure 1(b) shows the geometry of HVLS fans used in the simulation. The thickness of the blade has been neglected, but the curve of the blade is well captured. Nominal fan diameter is 6 m.

![Figure 1. The simulated free-stall barn and HVLS fan](image)

Two fan installation layouts were simulated. In the first case, a single HVLS fan was placed at the center of the barn, while the second case consisted of two fans installed on adjacent sides of the barn offset from the centerline. Figure 2 (a) and (b) show the
computational domain selected for case 1 and case 2 respectively. To maintain the periodic boundary conditions, the domain for the second case is twice as long in the length (x) direction compared to the first case.

![Figure 2. Layouts of the simulation domains: (a) Perspective for Case 1; (b) Elevation of Case 1; (c) Perspective for Case 2; (d) Plan view of Case 2.](image)

For this study the computational domain was discretized using an unstructured tetrahedral finite volume mesh. Generating a computational mesh with sufficient resolution is critical to obtain accuracy in CFD simulation. If the mesh is too coarse, then small scale velocity gradients may be poorly represented, resulting in an inaccurate simulation result. At the other extreme, an overly fine mesh will waste a lot of computational resources, and therefore should also be avoided. To determine the appropriate mesh size for an accurate simulation, several tests must be conducted using successively finer meshes to analyze the sensitivity of the solution to grid size. The final mesh size selected should be the coarsest mesh that fully resolves the desired flow phenomenon. The alternative meshing approach is to divide the volume into separate computational zones whose mesh size is assigned according to the complexity of the features within the zone itself. For current geometry setup, a small pie-shaped zone surrounding the blades is separated from the remaining volume. Fine mesh is created within this cylinder to capture the blade geometry and the high speed gradient airflow field, while coarser mesh is established for the remaining volume. Figure 3 shows the mesh setup related to the blade region. Overall, there are 607,081 tetrahedral cells created in the first case, and 1,281,400 cells created in the second one.
MATHEMATICAL MODEL AND BOUNDARY CONDITIONS As mentioned in section one, the RNG k-ε turbulence model was chosen to close the governing equations as proposed by Yakhot et al. (1992). It accounts for the contribution of small scales to turbulent diffusion by modifying the transport coefficient, as compared with what has been done in standard k-ε model: only the contribution from the specified scale is considered. By combining with appropriate treatment of the near-wall region it is suitable for low-Reynolds number flow, for example indoor airflow (Stamou & Katsiris, 2006). A detailed mathematical derivation of this model can be found in the references (Stamou & Katsiris, 2006; Yakhot et al., 1992), and will not be repeated here for brevity. There are many commercialized CFD codes that include a RNG k-ε model, such as CFX, PHOENICS, and Fluent. In the present work, the latest version of Fluent (v.6.3) was used as the numerical solver.

To obtain predictive accuracy and a stable solution, it is important to utilize a reliable simulation strategy. At the initial phase, a laminar flow model without considering the turbulent characteristics is applied to all cells. The very stable first order upwind discretization scheme was used. The resulting data serve as the initial values for the turbulent model simulation. Then, the turbulent kinetic energy and dissipation equations are coupled, and the second order upwind discretization scheme is used for these equations. Pressure and velocity coupling is handled through the use of the semi implicit pressure linked equations (SIMPLE) method. The result is considered to be converged if the scalar residues in the equations are smaller than $10^{-5}$.

Figure 4. Main boundary components in this CFD simulation
The main components in this simulation are depicted in Figure 4. Proper boundary conditions (BCs) are needed to represent the physical situation. Since Case 2 is very similar to Case 1 except that one additional fan is installed, BCs for Case 1 are described here. The velocity field within the building is driven by HVLS fans. It is impossible to know the flow direction and velocity magnitude at the outlets before the simulation. Therefore, we assume that the gauge static pressure at ceiling and side outlets are zero (atmospheric pressure). Each section of the building is connected through periodic boundary conditions, shown as periodic planes in Figure 4. As mentioned in Section 2, the whole volume is divided into two zones meshed by cells with different size. These zones are connected via an interface plane, which is defined as interior in Fluent. It will not affect the airflow field in the simulation. The fan is rotating at 6 rad/s. The shear condition on the blade surfaces is set as no slip, and their roughness constant is 0.5. The roof is simply defined as a stationary wall without slip.

RESULT AND DISCUSSION

Airflow fields In Figure 5, the airflow fields in the two cases calculated with RNG k-ε model are shown in the horizontal XY plane at different heights above the ground (Z = 1 m, 2 m, 3 m, and 3.47 m). In these figures, both air velocity magnitude contours and velocity vectors are shown. Although the maximum air velocity achieves approximately 20 m/s at the outer edge of the blade, the air velocity range at most other areas in the barn is within 0 to 1 m/s. Therefore, in the contour map, the velocity legend is set from 0 m/s to 1 m/s. To better visualize the airflow directions, a uniform length has been applied to all velocity vectors. The magnitude of air velocity is indicated with specified colors.

The most interesting observation of these results is the airflow pattern that the HVLS fan creates. Intuition may lead to the prediction that the fan will push the airflow downward in a uniform column below the fan. However, it turns out that HVLS fan generates a vortex airflow pattern, which rises towards the ceiling in the center. While counter intuitive, this may be explained by the sheer size of the fan. Due to their large diameter (6-m), the velocity of the blades near the hub is much smaller than the velocity at the tips. This velocity gradient creates a higher pressure near the tips and lower pressure in the center. The air is therefore sucked towards this central region from both above and below and then forced outward by the rotating blades. As a response to that, a tornado shape flow pattern is created below the blades, which can be seen through visualization of the streamlines, and has been noted with smoke tests. The airflow is moving upward in a conical section beneath the fan. It is also observed that the fan creates a fairly uniform velocity profile at the breathing zone level, from Z=1 m to Z=2 m, as shown in Figure 5 (c), (d), (g) and (h). In both cases, the velocity range is mostly less than 0.2 m/s.

In Figure 5 (a)-(d), the airflow pattern is dominated by one large vortex. However, in Figure 5 (e)-(h), more complicated flow structures are observed due to the interaction effect between two fans. Besides the two main vortices created by blades, there are some small scale flow structures. A relatively large vortex is created centered at about (X=10 m, Y=7 m as depicted in Figure 5 (f)). Because both fans are rotating clockwise as viewed from above, the air pushed by each tends to build a counter-clockwise vortex. At its opposite location centered around (X=10 m, Y= -7 m), a similar flow patterns is also produced. Since the airflows driven by two fans are moving in opposite directions at these regions, some very complicated flow details are created due to their conflict.
Figure 5. Simulated airflow fields. Note Z=3.47 m is directly beneath the fan blades.
Airflow streamlines Figure 6 shows the streamlines derived from the velocity field. The small spheres represent snapshots of virtual air particles at the time step for streamline tracking. A larger space between spheres means a higher local velocity. This velocity magnitude is also represented by different sphere colors. Figure 6 is created by releasing virtual spheres uniformly on the floor. At each time step, their velocity is determined by the local air velocity. The positions of the spheres at the next instance are predicted by multiplying their velocity and the time step. This process is repeated until the desired time length is obtained.

Figure 6. Airflow streamlines for the two simulated cases.

From the streamline patterns, the large vortices mentioned previously are clearly observed, with diameters a bit larger than the fan diameters. By tracking streamlines, the flow direction can also be determined. Air from the lower level within the building is sucked by the negative pressure at the fan center following the tornado shaped flow pattern and then forced away from the central region by the rotating blades.

Isovelocity contours The simulated isovelocity contours are shown in Figure 7. These contours provide an indication of the momentum transport within the building. The only kinetic energy source within the whole building is the rotating blades. This energy is transported from blades to their neighboring air through their stirring. Then the accelerated air propagates the momentum throughout the whole building through convection and diffusion. As shown in Figure 7, the neighboring air has the highest kinetic energy. The shape of the isovelocity surface is strongly affected by the blade shape and the rotation direction, as shown in Figures 7 (a) and (c). As the propagated kinetic energy goes far away from the blade, a large portion of the energy is used to involve more air, and the remaining energy goes to the small scale eddies and eventually
is dissipated. The intensity decays quickly. Finally in the area far from the fan(s), the blade rotation does not play the only role in the air movement anymore. The effect of geometries, boundaries conditions, and the interactions between the fans becomes significant. The isovelocity surface becomes more and more complicated as shown in Figure 7 (b) and (d).

![Figure 7. Isovelocity contours for the two simulated cases.](image)

**CONCLUSION** The RNG k-ε two-equation model is used to predict the airflow pattern created by HVLS stirring fan(s) in a typical dairy free-stall barn. The simulated air velocity contour maps, airflow patterns, streamlines and isovelocity surfaces are analyzed. The following conclusions are drawn:

- A tornado shape vortex is developed by the rotating of a HVLS stirring fan. The air below the fan is sucked upward toward the negative pressure region created at the central region of the blades and eventually forced outward by the blades. This phenomenon counters people’s intuitive feeling, in which a downward airflow would be created by a fan.
- HVLS fans can create a relatively uniform velocity field in the breathing zones.
- The effect of blade shape and rotation velocity play a dominated factor in the airflow pattern and air movement in the region close to them, but this effect decays quickly as air moves away from the blades.
- The installation of HVLS fans within buildings plays an important role in developing the air velocity field, but further research is required to develop consistent application guidelines for use in livestock facilities.

**Acknowledgements** We thank Delta-T Corporation (doing business as Big Ass Fan Co.), Lexington KY USA for blade geometry and related assistance in this study.
REFERENCES


