Assessing the ventilation effectiveness of naturally ventilated livestock buildings under wind dominated conditions using computational fluid dynamics

Tomas Norton, Jim Grant, Richard Fallon, Da-Wen Sun

1. Introduction

Natural ventilation is regularly used in calf housing. The design features of many naturally ventilated calf buildings usually meet environmental recommendations for both winter and summer climates by providing eave openings that allow prevailing winds to force fresh air into the building, and ridge openings that allow warmed air to escape, either by the venturi effect in windy conditions or by thermal buoyancy during calm conditions (Albright, 1990). Unfortunately, the ventilation requirements of these structures are frequently compromised by the need to avoid the cold stressing of calves.
as calf producers often need to modify eave openings in order to suppress the wind-induced ventilation of a building (Nordlund, 2007).

Providing adequate indoor conditions for farm animals requires an understanding of the climate distribution within the animal occupied zones (AOZ) of a building and its relationship to the ventilation configuration and the ambient outdoor environment. In naturally ventilated buildings, the indoor environment is directly influenced by the outdoor wind speed and direction, as well as its turbulent nature. Barrington et al. (1994) and Christiaens (1994) found that thermal effects can be neglected in cattle houses during most of the year and that airflow through the animal house is mainly affected by the orientation of cattle building with respect to the prevailing wind direction. Consequently, the indoor climate is predisposed to large variations, which can often have a detrimental effect on both air quality and animal comfort. Young livestock, such as calves, is susceptible to these variations in the environment (Kelly, 1983). In fact, environmental diseases act as a major limiting factor in Irish calf production (Doherty et al., 2001) and, as the links between these diseases and the housed environment are profound (Lago et al., 2006), preventing outbreaks among calves requires the ability to maintain ventilation and thermal comfort as homogenous as possible and within pre-specified thresholds in the AOZ’s.

Numerical modelling techniques such as computational fluid dynamics (CFD) can offer an effective means of accurately quantifying the climatic variables within ventilated buildings under various outdoor conditions. Over the last decade CFD has found widespread use by agricultural engineers, because of the advent of cost effective computer hardware and high quality CFD software. The present state-of-the-art in commercial CFD allows it to be used as a design-tool that allows several virtual prototypes to be efficiently tried and tested in sufficient detail before structural decisions are made (Norton et al., 2007). However, unlike some other disciplines, the modern capabilities of CFD have not been fully exploited in farm-building design. For example, in building physics many CFD applications combine flow-field analyses with ventilation effectiveness indices such as predicted percentage of dissatisfied people (PPD), draught ratio (DR), and local mean age of air (LMA) in order to gain a full description of the environment. In contrast, when using CFD for farm-building design, the studies to date have used velocity and temperature distribution to gain insight into the indoor environment, and as yet have not

**Fig. 1 – The design specifications of experimental building used in the present study, dimensions in m.**
determined ventilation effectiveness and thermal comfort on a local level, even though such influences can impact on the productivity of environmentally sensitive animals.

With respect to the environmental demands of calves, the typical windy climate experienced in Ireland, and the high-resolution offered by CFD, the objectives of this study were to conduct an assessment of ventilation effectiveness of a typical commercial naturally ventilated calf housing system on both a global and local level for different WIs, and for a range of different eave opening heights. Local ventilation and thermal comfort indices, which are described later, were used to determine the system performance. The CFD model was firstly validated using a half scale model before being applied to the full-scale system. The results of the present study should be helpful for farm-building engineers, and advisors, enabling them to determine correct building design, and to specify the functionality of zones inside the building, based on the prevailing outdoor wind environment.

2. Methods and materials

2.1. The experimental building

The duopitch climatic building is a structure widely employed to house animals in Europe, and consequently is used in the present experimental and numerical studies. The experimental facility is a reduced scale model of such a building with a length scale ratio of 1/2, and the roof pitch and side openings that can be altered as required under the research protocol. The structural envelope of the experimental building is illustrated in Fig. 1 and the locations of the measurement positions used in the experimental analysis are illustrated in Fig. 2.

Outdoor measurements were conducted in order to maintain consistency between the CFD model’s boundary conditions and the actual outdoor climate, and to provide a reference for any indoor measurements taken. A mast equipped with sensors was installed 5 m southeast of the experimental building. External wind speed and direction were measured at a frequency of 5 Hz using a 2D sonic anemometer (Windsonic, Gill, UK; accuracy 0.03 m s$^{-1}$), located on top of this mast, 4 m above the ground level. As the CFD model in this study describes only steady state conditions, all measurements were carried out under moderately stable weather conditions. Indoor measurements were taken at a frequency of 10 Hz, with a 3D sonic anemometer (CSAT3, Campbell Scientific, USA; resolution < 40 mm s$^{-1}$), and the data produced by the two instruments was averaged over 10 min periods. The sonic anemometer inside the building was mounted on a frame, which allowed the anemometer to be manually moved to each measurement position (illustrated in Fig. 2) once measurements at each location were complete.

![Fig. 2 – The plan and elevation showing the distribution of measurement points throughout the experimental building.](image)
For the current investigation, the experimental validation was completed for two wind conditions, namely a wind blowing at a mean azimuth of 215°, i.e. almost normal to the building eave openings, and a wind blowing at a mean azimuth of 295°, i.e. 75° to the normal vector of the building.

To determine the complete airflow pattern, a branch of statistics known as circular statistics was used to analyse the wind data. Circular statistics has been specifically developed for data that is cyclical in nature. A good example of the advantages of using circular statistics can be shown by using the geological definition of wind azimuths (Jones, 2006). Since 0° and 360° represent the same direction, i.e. due North, the cyclicity implies that if observations are made in directions of 2° and 358°, a simple linear mean would indicate a direction of 180°, or due South, whereas it is known that the correct average, or dominant azimuth, is actually North.

2.1.1. The angular mean (preferred direction)
With ordinary linear data, the sample mean is often calculated to indicate the central portion of the observed data. With an angular data set the preferred orientation of a sample is analogous to this measure, but takes cyclicity into account. It is calculated by summing the unit vectors to form a combined vector (Jones, 2006).

\[
\begin{align*}
\gamma &= \begin{cases} 
\tan^{-1} \left( \frac{S}{C} \right) & S > 0, C > 0 \\
\tan^{-1} \left( \frac{S}{C} \right) + \pi & C > 0, S > 0 \\
\tan^{-1} \left( \frac{S}{C} \right) + 2\pi & S > 0, C < 0
\end{cases},
\end{align*}
\]

(1)

where S and C are determined as follows

\[
C = \sum_{i=1}^{N} \cos \theta_i,
\]

(2)

\[
S = \sum_{i=1}^{N} \cos \theta_i.
\]

(3)

2.1.2. The von Mises (circular-normal) distribution
As with linear data the normal distribution of angular data is the most commonly used distribution from which one can make statistical inferences that allow decisions to be made about the data set. This is known as the von Mises distribution, for which the density distribution is given by

\[
f(\theta) = \frac{1}{2\pi I_0(\kappa)} e^{\kappa \cos(\theta - \gamma)}, 0 \leq \theta < 2\pi, 0 < \pi < 2\pi, \kappa \leq 0,
\]

(4)

where \( \gamma \) represents the preferred vector orientation of the population, \( I_0(\kappa) \) denotes the modified Bessel function of the first kind and order 0. \( \kappa \) (kappa) is the concentration parameter that indicates how closely the vectors \( \varphi \) cluster around \( \gamma \), i.e. it

Fig. 3 – Quantile plots of the distributional form of the sample observations to a theoretical distribution for both uniform and von Mises distributions, for (A) the 215°, and (B) the 295° wind conditions studied in the present investigation.
is an indication of the spread of the von Mises distribution, where a high concentration, i.e. >5, indicates a tightly grouped distribution and a low concentration reflects large variability in the distribution (Jones, 2006). For both wind conditions analysed in the present study, the distribution of observations of wind azimuths closely follows the von Mises distribution, thereby validating the use of angular mean and concentration quantities to determine wind directional statistics in the building (Fig. 3). To calculate the concentration, $\kappa$, the following equations was used,

$$
\kappa = \begin{cases} 
2 + \frac{R^3 + S^3}{R^3 + S^3} & R < 0.53 \\
-0.4 + 1.39 R + \frac{0.43}{R} & 0.53 \leq R < 0.85 \\
\frac{1}{(R^2 - 4R^2 + 3R)} & R \geq 0.85
\end{cases}
$$

(5)

where

$$
R = \sqrt{C^2 + S^2}
$$

(6)
is the vector length; $C$ and $S$ have been previously defined and

$$
\bar{R} = \frac{R}{n}
$$

(7)
is the resultant vector length of the directional sample for $n$ observations. Fig. 4 (a and b) shows that for both of the wind conditions studied the concentration values were typically very high, thereby promoting confidence in the value of the angular/vector mean predicted by Eq. (1). The time-series of outdoor wind azimuths for each measurement point (averaged over a ten minute periods) is presented in Fig. 5.

![Fig. 4](image-url)
length and 0.13 m wide (area 2.67 m$^2$). The eave openings and the total area of its openings was 9.6 m$^2$ (2.5%). Both eave and ridge opening areas were kept in line with recommendations of Defra (2003), who stated that inlet and outlet areas should be at least 0.05 m$^2$ and 0.04 m$^2$ per calf, respectively. These areas were adjusted in simulations that examined the influence of the building opening area on the ventilation performance.

2.2.2. Theoretical considerations of CFD

For all simulations a segregated solver was used to solve, in a sequential manner, the governing equations of fluid flow. These can be written as follows:

$$\nabla \cdot \vec{v} = 0. \quad \quad (8)$$

$\frac{\partial \vec{v}_i}{\partial t} + \rho \vec{v} \cdot \nabla \vec{v}_i = -\nabla p + \mu \nabla^2 \vec{v}_i. \quad \quad (9)$

As previously shown in greenhouse studies, airflows in naturally ventilated structures are highly turbulent (Boulard et al., 2000). Therefore, it was necessary to introduce a turbulence model in order to account for the fluctuating velocity components, which would otherwise cause the CFD solutions to diverge in steady state. The standard $k$-$\varepsilon$ model (Lauder and Spalding, 1974), which assumes isotropic turbulence, was adopted to describe turbulent transport in the present simulations. This choice represents a good compromise between a realistic description of turbulence and computational efficiency. The complete set of the equations of the $k$-$\varepsilon$ model can be found elsewhere (Mohammadi and Pironneau, 1994).

The commercial finite volume CFD code STAR-CCM+ (CD-Adapco, 2008) was used in all simulations. The governing equations were solved over discrete spatial volumes using the 2nd order convection scheme. A converged solution was achieved by solving directly in steady mode, i.e. using local time-steps that are small in regions where the mesh is fine and large where the mesh is coarse.

2.3. CFD evaluation of the building performance

2.3.1. Air change effectiveness parameters

In this study a quantity known as the LMA, $\theta$, was used to define an air change effectiveness parameter, called the air change ratio, which is introduced below. In building physics, LMA is usually defined as the average time for a parcel of air to travel from a supply inlet area to any location in a ventilated zone, $\theta = t_{av}/t'$ (Etheridge and Sandberg, 1996). Recently many authors have successfully used CFD to calculate LMA by including a transport equation for a passive scalar in the equation set (Chanteloup and Mirade, 2009; Abanto et al., 2004). For steady state conditions, this equation takes the following form:

$$\nabla \cdot (\rho \vec{v} \theta - I \nabla \theta) = S_c. \quad \quad (10)$$

In order to obtain an integrated scalar value of the “age” of the fluid it is necessary to set the source term in this equation equal to the fluid density. Also, because the age of air can only be convected in time, its ability to diffuse must be suppressed in CFD simulations. Bearing this in mind, the diffusion coefficient has been estimated using the following equation (Chanteloup and Mirade, 2009; Abanto et al., 2004).

$$\Gamma_s = 2.88 \times 10^{-5} \rho \frac{\mu_i}{S_c^{1/2}}. \quad \quad (11)$$

where $S_c$, the turbulent Schmidt number took the value of 0.7. The boundary conditions for the solution of Eq. (10) were zero at air inlets and a zero gradient at air outlets and at solid wall surfaces.

By volumetrically averaging the residence time for all air molecules, i.e. the LMA distribution, in a room or single-zone building, the mean total residence time of fresh air in the zone, $\theta_z$, can be determined. This variable can then be used in accordance with the distributed values of LMA to calculate ratio of the global to local air change rates:
The parameter $ACR_{AOZ}$ is determined using the volumetric average of LMA, i.e. the reciprocal of the local ventilation rate, in each of the pen regions within the building, and therefore quantifies the local environment using a technique that is potentially more relevant to animal health than just air velocity or LMA alone.

2.3.2. Ventilation rate

Many CFD studies of greenhouse ventilation have used the step-down method to compute ventilation rates. This can be done by including a tracer gas with the same physical properties as air and of a known uniform initial concentration. Then the tracer gas can be treated as a passive scalar and can be solved after the flow-field solution has been obtained, i.e. by decoupling it from the momentum equations. The species equation used can be written as follows:

$$\frac{\partial c}{\partial t} = \nabla \cdot (\rho \mathbf{v} c - f \nabla c),$$

where $c$ represents the concentration of the tracer gas in a cell; $t$ is the time in seconds; and the R.H.S of the equation represents the tracer gas flux through each computational cell in the building's volume.

Initially all the cells in the building have a fixed tracer gas concentration equal to unity and all the external cells equal to zero. In this way, once an appropriate time-step, $dt$, is
selected, the decay in concentration of the tracer can be predicted over time. The tracer gas concentration decreases in the building at a rate that depends on the local values of the air velocity. Once all the tracer gas has been evacuated from the building, the volumetric average tracer gas concentration can be tabulated as a function of time and exported to a statistical package so that an exponential decay of the following form can be fitted:

$$\tau = \tau(0)e^{-nt},$$  \hspace{1cm} (14)

where the exponent $n$ is the decay rate of this function and therefore describes the ventilation rate of the studied volume (air changes per hour).

2.3.3. Minimum comfort temperature

Evaluating the impact of specific climate parameters on animal performance is a difficult task, as not only does the ambient temperature affect the thermal environment, but so also does air speed, the radiant environment, and, in some circumstances, humidity. Unfortunately, temperature and air velocity are, in many cases, independently considered when designing livestock housing, which undoubtedly limits the ability to understand the full implications of these designs on animal comfort.

In order to combine the influence of the most important environmental parameters on a calf’s thermal comfort, an index called the “minimum comfort temperature” was developed for the purpose of this study. This index is a simple modification of the standard operative temperature (Beaver et al., 1996) and can be defined as the minimum blackbody temperature needed for an animal to maintain thermo-neutrality in prevailing environmental conditions:

$$T_{em} = T_b + \left(\frac{R_{bs}}{R_{b}}\right)(T_{crit} - T_b),$$ \hspace{1cm} (15)

where $T_{em}$, the equivalent blackbody temperature, is equal to the local air temperature (Campbell, 1986), $R_{bs}$ is the total thermal resistance afforded by the animal in the environment and $R_b$ is the total resistance afforded by the animal in a controlled thermo-neutral environment, i.e. in air speeds of 0.1. To calculate the total thermal resistance of each calf the following equations were employed (Bruce, 1993):

$$r_a = (5.3 + 15.7\overline{v}^{0.6}M^{-0.13})^{-1},$$ \hspace{1cm} (16)

$$r_h = 0.0074d,$$ \hspace{1cm} (17)

$$r_t = 0.03M^{0.33},$$ \hspace{1cm} (18)

where $r_a$ is the thermal resistance of the boundary layer of air around the animal, $r_h$ is the coats thermal resistance, $r_t$ is the thermal resistance afforded by the tissue. It is important to note that Eq. (16) also includes the net radiative heat transfer between the housed animal and its surroundings. By summing Eqs. (16)–(18) $R_b$ of Eq. (15) was calculated, where $\overline{v}$ = local velocity magnitude. On the other hand $R_{bs}$ was calculated similarly with the exception of $\overline{v} = 0.1 \text{ m s}^{-1}$. Following the work of Bruce (1993) $T_{crit}$, the calf’s lower critical temperature, was calculated using the following equation:

$$T_{crit} = T_b - \left(\frac{Q_s}{A_{crit}}\right)(r_a + r_h) + \left(\frac{Q_s}{A_{crit}}\right)(r_s + r_h).$$ \hspace{1cm} (19)

where $T_b$, the deep body temperature of the calf, was taken to be 39 °C and $Q_s$, the total sensible heat emitted, was calculated according to the following equation (CIGR, 2002):

$$Q_s = 6.44M^{0.7} + \frac{6.65(6.28+0.0188M)}{0.85}.$$ \hspace{1cm} (20)

To estimate the evaporative heat loss ($Q_e$), cutaneous and respiratory water loss data presented by Gebreinedhin et al. (1981) for Holstein calves over a range of room temperatures was averaged between 0 and 10 ºC. This led to the following equation for total evaporative heat loss by considering the latent heat of evaporation of water:

$$Q_e = 2270(9.44 \times 10^{-5}M).$$ \hspace{1cm} (21)

The body weight of the calves to be housed was determined, given their age in days, using the relation presented by Gebreinedhin et al. (1981). In the present study, calves were assumed to be a week old and therefore had a body weight of approximately 40 kg.

By computing the values of $T_{crit}$ in the building during the CFD simulations, it can be combined with Eq. (15) to determine the minimum comfort temperature ($T_{em}$) as a function of the AOZ location within the building. Then, from the distribution of $T_{em}$ within each AOZ, a maximum value of $T_{em}$ can be chosen to act as the minimum design temperature for the building.

---

**Table 1 – Details of the mesh refinement study**

<table>
<thead>
<tr>
<th>Case</th>
<th>Overall grid density</th>
<th>Building mesh density</th>
</tr>
</thead>
<tbody>
<tr>
<td>Grid 1</td>
<td>121,073</td>
<td>50,000</td>
</tr>
<tr>
<td>Grid 2</td>
<td>150,000</td>
<td>70,400</td>
</tr>
<tr>
<td>Grid 3</td>
<td>200,000</td>
<td>100,000</td>
</tr>
</tbody>
</table>

---

Fig. 7 – Mesh refinement study which compares velocity predicted along three lines across the building, at 0.5 m above floor level. The red line represents the finest mesh; the blue line represents the coarsest mesh and the red line represents the moderately refined mesh.
2.4. CFD mesh and boundary conditions

2.4.1. Mesh details
The most important aspect of the CFD model, as regards its meshing, was the eave and ridge openings areas which, in order to resolve the small length scale and sharp gradient in the airflow, required a high-resolution mesh. On the other hand, near the top of the computational domain, cells could be relatively large. The governing equations were solved using an unstructured mesh, comprising polyhedral elements, which was built in commercial CFD software STAR-CCM+ (CD-Adapco, 2008). Such a meshing system was found to conform easily to the exact boundary of the building. The size of the CFD domain used during the simulations was chosen in

Fig. 8 – Experimental and numerical predictions of airflow patterns in the experimental building on the vertical plane x(1): (a) polar plots on the plane at the measurement points; (b) CFD predicted 2D velocity vectors on the plane; and (c) comparison of predicted and measured velocity vectors at the measurement points overlaying interpolated concentration values [Eq. (5)].

Fig. 9 – Experimental and numerical predictions of airflow patterns in the experimental building on the vertical plane y(1): (a) polar plots on the plane at the measurement points; (b) CFD predicted 2D velocity vectors on the plane; and (c) comparison of predicted and measured velocity vectors at the measurement points overlaying interpolated concentration values [Eq. (5)].
order to ensure that the position of the outer boundaries did not compromise the CFD solution. Fig. 6(a–e) illustrates the CFD model of the full-scale building, the volume mesh distribution, as well as the typical residual decay and monitoring point curves obtained during the course of the simulations.

The optimum mesh density was determined by completing a mesh-convergence study, during which a number of simulations were run with different mesh densities, until finally a representative mesh density was deemed an adequate compromise between computational efficiency and model accuracy. Three different meshes have been made by refining the cells of the coarsest (initial) mesh by about a factor of $\sqrt{2}$. Some of the mesh characteristics are given in Table 1. The results of the mesh-convergence study for the experimental building are displayed in Fig. 7, where the predicted air velocity magnitudes along three horizontal lines A, B and C at a height of 0.5 m from the floor of the experimental building are presented. No large differences

![Image](58x130 to 285x600)

![Image](314x130 to 543x585)

Fig. 10 – Experimental and numerical predictions of airflow patterns in the experimental building on the vertical plane $x(4)$: (a) polar plots on the plane at the measurement points; (b) CFD predicted 2D velocity vectors on the plane; and (c) comparison of predicted and measured velocity vectors at the measurement points overlaying interpolated concentration values [Eq. (5)].

Fig. 11 – Experimental and numerical predictions of airflow patterns in the experimental building on the vertical plane $y(2)$: (a) polar plots on the plane at the measurement points; (b) CFD predicted 2D velocity vectors on the plane; and (c) comparison of predicted and measured velocity vectors at the measurement points overlaying interpolated concentration values [Eq. (5)].
between the solutions on the different meshes can be observed. This is probably because in all meshes the control volumes have been clustered quite densely around and within the building. Nonetheless, because velocity magnitudes calculated on mesh 2 and mesh 3 only show minor differences, mesh 2 (medium density) has been used for further study. Mesh 2 was then applied in both experimental building and the full-scale model simulations by maintaining the relative sizes between the building/building openings and mesh control volumes.

2.4.2. Modelling the wind profile

The well-known logarithmic profile, which has been developed on sound physical principles, has been used in successfully CFD wind-engineering studies to represent the lower regions of the atmospheric boundary layer. As the fetch surrounding the experimental building was rural and reasonable planar, the wind boundary layer is well represented by a log-law profile with a roughness length, $z_o$, of 0.01 m for a barren rural area (Richardson and Blackmore, 1995) and is thus used in the present CFD simulations. It can be represented in the following form:

$$U_z = \frac{u'^2}{k} \ln \left( \frac{z + z_o}{z_o} \right).$$  \tag{22}

Richards and Hoxey (1993) 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 wind-engineering simulations employing $k$-$\varepsilon$ turbulence model representation of turbulence. The variables $k$, $\varepsilon$ and frictional velocity ($u_*$) represented by the following equations were used in all the simulations:

$$k = \frac{u'^2}{\varepsilon},$$  \tag{23}

$$\varepsilon = \frac{u'^3}{K(z + z_o)},$$  \tag{24}

$$u_* = \frac{Ku_{ref}}{\ln [(z_{ref} + z_o)/z_o]}.$$  \tag{25}

At the outlet of the computational domain, the flow-field is extrapolated from the adjacent interior cells by assuming the normal gradient vanishes, and an overall mass balance correction is then performed.

A no-slip condition was enforced at the bottom of the computational domain, with the equivalent sand-grain roughness ($k_o$) modifications according to the formulae by Cebeci and Bradshaw (1977). The turbulent wall function in STAR-CCM+ is

![Fig. 12 – Comparison of predicted and experimental velocity profiles in the experimental building for a wind blowing normal to the building, at a vertical line positioned at the intersection of plane y(2) with all x planes illustrated in Fig. 2.](image-url)
equivalent to that used in FLUENT CFD software, and can be represented by the following (Blocken et al., 2007):

$$\frac{u_p}{u_e} = \frac{1}{k} \ln \left( \frac{u_p y_p}{u e C_S k_s^+} \right) + 5.43,$$

(26)

where the dimensionless sand-grain roughness height $k_s^+ = (u_k k_S) / v$ and the required relationship between the “equivalent” sand-grain roughness height $k_S$ (m), the aerodynamic roughness length $z_o$ (m) and the roughness constant $C_S$ have been derived by Blocken et al. (2007), via first order matching of the wall function and the atmospheric boundary layer:

$$k_S = \frac{9.793 z_o}{C_S},$$

(28)

In this study $k_{S,ground} = 0.1$ m, which was as large as possible while satisfying $k_{S,ground} < y_P$. While this resulted in $k_{S,ground} < k_{S,ABL} = 0.39$ m, the boundary layer retained its horizontal homogeneity because the region of interest, i.e. the building, was placed reasonably close to the CFD inlet boundary without effecting the solution, as advised by Blocken et al. (2007).

As discussed above, the mean outdoor wind direction for both experimental runs was found to be $215^\circ$ and $295^\circ$. Because the experimental building was orientated at an azimuth of $220^\circ$, and because significant deviations in the direction of the wind were observed (Fig. 5), a wind blowing parallel to the buildings normal vector was confidently assumed in the present numerical analysis. Also, for the mean outdoor wind direction of $295^\circ$, a wind direction of $75^\circ$ to the normal vector was modelled in the CFD simulations. A wind speed of $5$ m s$^{-1}$ was taken as representative of typical wind conditions in Ireland (Met Eireann, 2009). It is recognised that the ventilation efficiency and the thermal comfort of a naturally ventilated building must also be understood under periods of high and low wind speed. However, because such a topic deserves full and comprehensive study, only one wind speed is currently considered.

3. Results

3.1. Model validation

3.1.1. Indoor airflow patterns

In order to validate the CFD predictions of airflow patterns in the experimental building, two measurement planes for each wind condition were taken as the reference planes for

![Fig. 13 – Comparison of predicted and experimental velocity profiles in the experimental building for a wind blowing at an oblique angle of 75° to the building, at a vertical line positioned at the intersection of plane y(2) with all x planes illustrated in Fig. 2.](image-url)
comparing experimental with numerical results. Specifically, the measurement planes $x(4)$ and $y(2)$, shown in Fig. 4, were chosen for this task, as the flow features found on these planes were unique to the specific wind condition analysed.

In Figs. 8(a)–11(a) the experimentally obtained frequency distributions of airflow patterns are illustrated as polar plots on the utilised measurement planes of the building. Following the technique described by Boulard et al. (2000), the probability densities of indoor air speed and direction were calculated, then placed into their appropriate bins at 10° intervals from 0° to 360°, and finally plotted with the centre of each plot located at the measurement position. In this way, the mean flow direction on the 2D plane together with the deviations in the flow could be easily represented (Boulard et al., 2000). From the frequency distributions of airflow patterns Eqs. (1)–(7) were used to accurately determine the mean direction at the measurement position and the mean air speed was taken as the average value exhibited in a 10° interval of the resultant vector.

In Figs. 8(b)–11(b), the 2D velocity vectors predicted by the CFD simulations are displayed. While discrepancies exist, similarities are evident when Figs. 8(a)–11(a) and Figs. 8(b)–11(b) are visually compared. However, in order to reduce the subjectivity when analysing the relative flow distribution patterns, both the measured and predicted 2D velocities are displayed as vectors at each measurement position of the plane in question [Figs. 8(c)–11(c)]. Moreover, these vector plots overlay interpolated concentration values for the measured airflow, determined with Eq. (5), which acts as a metric for the confidence in the measured airflow pattern. As mentioned previously, concentration values $>5$ mean a tightly grouped data set that can be considered accurate for use in CFD validation. However, for concentration values $<5$ the comparison between predictions and measurements must be viewed cautiously, and closer examination of the directional distributions must be considered before using the data to validate the CFD model.

In Fig. 8 (a–c) the recirculating airflow pattern which is often associated with this type of building is evident. Both experimental and numerical results show that the recirculation is positioned in the windward region of the building. Moreover, even though the concentration values are below 5, confidence in the experimental representation of this flow pattern is strengthened by noting that the distribution of flow direction in the most windward portion of the building is grouped in a way that shows a mainly positive vertical airflow,

---

Fig. 14 – Comparison of experimental data (Bartak et al., 2002) published CFD predictions (Chanteloup and Mirade, 2009) and present CFD predictions of profiles of normalised LMA in three areas of a ventilated room.
whereas in the middle and leeward portion of the building the results are strongly grouped in a negative vertical direction. The CFD predictions illustrated in Fig. 9(b) show that, on the \( y(2) \) measurement plane, two counter-rotating vortices exist adjacent to each other in the lower region of the experimental building. In this case, less agreement is found with experimental measurements in the upper region of the building. However, the measured airflow was found to flow towards the ground when immediately adjacent to the both gable walls. While this airflow was found to rise again in the middle of the building, less confidence can be placed in the experimentally determined flow pattern in this region as not only are the concentration values quite low, but other smaller flow features appear to occur. This is more than likely to be caused by the variations in outdoor wind direction over the course of the measurements (Fig. 5). Nevertheless, the symmetry that exists between flow patterns in the gable regions of the building suggest that in a more unidirectional outdoor wind condition CFD simulations may form a reasonably close representation of reality.

Fig. 10(a–c) shows the measured and predicted flow regime found along the gable wall [at plane \( x(4) \)] for a wind azimuth of 295°, i.e. 75° to the normal vector of the building. Here, the most important feature is the high concentration values coupled with the excellent agreement between predictions and measurements of the airflow pattern in this region. This strengthens the use of circular statistics when validating CFD predictions in naturally ventilated buildings, and presents concentration as an excellent metric for quantifying the spread of directional data such as airflow. While the main flow feature is predicted well, the sparse array of measurements mean that it not possible to know for sure whether the small recirculation zone predicted towards the windward region of the building occurs in reality.

Finally, Fig. 11(a–c) illustrates the predicted and measured flow patterns along the \( y(2) \) plane for the oblique wind condition. Here also, good qualitative agreement between two numerical and experimental results is evident. Of greatest interest is the large recirculation shown to exist in the far right hand side of the building by both the experimental measurements and the CFD predictions. However, by studying more closely the appropriate polar plot, positioned at \( x = 4 \) m and \( y = 0.3 \) m, it can be seen that the centre of the recirculation zone actually exists in the upper region of the building. In contrast, the centre of the recirculation zone is predicted by CFD to occur in the lower region of the building.

3.1.2. Indoor velocity magnitude
Figs. 12 and 13 compare the predicted and experimental velocity profiles in the experimental building for winds blowing normal and oblique to the building at a vertical line formed by the intersection of plane \( y(2) \) with all \( x \) planes (Fig. 2). The normalised air velocity magnitude was obtained by the ratio of the interior air velocity to the mean external wind speed at 4 m high in both experiments and CFD simulations. As expected, for the normal wind direction, good agreement is evident at the planes nearest the gable wall. However, the CFD model under-predicts the high velocity measured in the vicinity closest to the ground for the both middle planes. The normalised velocity magnitudes predicted for the oblique wind condition are in poor agreement for measurement lines at \( x(1) \) and \( x(2) \). However, in line with the airflow pattern analysis completed above, the agreement improves at measurement line of \( x(4) \).

Overall, the agreement with experiments was reasonably good considering that in all cases the value of experimental wind velocity represents an average value over the measurement period and therefore neglects turbulence, while the
value estimated by the CFD model includes turbulence even if the $k-\varepsilon$ model is only a rough approximation of reality (Bartzanas et al., 2004). Moreover, the variation of the outdoor wind condition can cause increase in the complexity of air distribution patterns, which can in turn reduce the agreement between experimental and numerical results.

3.1.3. Local mean age of air (LMA)

The LMA predictions were compared to experimental measurements completed by Bartak et al. (2001, 2002) for a 45 m$^3$ (4.2 m long, 3.6 m wide and 3 m high) laboratory with isothermal mixing conditions. This data has also recently been used to validate CFD predictions of LMA (Chanteloup and Mirade, 2009), and so was used for validation purposes in this study. The room in question had only one 0.3 m x 0.2 m supply-air opening placed symmetrically on a lateral wall and only one air outlet, of the same dimensions, located on the ceiling close to the lateral wall opposite the air supply. The air inlet was specifically designed in order to generate the most uniform as possible profile of air inlet velocity. The LMA distribution in the room was evaluated experimentally at 23 locations in the test room with a reasonable repeatability of 6.5% using SF$_6$ as tracer gas coupled with the step-down method. In addition, the results of Bartak et al. (2002) were presented in a dimensionless form, with the LMA quantity was normalised as follows:

$$\bar{\gamma} = \frac{\theta}{V/\bar{q}}$$  \hspace{1cm} (29)

Fig. 14(a–c) illustrates similar CFD predicted profiles for LMA to those measured by Bartak et al. (2001, 2002). In fact, both the CFD and experimental results exhibit approximately constant values of LMA for approximately 2/3 of room height, with a sharp decrease in LMA in the zone above this, i.e. closer to

Fig. 17 – LMAV and VIM predicted ventilation rates under various WIs for the three buildings analysed.

Fig. 18 – Streamlines illustrating the airflow patterns coloured by LMA distribution Building (3). (A) The building orientation, (B) 30° WI, and (C) 20° WI.
the inlet. Furthermore, predictions of LMA values are in very close agreement with the numerical values reported by (Chanteloup and Mirade, 2009).

Nonetheless, while differences between the experimental and the numerical predictions exist, this does not exceed 20%, which is a discrepancy commonly reported in this type of computation (Bartak et al., 2001). Bartak et al. (2002) attributed this discrepancy to the deviation in experimental repeatability (6.5%), experimental accuracy (5%), uncertainty in accurately determining the volume of the test room (5%), and the use of mean velocities instead of instantaneous values.

3.2. Using CFD to calculate ventilation rates with LMA and tracer-decay methods

A popular method of determining the ventilation rate of a building is by quantifying the decay rate of a tracer gas, using either the step-up or step-down method (Chanteloup and Mirade, 2009). The step-down method is commonly incorporated in CFD investigations of natural ventilation, most notably in studies of greenhouse technology (Bartzanas et al., 2004). In order to use this technique during CFD simulations all computational cells in the building model must be filled with a virtual tracer gas of known concentration (usually unity). Also, the computational cells in domain surrounding the building must contain a known concentration of the tracer gas (usually zero). The CFD simulation is then run to determine to decay rate of the tracer in the building. The main effectiveness of this method can be attributed to the fact that the exact knowledge of the inlet and outlet opening areas are not demanded from the CFD solution. For some wind conditions and depending on the ridge and/or eave openings are employed, air can enter while at the same time leaving the building through these openings (Kelly, 1983). Therefore, in such circumstances the exact knowledge of the ventilation rate can be tedious to determine from CFD solutions without employing a contaminant decay method.

Unfortunately, although being an accurate and effective technique of computing the ventilation rate, use of the step-down technique in CFD simulations is time consuming, since not only does the solution need to be restarted in transient mode, but further processing is required by an external statistical package to compute the ventilation rates. Alternatively, by volumetrically averaging the LMA distribution it is possible to compute the residence time of air in the building, and by inverting this value the ventilation rate can be determined far more efficiently than by using the step-down tracer-decay method. This means that once the LMA solution is obtained in steady state, the ventilation rate can be determined with very little further processing and without exporting results from the CFD package.

![Image](image_url)

**Fig. 19** – Streamlines illustrating the airflow patterns coloured with LMA distribution in Building (3). (A) The building orientation, (B) 90° WI, and (C) 60° WI.
Fig. 15 compares the predictions of ventilation rate by both LMA and step-down techniques, for different size time-steps used in the step-down method. Fig. 15 shows that the solutions predicted by both methods are similar for all time-steps with an error of around 4% and 0.25% evident for time-step size of 30 and 1 s, respectively. Most importantly, the step-down solution converges to the LMA solution upon decreasing time-step size. Therefore, all the ventilation rates computed in this study were done using the LMA method.

3.3. Environmental study of the commercial calf house for different WIs

To examine the influence of WI and opening area on the buildings ventilation performance, three naturally ventilated buildings were modelled with the same physical configuration as the full-scale building presented above, apart from eave opening areas, which were varied as a multiple of the Building (1) opening area. Therefore, Building (1) (the basic structure) had an eave opening area of 3.5 m², Building (2) had an eave opening area of 7 m² and Building (3) had an eave opening area 10.4 m². WI angles were simulated in 10° increments from 0° to 90° using the same wind boundary conditions as those presented earlier.

3.3.1. Building ventilation rates

The openings in naturally ventilated greenhouses and livestock buildings can simultaneously allow air to enter and exit the building, depending on the incidence of the oncoming wind. Using the STAR-CCM + CFD package it was possible to determine the proportion of the buildings opening area used as an inlet by each building, for all WIs. Fig. 16(a) shows the percentage of opening area (POA) used as an inlet by Building (1), where it is interesting to find that for most wind directions, i.e. from 0° to 50°, a portion of all the buildings openings always acts as an inlet. Specifically, the windward eave opening provides 66% POA as an inlet at a WI of 0°, which then exponentially rises to 100% at a WI of 30° to the building. For the ridge opening, there is a non-linear decrease in the POA available for inflow of 50–0%, at in the WI range of 0–90°, respectively. Similarly, for the leeward eave opening, the POA available for inflow decreases from 66% to 25% in the WI range of 0–90°, respectively.

As shown in Fig. 16(b), both Building (2) and Building (3) exhibit similar profiles in terms of the POA used for inflow as a function of WI. Most notably, there is evidence of symmetry in the percentage area used for inflow by both the windward and leeward openings, i.e. the windward opening area accepting incoming air increases from 60% at WI of 0° to 100%
at a WI of 30%, with the decrease in the leeward POA occurring with a similar profile.

Fig. 17 shows the air inflow across the building openings determined for each opening by integrating the incoming velocity component over the appropriate cross-sectional area [Fig. 16(a and b)] and then summing the individual flow rates. To keep the terminology concise in this paper, from this point on the technique for predicting the air influx through the building openings will be termed the ventilation influx method (VIM). The ventilation rate has also been predicted using the LMA method LMAV, and is also presented in Fig. 16, where it is evident that general agreement exists between the predicted profiles of VIM and LMAV. However, unexpectedly, there are discrepancies between VIM and LMAV profiles for many WI, suggesting that the level of mixing in the building changes as function of WI. Closer examination of the data in Fig. 17 shows that LMAV is generally greater than VIM for WI in the range of 0–40\degree, whereas between 60\degree and 90\degree the LMAV is less than VIM. These differences suggest that two different ventilation phenomena occur in the building when it is exposed to these two sets of WI ranges. The flow regimes that yielded these contrasting results will be discussed in the following paragraphs. Bearing in mind that the ventilation performance of Building (2) and Building (3) are very similar (Figs. 15 and 16), and as the profiles of LMAV and VIM are similar for Building (1), only Building (3) will be the subject of the following analysis.

For Building (3), it is possible to give a physical explanation of why the LMAV is higher than VIM in the WI range of 20–40\degree through a streamline analysis of the 3D indoor airflow patterns. Fig. 18 (a and b) illustrates these indoor flow patterns at WI of 20\degree and 30\degree, respectively. Specifically, Fig. 18(a) shows that one recirculation zone, induced by a WI of 20\degree, successfully occupies the full building. Consequently, the LMAV value for this WI is quite similar the VIM value (Fig. 16). However, as illustrated in Fig. 17(b), when the wind blows at a 30\degree incidence to the building the primary recirculation zone only occupies about 2/3 of its total volume. Thus, air movement in the remaining portion of the building is mildly influenced by the primary vortex, and it therefore rotates with much less vigour. So, in effect the eave opening ventilates a smaller volume than that of the building itself, resulting in a volumetric average of LMA equal to 48 s comparing to the theoretical residence time of 59 s, which has been calculated by dividing the volume of the building, 872 m\(^3\), by the ventilation influx, 11.7 m\(^3\) s\(^{-1}\).

While LMAV is a more accurate measure of the ventilation rate experienced by the building, its value relative to VIM seems useful as a quick indicator of environmental homogeneity. For example, considering the results of these simulations, when a value of LMAV is higher than VIM the level heterogeneity in the building has been augmented by the outdoor wind condition, with a secondary recirculation zone now present in the buildings volume.

In order to rationalise why LMAV was less than VIM for WIs between 60\degree and 90\degree, a further streamline analysis was conducted [Figs. 19(a and b) and 20(a and b)]. For a wind blowing normal to the building, Figs. 19(a) and 20(a) show that a large quantity of the high velocity air stream leaves the building before mixing with the indoor air. In fact, only the air entering the building in the regions nearest to the gable walls, which extend into the building for about 1/8 of the buildings length, integrates with the indoor flow regime. Fig. 20(a) illustrates well the “short-circuiting” that occurs over most of the length of the building. This phenomenon results in lower LMAV values for a WI of 90\degree than for a WI in the range of between 40\degree and 50\degree. The rapid movement of air into the building at the gable walls acts as the main driving force behind the two vortices that rotate in each half of the building. Indeed, because two small vortices are formed, instead of the large single vortex witnessed above for a WI of 20\degree and 30\degree.
conditions inside the building are heightened, and as a result the size of the zones experiencing lower than average ventilation is reduced considerably. As evident in Figs. 19(b) and 20(b), the growth of the second vortex in the building begins at a WI of 60°, and increases until two vortices of equal size are rotating in the building at a WI of 90°. The overall level of mixing in the building is at its maximum at 70° and no further improvement in LMAV is found when the wind veers towards the buildings normal vector. In the WI range of 70–90°, LMAV predictions were found to be lower than VIM because “short-circuiting” existed in the flow regime. This further adds to the ability of these two parameters (VIM and LMAV) to provide an indication of the quality of the indoor ventilated space.

The effect of short-circuiting (Fig. 17) should be considered when calculating the ventilation rate of naturally ventilated livestock structures using formulae based on the ventilation rate coefficient, such as those developed by Hellickson and Walker (1983), Choinière (1991), Verlinde et al. (1998) and ASHRAE (2001):

\[
q = C_v A_{\text{inlet}} u_{\text{ref}} ; \quad (30)
\]

where \(C_v\) takes on different values depending on which of the above studies is referenced. For example, if the ventilation rate of Building (3) was calculated using VIM then the present CFD simulations show that the building would be actually under-ventilated by 18 air changes h\(^{-1}\) below the design value calculated with Eq. (30) at a 5 m s\(^{-1}\) outdoor wind speed. Unfortunately, considering the effect of short-circuiting is not easily quantifiable as the discrepancies between LMAV and VIM predictions may not only be a function of the eave opening size but may also be influenced by roof pitch, presence of overhangs, baffles etc. Therefore, further studies are required to provide better understanding of the influence of these building parameters on the ventilation performance in respect of LMAV and VIM. Until such research has been conducted, design ventilation rate values predicted by formulae of the type given by Eq. (30) should be reduced by about 20% based on the findings of the present study. However, this value should be considered as more indicative than precise.

3.3.2. Local ventilation rates

Fig. 21(a–c) presents the strong environmental heterogeneity that exists inside the calf house at WI between 0° and 30°, and shows that this diminishes as the WI approaches the buildings normal vector. In fact, the ACR\(_{\text{AOZ}}\) in all pens does not become stable until the WI lies in the range 30–90° for the tested buildings. Within this stable region, 3 out of 6 pens in Building (1) performs satisfactorily increasing to 4 pens for Building (2) and (3). Overall, the best performing AOZ’s are represented by pens 2 and 5 for the tested buildings, as these provide a level of ventilation close to that of the building under most WIs. The level of heterogeneity inside the building increases in accordance with increasing opening area, as the standard deviation of ACR\(_{\text{AOZ}}\) increases from 18% of the average ACR\(_{\text{AOZ}}\) in Building (1) to 28% of the average ACR\(_{\text{AOZ}}\) in Building (3).

Fig. 22 (a–c) illustrates the volumetric average of LMA*, which is the LMA normalised by the maximum value in the building, for each pen as a function of WI for the tested buildings. As the distribution of LMA* in pens 1, 2, 4 and 5 follow similar profiles they are depicted together in Fig. 22(a),
whereas Fig. 22(b and c) contain LMA* profiles within pens 3 and 6, respectively. From Fig. 22(a) the distributions of LMA* as a function of WI show an increasing level of mixing in the building as the WI approaches the normal vector of the building. For a WI of 10° and 20°, strong heterogeneity in LMA* is evident. However this reduces to constant relative values for all WI from 40° through 90°. Importantly, Fig. 22(a–c) shows that pens 3 and 6 are under-ventilated, i.e. containing close to the oldest air in the building. In fact, these two pens under-perform by about 40% when compared to all the other pens, with pen 6 containing high values LMA* when the level of mixing in the building is weakest, i.e. at WIs of 40–50°.

3.3.3. Minimum comfort temperatures

Using the relevant equations, the minimum comfort temperatures in each building were calculated for each WI and then volumetrically averaged over each AOZ. To ensure that this comfort criterion could be met throughout the building, it was necessary to determine the maximum AOZ averaged value for each WI. These maximum values are presented in Fig. 23, where it is evident that to operate without affecting calf comfort, minimum comfort temperatures of 12.7 °C, 13.8 °C and 15.2 °C are, on average, required for Buildings (1), (2) and (3), respectively. These temperatures are quite high when considering that Ireland experiences lower average temperatures for most months of the year, and that supplementary heat is rarely employed in climatic animal housing. Therefore, using climatic buildings with unobstructed openings to house young calves may not provide adequate thermal comfort for the animal in cold weather conditions, unless feed rate is increased.

4. Discussion

In the streamline study, the development of a second vortex in the building over the WI range of 60–90° was shown to influence strongly the degree of mixing of air in the indoor flow regime. This is because the second vortex is directly driven by the wind rather than it being a secondary recirculation zone. For the buildings tested, secondary recirculation zones, whose motion is mildly influenced by the buildings primary vortex, have been observed to form in the WI range of about 20–50°. Thus, these zones lack a sufficient supply of fresh air for adequate air turn over in the affected regions especially during periods of low wind speeds. More research is required for a complete analysis of the influence of various wind speeds and wind direction on the air change rate in these zones.

Similar variations in ventilation rate as a function of WI to those presented in Fig. 17 have recently been reported by Teitel et al. (2008) for greenhouse structures of a different ventilation configuration. A ventilation study with a building of similar configuration to that analysed here was conducted by Shklyar and Arbel (2004), where they observed that the highest ventilation rate occurred with a wind blowing perpendicular to the openings. However, considering that Shklyar and Arbel (2004) used the VIM to determine the ventilation rate, their predictions agree very well with the results reported in the present study. Unfortunately, Shklyar and Arbel (2004) considered their VIM predictions to represent accurately the ventilation rate without consideration for the dynamics of the internal flow regime. In the current investigation, we have demonstrated that consideration must be given to the decay rate of a tracer gas, either via numerical (LMAV or step-down method) or by experimental analyses, before an accurate measure of the ventilation rate of a building can be concluded. This is especially important when housing young animals who may be sensitive to poorly ventilated regions in a building.

From the current investigation, it can be concluded that when completing a tracer gas study, the placement of the sampling equipment in the regions that will give the representative measure of ventilation is extremely important because ventilation rates can be easily over predicted if sampling was not conducted in the secondary recirculation zones. The correct choice of sampling points should arise from an investigation of air turn over in various regions of the
building through the coupling CFD and experimental analyses. In the current investigations a rapid technique for determining whether much effort should be given to the selection of sampling points for a climatic building is to compare the relative values of the LMAV and VIM ventilation rates. When LMAV/VIM > 1 heterogeneity in ventilation performance, i.e. a secondary recirculation zone, exists in the building, and therefore emphasis should be placed on sampling point position; for LMAV/VIM = 1 the building can be assumed uniformly mixed and for LMAV/VIM<1 short-circuiting occurs in the flow regime. Such quantities can be easily obtained from CFD investigations.

The results obtained show the difficulty associated with achieving adequate thermal comfort for all the calves housed in a climatic building exposed to different wind conditions. However, this study has successfully highlighted the regions of the building where ventilation performance may be problematic. The results obtained in this study have application in the design of many European livestock buildings. With this in mind, the location of animals and other obstructions such as pen partitions, feed storage or quarantine regions etc., which vary from building to building, were not considered in the present study. Considering that a CFD model is only as good as the boundary conditions with which it is supplied, little perceived advantage is to be gained by using sophisticated meshing techniques and mathematical models to artificially try to increase the accuracy of the simulations.

While in the present study CFD predictions agreed reasonably well with experimental results, limitations in the physical modelling of turbulence means that some features of the flow regime may not have been adequately captured by the simulations (Norton et al., 2007; Teitel et al., 2008). More specifically, when modelling flow regimes with $k$-$\varepsilon$ turbulence models, inaccuracies often occur because of adverse pressure gradients, impinging flows, and strong streamline curvature occurring in the flow (Norton et al., 2007). Finally, it is important to aware of the opportunities and limitations associated with CFD before implementing the results found in this paper.

5. Conclusions

A CFD model was developed to investigate the ventilation effectiveness of a naturally ventilated livestock building under various WIs and with three different eave opening conditions. Two different methods of numerically determining tracer-decay rate in these buildings with CFD were tested, and the LMA technique termed the LMAV, was found to outperform the step-down technique both in terms accuracy and in terms of efficiency. It was found that ventilation rates were not at their highest when wind was blowing normal to the building because a considerable quantity of the flow left the building via “short-circuiting”. However, the greatest ventilation homogeneity was experienced when the wind was blowing normal to the building, owing to the formation of two wind-driven vortices in the building. Results also showed that the highest level of environmental heterogeneity occurs at WI of 10–40° because the primary vortex only occupies a portion of the total building volume. Moreover, the present CFD simulations have shown that in some circumstances the ventilation rate determined from the flow rate through the building openings may not accurately represent the actual ventilation rate of a building, with measurements/simulations of contaminant decay being a more accurate measure of ventilation rate. A thermal comfort index has also been developed during the course of this study, which has shown that climatic livestock buildings perform best in temperatures ranging from 13 to 15 °C in typical wind conditions found in Ireland.

REFERENCES


Choini`ere Y (1991). Wind induced natural ventilation of low-rise buildings for livestock housing by the pressure difference


Norton T; Sun D -W; Grant J; Fallon R; 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(12), 2386–2414.


