Three-Dimensional Buoyant Turbulent Flows in a Scaled Model, Slot-Ventilated, Livestock Confinement Facility

Steven J. Hoff  
_Iowa State University_, hoffer@iastate.edu

Kevin A. Janni  
_University of Minnesota–Twin Cities_

Larry D. Jacobson  
_University of Minnesota–Twin Cities_

Follow this and additional works at: [http://lib.dr.iastate.edu/abe_eng_pubs](http://lib.dr.iastate.edu/abe_eng_pubs)

Part of the [Agriculture Commons](http://lib.dr.iastate.edu/abe_eng_pubs) and the [Bioresource and Agricultural Engineering Commons](http://lib.dr.iastate.edu/abe_eng_pubs)

The complete bibliographic information for this item can be found at [http://lib.dr.iastate.edu/abe_eng_pubs/338](http://lib.dr.iastate.edu/abe_eng_pubs/338). For information on how to cite this item, please visit [http://lib.dr.iastate.edu/howtocite.html](http://lib.dr.iastate.edu/howtocite.html).
Three-Dimensional Buoyant Turbulent Flows in a Scaled Model, Slot-Ventilated, Livestock Confinement Facility

Abstract
A three-dimensional turbulence model was used to determine the effects of animal-generated buoyant forces on the airflow patterns and temperature and airspeed distributions in a ceiling-slot, ventilated, swine grower facility. The model incorporated the Lam-Bremhorst turbulence model for low-Reynolds Number airflow typical of slot-ventilated, livestock facilities. The predicted results from the model were compared with experimental results from a scaled-enclosure. The predicted and measured results indicated a rather strong cross-stream recirculation zone in the chamber that resulted in substantial three-dimensional temperature distributions for moderate to highly buoyancy-affected flows. Airflow patterns were adequately predicted for Arc > 40 and J values < 0.00053. For Arc < 40 and J values > 0.00053, the visualized patterns indicated that the jet separated from the ceiling before the opposing end-wall. This discrepancy was attributed to variations in the experimental and numerical inlet flow development assumptions.

Keywords
Animal housing, Ventilation, Airflow

Disciplines
Agriculture | Bioresource and Agricultural Engineering

Comments
This article is from Transactions of the ASAE 35, no. 2 (1992): 671–686.

This article is available at Iowa State University Digital Repository: http://lib.dr.iastate.edu/abe_eng_pubs/338
THREE-DIMENSIONAL BUOYANT TURBULENT FLOWS IN A SCALED MODEL, SLOT-VENTILATED, LIVESTOCK CONFINEMENT FACILITY

S. J. Hoff, K. A. Janni, L. D. Jacobson
MEMBER ASAE  MEMBER ASAE  MEMBER ASAE

ABSTRACT
A three-dimensional turbulence model was used to determine the effects of animal-generated buoyant forces on the airflow patterns and temperature and airspeed distributions in a ceiling-slot, ventilated, swine grower facility. The model incorporated the Lam-Bremhorst turbulence model for low-Reynolds Number airflow typical of slot-ventilated, livestock facilities. The predicted results from the model were compared with experimental results from a scaled-enclosure. The predicted and measured results indicated a rather strong cross-stream recirculation zone in the chamber that resulted in substantial three-dimensional temperature distributions for moderate to highly buoyancy-affected flows. Airflow patterns were adequately predicted for $AR_c > 40$ and $J$ values $< 0.00053$. For $AR_c < 40$ and $J$ values $> 0.00053$, the visualized patterns indicated that the jet separated from the ceiling before the opposing end-wall. This discrepancy was attributed to variations in the experimental and numerical inlet flow development assumptions.

KEYWORDS. Animal housing, Ventilation, Airflow.

INTRODUCTION
Winter ventilation of livestock confinement facilities is characterized by low ventilating rates in relatively large spaces and high animal densities. The animals produce a substantial amount of sensible and latent heat, which, combined with low ventilating rates, can produce non-uniform temperature distributions and air velocities within the ventilated space. The goal of the engineer is to distribute this limited amount of fresh inlet air as efficiently as possible and to provide an acceptable microclimate near the animals that maintains well-being.

A typical arrangement has fresh air entering the ventilated space through a slot-diffuser adjacent to the ceiling along one wall. Fans in the opposing wall provide the required pressure differential necessary for the desired fresh-air exchange rates. The resulting ceiling jet influences the velocity and temperature distributions throughout the space.

Little information exists on velocity and temperature distributions throughout a ventilated space beyond the inlet-jet-affected region. Spatial conditions generally have been described qualitatively based on overall airflow patterns. The microclimate of the animal, as a function of the inlet conditions for winter conditions when the potential for animal chilling is the greatest, is of prime concern.

Mixed-flow ventilation research related to animal confinement facilities has attempted to characterize the desired inlet conditions based on the inlet jet behavior as it enters the ventilated space. In the past, recommendations specified that the desired inlet velocity be maintained between 4.0 and 5.0 m/s. Recent research highlighted the importance of air-mixing and the effect of buoyant forces on the behavior of the inlet jet as it enters the confinement facility (Barber et al., 1982; Leonard and McQuitty, 1986). The effect of buoyant forces on inlet jets were summarized by Randall and Battams (1979) using the inlet corrected Archimedes Number ($AR_c$).

The application of mathematical models to simulate airflow in livestock facilities has been pursued before. Timmons et al. (1980) applied an inviscid two-dimensional model to a slot-ventilated livestock facility. Janssen and Krause (1988) applied a two-dimensional model that described velocity, temperature, and contaminant distributions in slot-ventilated livestock facilities. The model used an augmented laminar viscosity to account for turbulence effects in the building. Choi et al. (1987, 1988, 1990) applied the isothermal fully turbulent k-ε model to a two-dimensional slot-ventilated enclosure. They investigated the distributions of velocity and contaminants with and without obstructions and found very reasonable agreements with experimental results.

Many questions exist regarding the relation between the $AR_c$ and the spatial variability of temperature and velocity, which ultimately affects the thermal comfort of the confined animals. Animals can be adversely affected at some levels and combinations of velocity and temperature (Riskowski and Bundy, 1990). Current computing resources and modeling equations and techniques make it possible to address questions regarding velocity and temperature distributions in confinement livestock facilities.

The purpose of this project was to develop and verify a three-dimensional, turbulent-buoyant, numerical model for the analysis of slot-ventilated, livestock confinement facilities. Existing turbulence models and solution techniques were used to develop a three-dimensional...
numerical model for the analysis of slot-ventilated livestock confinement facilities. The numerically predicted airspeed and temperature distributions were compared with data from a 1/5 scale-model slot-ventilated swine growing/finishing facility. The numerical model's capabilities for analyzing ventilating airflow in livestock confinement facilities was assessed.

This study makes frequent reference to a 1/5 scale-model facility. The 1/5 scale-model is similar to that used in the study of Barber and Ogilvie (1984). That is, the air chamber used to verify the turbulence model is geometrically 1/5 the size of a typical swine growing/finishing facility.

BUILDING DESCRIPTION AND GOVERNING PARAMETERS

A schematic of the building studied is shown in figure 1. This facility represents a 1/5 scale-model swine growing/finishing facility identical in size to the apparatus reported in Barber and Ogilvie (1984). This facility had a single port exhaust located at the center of the exhaust end-wall. Ventilating air entered the building through a continuous end-wall slot located adjacent to the ceiling.

The chamber was 2.40 m long, 2.00 m wide, and 0.64 m high. The building was empty, with a uniformly heated floor simulating a dense population of animals. The building was symmetrical with respect to the X-Y plane located through the exhaust fan center-line. Only data from one-half the chamber is reported. It is shown in Hoff (1990) that symmetry did exist in the experimental apparatus.

DIMENSIONLESS PARAMETERS

Recent research projects have characterized the desired inlet-jet conditions in terms of how the inlet buoyant and inertial forces affect the trajectory of the inlet jet. The ratio between inlet buoyant and inertial forces characterizing the inlet-jet can be summarized in terms of an inlet-corrected Archimedes Number ($\text{Ar}^e$).

Winter inlet design recommendations also were developed to create a horizontally stable jet pattern upon entrance to the ventilated space (Randall and Battams, 1979; Barber et al., 1982; Leonard and McQuitty, 1985, 1986). Recommendations were given for inlet conditions that result in acceptable air mixing conditions in the ventilated space. The air-mixing criteria is summarized in a Jet Momentum Number ($J$) (Barber et al., 1982).

Three dimensionless parameters were used to identify the ventilating conditions for this non-isothermal arrangement. The Jet Momentum Number ($J$) (Barber et al., 1982) was used to describe the momentum of the entering jet and is defined as (all variables defined in nomenclature section):

$$J = \frac{Qu_0}{gV}$$

The inlet corrected Archimedes Number ($\text{Ar}^e$) defined as (Randall and Battams, 1982):

$$\text{Ar}^e = \frac{C_d g A_0 WH (W + H) (T_s - T_i)}{Q^2 (546. + T_s + T_i)}$$

was used to estimate the buoyant forces on the incoming chilled jet. It has been widely used for livestock facilities (Barber et al., 1982; Leonard and McQuitty, 1985, 1986). Finally, the Raleigh Number ($\text{Ra}_H$) based on building height ($H$):

$$\text{Ra}_H = \frac{\rho g B (T_s - T_i) H^3}{\mu^2}$$

was used to estimate the natural (or free) convection effects expected in the building as a result of the heat from the simulated animals.

These three parameters defined key elements of this ventilation arrangement. Jet Momentum describes the anticipated level of air-mixing, the Archimedes Number describes buoyancy affects on the inlet jet, and the Raleigh Number describes the overall buoyancy effects in the building.

For well-mixed, non-drafty flows, the Jet Momentum should be kept between 0.00075 and 0.0015 (Barber et al., 1982; Ogilvie et al., 1988). Minimizing buoyancy affects requires that the Archimedes Number be kept below 50.0 (Leonard and McQuitty, 1986), and minimizing natural or free-convection effects requires the Raleigh Number based on building height to be kept at or below $4.0 \times 10^8$ (Torrance and Rocket, 1969).

For this project, the $J$ values varied between 0.00029 and 0.00126, the $\text{Ar}^e$ values varied between 13.2 and 88.5, and the $\text{Ra}_H$ varied between $1.1 \times 10^8$ and $4.0 \times 10^8$. Thus, it was expected that buoyancy effects, both on the jet and internally, would play a vital role in the overall airflow patterns and airspeed and temperature distributions. This was indeed the case as the following results indicate.

The governing dimensionless parameters and levels tested are with respect to the 1/5 scale-model chamber shown in figure 1. The scaling guidelines used were those reported in past research studies. In particular, Barber and Ogilvie (1984), as well as Moog (1981), concluded that for buoyancy affected flows, the scaling guidelines for minimizing distortion should include geometric similarity for all physical dimensions and dynamic similarity using...
the inlet corrected Archimedes Number. Inlet corrected Archimedes Number similarity required the maintenance of similar temperature differences between model and prototype and adjusting the inlet velocity to achieve an equivalent \( \text{Ar}_{c} \).

The Jet Momentum Number (J), Raleigh Number (Ra), and Inlet Corrected Archimedes Number (Ar) are included to indicate the flow regime in the model chamber. A full-scale prototype was not available to assess the similarity with the findings from the model chamber. In particular, the \( \text{Ar}_{c} \) is assumed as the appropriate scaling factor for this non-isothermal ventilation arrangement. Future research will determine the adequacy of this assumption. The results presented here apply only to the model chamber simulated and experimentally studied.

MODEL DESCRIPTION

Two major turbulence models have been developed. The Fully-Turbulent \( k-\varepsilon \) model (FTKE) was developed by Harlow and Nakayama (1969) and refined by Launder and Spalding (1972). The FTKE assumes that all points within the solution grid exist in a region of full-turbulence \( (y^+ > 11.63) \). For regions where this requirement is not met, such as solid boundaries, wall-functions are used (Patankar, 1970; Launder and Spalding, 1974).

The Low-Reynolds Number \( k-\varepsilon \) model (LRKE) was developed (Launder and Spalding, 1972) to eliminate the need for special treatments, such as wall-functions, in regions where fully-turbulent conditions do not exist. The LRKE model requires a more refined grid, relative to the FTKE model, and thus has been limited to those cases where the FTKE cannot be used.

Lam and Bremhorst (1981) developed a LRKE model which retains the features of the FTKE model and is applicable to near-wall boundaries. In this version, if the flow is indeed turbulent, features of the well-tested FTKE are retained. The Lam-Bremhorst Low-Reynolds Number (LBLR) model was used for this project. It can be shown (Hoff, 1990) that the physical dimensions of the inlet slot, used for this project, prevented the FTKE model from being used. The following describes the main features of the turbulence model incorporated for this research project. The description is brief. References have been given for those interested in a more complete discussion.

GOVERNING DIFFERENTIAL EQUATIONS

All equations solved for this research project were cast into:

\[
\frac{\partial (\rho u \phi)}{\partial x_i} = \frac{\partial}{\partial x_i} \left( \Gamma_{\phi} \frac{\partial \phi}{\partial x_i} \right) + S_{\phi} \tag{4}
\]

Each transport equation is characterized by the scalar quantity of interest, \( \phi \), a diffusion coefficient, \( \Gamma_{\phi} \), and the expression for the source terms, \( S_{\phi} \). Equation 4 represents the generalized steady-state form of the defining relation used for the numerical technique developed by Patankar and Spalding (1972). All differential equations involving convective and diffusive transport processes can be cast into equation 4. Table 1 lists the defining relations for the seven, time-averaged, partial differential equations solved for this research project. The seven partial differential equations, shown in Table 1, represent the conservation of mass \( (\phi = 1) \), conservation of momentum \( (x, y, \text{ and } z\text{-directions}) \), conservation of energy \( (\phi = C_{p}T) \), conservation of turbulent kinetic energy \( (\phi = k) \), and the dissipation of turbulent kinetic energy \( (\phi = \varepsilon) \). Several auxiliary relations are required for the successful solution to this problem. The auxiliary relations are summarized in Table 2. The major features of the model and required auxiliary, relations are outlined below. Interested readers are referred to Patel et al. (1985) and Chen et al. (1990) for a more detailed description.

**Effective Viscosity**

Turbulent flow is characterized by random, chaotic fluid motion at any given point over time. Numerically modeling this behavior would require a very fine grid to resolve the transport of scalar components from point-to-point.
accommodate this phenomena, relatively coarse grids are used in conjunction with additional transport equations to account for the turbulent distributions of scalar quantities. In particular, an equation describing the kinetic energy of turbulence, \( k \), is incorporated along with an estimate of viscous dissipation, \( \varepsilon \), on turbulent transport (see Table 1). The LBLR model combines these parameters to define an effective viscosity, as (Chen et al., 1990):

\[
\mu_{\text{eff}} = \mu_l + \rho c_p f_\mu \frac{k^2}{\varepsilon}
\]  

(5)

The factor, \( c_\mu \), is a constant of 0.09 and \( f_\mu \) is a damping function that depresses the effect of turbulence near a solid boundary. The governing differential equations for \( k \) and \( \varepsilon \) are solved with the mass, momentum, and energy equations which in turn are used to calculate the effective viscosity. The process evolves in an iterative fashion until convergence is attained.

**BUOYANCY CORRECTIONS**

The problem solved for this project was mixed-flow, thus a description of the temperature distribution was sought along with the momentum and turbulence equations. Two buoyancy terms are included in the model to account for the effects of temperature.

The vertical momentum was adjusted by the addition of a vertical force associated with density variations as:

\[
F_B = -\rho_{\text{ref}} (1 + \beta (T - T_{\text{ref}})) g
\]  

(6)

Implied within equation 6 is the Boussinesq assumption where the density can be estimated as:

\[
\rho \propto \rho_{\text{ref}} (1 + \beta (T - T_{\text{ref}}))
\]  

(7)

The Boussinesq assumption is applicable when the expected maximum temperature difference is small relative to the absolute mean temperature of the ventilated space \( (T/(T_{\text{ave}} + 273.) < 1.0) \). The second buoyancy term added was the assumed turbulence production accounted for by buoyancy affects. This term is incorporated as:

\[
G_B = -\rho \beta \frac{\partial (T - T_{\text{ref}})}{\partial y}
\]  

(8)

**AUXILIARY RELATIONS AND CONSTANTS**

The governing differential equations outlined in Table 1 need many auxiliary relations and constants to successfully solve the governing differential equations. These auxiliary relations (Table 2) account for the turbulence level near solid boundaries. The damping function incorporated in the LBLR model was (Lam and Bremhorst, 1981):

\[
f_\mu = \left(1.0 - e^{-A_R R^+}\right)^2 \left(1.0 + \frac{A_k}{R^+}ight)
\]  

(9)

For viscous dominated regions \( (y^+ < 11.63) \), \( f_\mu \) nears 0.0. Conversely, for fully turbulent regions \( (y^+ > 1.63) \), \( f_\mu \) approaches a value of 1.0. In fully turbulent regions, \( f_\mu \) can exceed (slightly) a value of 1.0 (Lam and Bremhorst, 1981). Thus, for viscous dominated regions, the laminar viscosity governs momentum diffusion:

\[
\mu_{\text{eff}} = \mu_l
\]  

and for fully turbulent regions, the turbulent viscosity governs:

\[
\mu_{\text{eff}} = \mu_t
\]  

(11)

The damping function \( f_\mu \) is in turn a function of the level of turbulence at any given point in the solution domain. Turbulence levels were measured with a turbulent Reynolds's Number defined as:

\[
R_t = \frac{k^2 \rho}{\mu_t}
\]  

(12)

or

\[
R_k = \frac{k^{1/2} \rho y_p}{\mu_l}
\]  

(13)

The relation \( R_t \) describes the turbulent conditions within the ventilated space based on the calculated turbulent parameters \( (k \) and \( \varepsilon ) \). The \( R_t \) values range from 0.0 near a solid boundary to infinity for regions that are fully turbulent. Similarly, \( R_k \) describes the turbulent conditions as a function of the location relative to the nearest solid boundary. The \( R_k \) values vary between 0.0 near a solid boundary to a positive bounded number at locations removed from a solid boundary. The quantity \( R_k \) is approximately equal to the dimensionless distance, \( y^+ \), defined as:

\[
y^+ = \frac{c_{\mu}^{1/4} \rho k^{1/2} y_p}{\mu_l}
\]  

(14)

The \( y^+ \) relation is used to describe the dimensionless distance used in describing the “law of the wall” for turbulent boundary layers (Kays and Crawford, 1980). The difference being a multiple of the turbulent viscosity constant, \( c_{\mu} \).

The relations \( R_{1t} \), \( R_k \), and \( y^+ \) are turbulent-based Reynolds's Numbers which define the level of turbulence near solid boundaries. These dimensionless numbers are used exclusively in the turbulence model to properly adjust the near-wall viscosity term for the prediction of shear stress. In equation 14, \( y^+ \) is introduced to relate the near-wall turbulence levels commonly referred to in the literature (Kays and Crawford, 1980).

Figures 2 and 3 represent these relations graphically for the grid points located near the ceiling. Figure 2 shows the predicted damping function \( (f_\mu) \) for each of the first four grid points removed from the ceiling. At a physical distance of 19.0 mm from the ceiling, the damping function approaches (and slightly exceeds) 1.00 implying fully
turbulent airflow. Figure 3 plots the turbulent Reynolds Number ($y^+$) for the first four grid points removed from the ceiling. Near the ceiling (lines + and x), the $y^+$ values are at or below the viscous dominated levels ($y^+ < 11.63$). Likewise, removed from the ceiling (lines $\diamond$ and $\Delta$), the $y^+$ values approach levels associated with fully turbulent airflows ($y^+ > 60$).

Figure 4 summarizes the overall effect of the damping function ($f_D$) on adjustments made to the turbulent viscosity ($\mu_t$) according to the level of turbulence ($y^+$). The $\mu_t$ distributions shown in figure 4 indicate that as the distance from the ceiling increases (+ line vs. $\Delta$ line), the turbulent viscosity increases. Also, very near the ceiling (lines + and x), where viscous effects dominate, the turbulent viscosity nears a value of 0.0.

Figures 2 to 4 represent the major differences in capability between the FTKE and LBLR models. The FTKE model does not provide a means of adjusting the turbulent viscosity near solid boundaries. As a consequence of this, all grid points that are placed near a solid boundary must be in a region of full turbulence. If grid points are placed too near a solid boundary, the effective viscosity will be augmented relative to the actual turbulence levels. The end result is false diffusion near the boundary which, for ceiling or wall jets can greatly affect the resulting airflow patterns (Hoff, 1990). Normally, one would adjust the grid to accommodate this discrepancy. For ceiling-slot ventilation arrangements however, the near-ceiling grid point is fixed at one-half the inlet slot-width. This subtlety becomes especially important when modelling airflow behavior in scaled-enclosures where the inlet slot-width becomes very small.

**SOLUTION PROCEDURE**

Solutions of the governing differential equations was obtained using the control-volume based numerical scheme developed by Patankar and Spalding (1972) and summarized in Patankar (1980). The code, written in FORTRAN, was extended by Hoff (1990) from two to three dimensions. The resulting code was verified using available literature accompanied with numerical and experimental results (Patankar, 1988; Hjertager and Magnussen, 1976). All equations were solved simultaneously using under-relaxation techniques in a purely iterative line-by-line sweeping fashion (Patankar, 1980). The solution technique used was the SIMPLER algorithm developed by Patankar (1980) and represents a revised version of the previously used SIMPLE algorithm (Patankar and Spalding, 1972).

All solutions were performed on supercomputers available at the Minnesota Supercomputer Institute. Initial trials were performed on the ETA-10, with final runs...
completed on the Cray-2 and Cray-XMP supercomputers. In general, a total of 3,000 (with mildly buoyancy affected) to 12,000 (with highly buoyancy affected) iterations were required at about 1.0 CPU second per iteration to obtain a converged solution.

**Boundary Conditions and Numerical Grid**

The boundary conditions and numerical grid used are shown in Table 3. Symmetry was used and these are indicated by the boundary conditions at z = 1.00. The inlet conditions are also shown in Table 3. The turbulent kinetic energy and viscous dissipation inlet conditions used were the same as those used by Patel et al. (1985).

The numerical grid used was a 21 x 22 x 10 grid for the x, y, and z-directions, respectively. Grid points in the y-direction were spaced evenly throughout, while grid points in the y and z-directions were non-uniform. In the y-direction, grid points were concentrated near the ceiling-slot region and near the floor region. Spacings were governed by a geometric progression factor of about 1.8 to alleviate stability problems associated with high aspect ratio control-volumes (Kuehn, 1990). In a similar fashion, the z-direction grid was concentrated near the symmetry plane.

The overall grid used was coarse. The non-uniform grid spacing in the y and z-directions concentrated the grid near regions where large gradients were expected (i.e., ceiling-jet profile) which resulted in an efficient use of a limited grid (Patankar, 1980).

**Experimental Verification**

The numerically predicted results were compared with experimentally determined airspeed and temperature distributions from a 1/5 scale-model swine growing/finishing facility. The facility is nearly identical in size and configuration to the apparatus used by Barber (1981) and reported in Barber and Ogilvie (1984).

Air enters the chamber through a slot-diffuser located adjacent to the ceiling. The inlet slot-diffuser width was adjustable between 0.0 and 20.0 mm. Controlled temperature and humidity air (Aminco-Aire, Model 8-5540) entered the ceiling slot-diffuser through an inlet plenum. The velocity profile of air entering the chamber was fully-developed. The ventilating rate of the entering air was measured with an orifice plate located upstream of the inlet plenum. Orifice plate design and entrance/exit lengths required were designed according to ISO Standards (1983).

**Measuring Apparatus**

Airspeed and temperature results were recorded at 105 locations within the chamber. Simultaneous airspeed and temperature results were recorded. Airspeed measurements were taken with an omnidirectional transducer (TSI, Inc., Model 8473). This transducer was developed to accommodate low airspeed levels typical of this research project. Temperatures were measured with a t-type thermocouple attached directly to the airspeed transducer. Time-averaged behavior was estimated by collecting samples at 5 Hz for 180 seconds. For each point, 900 measurements were used to estimate the average and standard deviation. The sampling rate and duration were verified experimentally and agreed with past work on similar arrangements (Thorshaug, 1982). Recent research indicates that the sampling rate may be lower than optimal (Jin and Ogilvie, 1990).

Data acquisition was performed with a Metra-Byte, Inc. (Model DAS-16/EXP-16) board in conjunction with a Zenith 286 computer. Data collection and reduction was performed with Labtech Notebook (Measurement Engineering, Inc.).

**Flow Visualization**

The airspeed transducer was directionally insensitive requiring visualization methods to determine airflow patterns. Flow visualization was accomplished by illuminating selected two-dimensional planes to determine airflow patterns. Flow visualization was accomplished by illuminating selected two-dimensional planes to determine airflow patterns. A light box using tungsten lamps was used to illuminate the planes. Titanium Tetrachloride vials (E. Vernon Hill, Inc.) provided the smoke medium for flow visualization. Airflow patterns were recorded using still photography (1600 ASA) and real-time video.

**Boundary Conditions**

The boundary conditions solved for this problem were adiabatic surfaces except the floor which was maintained at 33.0°C to simulate the pig’s surface temperature. The inlet conditions were specified with a known temperature, slot-width and velocity. A total of six experimental treatments were analyzed for this project. Of these six treatments, three representative runs are presented for brevity. The boundary conditions and corresponding inlet corrected Archimedes (Ar^2), inlet Jet Momentum (J), and Raleigh (Ra_th) Numbers are shown in Table 4. Also included in Table 4 are the expected uncertainties derived from a first-order analysis. Additional data and details are presented by Hoff (1990).

**TABLE 3. Boundary and inlet conditions used in LBLR model**

<table>
<thead>
<tr>
<th>Plane Location</th>
<th>u</th>
<th>v</th>
<th>w</th>
<th>T</th>
<th>k</th>
<th>ε</th>
</tr>
</thead>
<tbody>
<tr>
<td>x = 0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>x = L</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>y = 0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>33.0</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>y = H</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>z = 0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>z = w/2</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
</tr>
</tbody>
</table>

**Inlet Conditions**

- $U_{in}$, $T_{in}$ = specified with experimental treatment
- $V_{in}$, $W_{in}$ = 0.0
- $k_{in}$ = 0.005 $U_{in}^2$
- $\varepsilon_{in}$ = 0.27 $k_{in}^{3/2}$
TABLE 4. Inlet conditions from experiment data for treatments modeled (standard deviation in parentheses)

<table>
<thead>
<tr>
<th>Treatment</th>
<th>Qin (m^3/s)</th>
<th>Tin (°C)</th>
<th>Ts (°C)</th>
<th>As (m^2)</th>
<th>ARE</th>
<th>J</th>
<th>GRe***</th>
<th>RaH</th>
<th>GReH***</th>
</tr>
</thead>
<tbody>
<tr>
<td>one</td>
<td>0.004**</td>
<td>10.2 (0.3)</td>
<td>33.5 (0.3)</td>
<td>0.00018</td>
<td>88.5</td>
<td>18.0</td>
<td>29 x 10^-4</td>
<td>5.9 x 10^-5</td>
<td>4.0 x 10^4</td>
</tr>
<tr>
<td>two</td>
<td>0.012**</td>
<td>11.7 (0.4)</td>
<td>33.3 (0.3)</td>
<td>0.0090</td>
<td>45.5</td>
<td>9.3</td>
<td>53 x 10^-4</td>
<td>11.0 x 10^-5</td>
<td>2.6 x 10^4</td>
</tr>
<tr>
<td>three</td>
<td>0.012***</td>
<td>16.4 (0.5)</td>
<td>33.4 (0.2)</td>
<td>0.0038</td>
<td>13.2</td>
<td>2.7</td>
<td>126 x 10^-4</td>
<td>25.0 x 10^-5</td>
<td>1.1 x 10^4</td>
</tr>
</tbody>
</table>

* ± 6.0 x 10^-5 m^3/s certainty
** ± 1.8 x 10^-4 m^3/s uncertainty
*** ± uncertainties

RESULTS AND DISCUSSION

The results and discussion are presented in three sections. The first section compares the visualized and predicted symmetry plane airflow patterns for three representative inlet and boundary conditions. The second section compares symmetry plane contour plots of airspeed and temperature in the X-Y plane. The third section compares contour plots, surface plots, and airflow patterns in the Y-Z (cross-stream) direction. All orientation references are with respect to figure 1. The accumulated data from a three-dimensional experimental and numerical study is vast. Selected pieces of information are presented to highlight the major features expected in buoyancy-affected turbulent airflow in livestock confinement facilities. Attention is focused on trends and extensive use is made of contour and surface plots.

SYMMETRY PLANE AIRFLOW PATTERNS

Figures 5 to 7 present the predicted (a) and visualized (b) airflow patterns for treatments one (Ar_c = 88.5, J = 0.00029), two (Ar_c = 45.5, J = 0.00053), and three (Ar_c = 13.2, J = 0.00126), respectively. Figures 5 to 7 represent the predicted and visualized airflow behavior at the symmetry plane (x, y, z = 1.00) (see fig. 1). Figure 5 represents typical airflow behavior for a highly buoyancy affected flow, figure 6 represents typical airflow behavior for moderately buoyancy affected flow, and figure 7 represents typical airflow behavior for mildly buoyancy affected flow. In figures 5 to 7, airflow direction is shown as — which is to be interpreted as →.

For highly (Ar_c > 70) buoyancy affected flows, the inlet jet fell almost immediately after entering the ventilation chamber (fig. 5). The behavior was characterized by a sudden drop near the inlet, with reattachment to the floor at X/L = 0.18. One of the more interesting features of highly buoyancy affected flows was the pronounced fluid rise at the floor after floor impingement occurred. After the inlet jet had fallen to the floor, it began to rise almost immediately after traversing along the floor for a very short distance (x/L = 0.24). This behavior was predicted numerically (fig. 5a) and observed experimentally (fig. 5b).

For moderately (30 < Ar_c < 70) buoyancy affected flows, the inlet jet traversed along the ceiling to X/L = 0.48 before separating and falling to the floor (fig. 6). This behavior was predicted numerically (fig. 6a) and experimentally observed (fig. 6b). The agreement was quite good as indicated in figure 6. For mildly (Ar_c < 30) buoyancy affected flows, discrepancies existed between the predicted and observed airflow behavior (fig. 7). For Ar_c < 30, it was numerically predicted that the jet would remain horizontal and attached to the ceiling until impinging upon the exhaust end-wall (fig. 7a). Experimentally, however, this was not the case. For
Ar_c < 30, the horizontally stable jet was observed to lose kinetic energy before the exhaust end-wall. The throw of the jet was insufficient to reach the exhaust end-wall, resulting in ceiling separation before the opposing end-wall. This was a repeatable, observed discrepancy between the numerically predicted and observed jet behavior. The predicted results presented (fig. 7a) actually agree with the jet behavior observed in past studies (Black et al., 1970; Leonard and McQuitty, 1985, 1986), but were in disagreement with what was observed in the chamber used for this research. The discrepancy was attributed to differences in inlet conditions between the experimental apparatus and the numerical model. The numerical model assumed a pure inlet slot diffuser, whereas the experimental apparatus used a ducted approach to the slot inlet.

The different velocity approaches, (i.e., fully developed profile vs. constant inlet profile) affected the throw of the jet. The jet throw of fully developed inlet jet is diminished relative to the throw of a pure slot diffuser jet (Zerbe and Selnas, 1946; Sigalla, 1958). This agrees with the predicted (fig. 7a) and experimentally observed (fig. 7b) results. This effect was noticeable for mildly buoyancy affected flows.
Figure 9—Predicted (a) and measured (b) symmetry plane temperature and airspeed distributions for moderately buoyancy affected flows ($Ar_c = 45.5, J = 0.00053$).

For moderate and highly buoyancy affected flows the buoyancy affects apparently superseded any jet-throw discrepancies encountered as a result of inlet flow development differences. The experimental apparatus was not designed to test the inlet velocity profile discrepancy.

X-Y PLANE AIRSPEED AND TEMPERATURE DISTRIBUTIONS

Figures 8 to 10 present the predicted and measured X-Y symmetry ($z = 1.00$) plane contour plots of temperature (a) and airspeed (b) for treatments one, two, and three, respectively. Contours for airspeed are in 0.01 m/s increments and contours for temperature are in 0.50°C increments. The contour plots are presented for the measured boundary and not the physical boundary of the chamber. For example, the vertical measuring boundary went from $y = 0.03$ m to a maximum of $y = 0.61$ m. This is in contrast to the physical boundary which went from $y = 0.00$ to a maximum of $y = 0.64$ m.

Figure 8 indicates the X-Y plane distributions of temperature (a) and airspeed (b) for highly ($Ar_c = 88.5, J = 0.00029$) buoyancy affected flow and correspond to the airflow patterns shown in figure 5. The inlet jet (fig. 5) falls within $X/L = 0.18$ of the inlet and the effects are apparent from the temperature and airspeed distributions. Near-floor temperatures were lowest near the inlet end-wall (fig. 8a) corresponding to the chilled jet falling. Likewise, near-floor airspeeds (fig. 8b) were largest near the inlet corresponding to the effect of the falling jet. The net effect was a near-floor region at $X/L = 0.17$ that was the coldest, draftiest region in the building.

Figure 9 indicates the X-Y plane distributions of temperature (a) and airspeed (b) for moderately ($Ar_c = 45.5, J = 0.00053$) buoyancy affected flow and corresponds to the airflow patterns shown in figure 6. As for highly buoyancy affected flows, the effect of the intermediately falling chilled-jet is quite noticeable from the contours of temperature (fig. 9a). Figure 9a indicates that there was a measurable depression in temperature corresponding to the falling chilled jet at $X/L = 0.48$. The 30.5°C isotherm was swept well into the floor region as a result of the falling chilled-jet (fig. 9a).

Figure 10 indicates the X-Y plane distributions of temperature (a) and airspeed (b) for mildly ($Ar_c = 13.2, J = 0.00126$) buoyancy affected flow and corresponds to the airflow patterns shown in figure 7. Observed jet behavior indicated that the throw of the jet was insufficient...
(a) Predicted Results

Predicted Temperature (°C)

Predicted Airspeed (m/s)

(b) Measured Results

Measured Temperature (°C)

Measured Airspeed (m/s)

NOTE:

Figure 10—Predicted (a) and measured (b) symmetry plane temperature and airspeed distributions for mildly buoyancy affected flows ($Ar_c = 13.2, J = 0.00126$).

to traverse the entire building length and this is clearly shown with the contours of temperature (fig. 10a) and airspeed (fig. 10b). The measured airspeeds shown in figure 10b clearly indicate the path of the jet as it separates ($X/L = 0.72$) and falls to the floor ($X/L = 0.81$). Additionally, the measured isotherms (fig. 10b) indicate the jet path especially with the 29.0°C isotherm.

Y-Z PLANE (CROSS-STREAM) TEMPERATURE DISTRIBUTIONS

Figures 11 to 13 compare the predicted and measured Y-Z plane contour and surface plots of temperature, and the predicted airflow patterns for moderate-to-highly buoyancy affected flows. The most fascinating aspect of this research was the pronounced effect that buoyant forces had on the overall distributions of airspeed and temperature. To exemplify this “cross-stream” behavior, the Y-Z plane ($x = 0.44$ m) is shown with contour and surface temperature plots.

Figure 11 presents the predicted (a) and measured (b) cross-stream isotherms at $X/L = 0.18$ for treatment two. The cross-stream behavior is best exemplified with temperature surface plots as shown in figures 12a and 12b. Figure 12a is the near-floor ($Y/H = 0.05$) temperature distribution for treatment one. The depression in temperature at $X/L = 0.18$ corresponding to the location where the inlet jet is falling (fig. 5a) is quite noticeable. A subtle note is the increased temperature near the symmetry plane at $X/L = 0.18$. Figure 12b, corresponding to treatment two ($Ar_c = 45.5, J = 0.00053$), indicates this same general trend. The inlet jet was predicted (fig. 6a) and observed (fig. 6b) to fall at $X/L = 0.48$ and this is the location where the temperature depression exists (fig. 12b).

Figure 13 presents three representative predicted cross-stream airflow patterns for treatment one ($Ar_c = 88.5, J = 0.00029$). Figure 13a presents typical cross-stream behavior for Y-Z planes located at or very near the $X/L$ location where the jet is dropping. As indicated in figure 13a, the falling inlet jet is deflected towards the outer side-wall. Figures 13b and 13c present typical cross-stream behavior for Y-Z planes located away from the falling jet region. The airflow behavior indicates that fluid rises at the symmetry plane, impinges at the top of the chamber, traverses along the ceiling towards the outer side-wall, and downward at the outer side-wall to the floor. This was predicted for each treatment analyzed and was verified with cross-stream flow visualization.

FALING JET AND BUOYANT FORCE INTERACTIONS—THREE-DIMENSIONAL FLOW

The cross-stream “buoyant plume” behavior near the symmetry plane shows the effect of animal-generated
buoyant forces on the distributions of airspeed and temperature for moderate-to-highly buoyancy affected flows ($\text{Ar}_c > 40$). This effect can be interpreted as a superposition of two, assumed independent two-dimensional processes as follows: The heated floor (simulating animals) provided a predominately two-dimensional buoyant plume where fluid rising near the symmetry plane creating a cross-stream recirculation zone. The airflow produced by the buoyant plume tends to raise the temperature near the symmetry plane and reduces it near the outer side-wall. Concurrently, a predominately two-dimensional airflow pattern is established in the X-Y plane as a result of the inlet jet. These two assumed independent, two-dimensional effects (one buoyant and one inertial) interact at the location where the inlet jet falls to the floor. It is conceived then, that the force (downward
inertial or upward buoyant) which is greatest will dominate the resulting three-dimensional flow behavior. It is clear, however, that for winter-time ventilation arrangements, the upward buoyant force appears to govern the three-dimensional behavior. With respect to the downward inertial force, the upward buoyant force causes the falling jet to "slide off" the buoyant plume towards the outer side-wall. This effect was shown from the near-floor temperature surface plots presented in figures 12a and 12b.

The interaction of the falling jet and buoyant forces resulted in a three-dimensional airflow pattern throughout the ventilated chamber. The overall trend is a helical (spiraling) airflow distribution pattern from the inlet to the exhaust end-wall. In fact, given the symmetry of this
The interactions between the inertial forces of the inlet jet and the animal-driven buoyant forces resulted in three-dimensional airflow patterns for moderate and highly buoyancy affected flows. To investigate the thermal environment at animal-level, the numerically predicted distributions of airspeed and temperature were combined to indicate the environment. The index incorporated was the modified ambient temperature (MAT) described in Hoff et al. (1987). This index modifies the air temperature as a function of airspeed to give an approximate indicator of the thermal demand of the environment on the animal.

Figures 14a and 14b present the results of this exercise. Figures 14a and 14b represent the near-floor (y = 0.04 m) MAT contours and surface plots for treatments one and two respectively. As a result of the interaction between the inlet jet and the animal-driven buoyant forces, the chilled inlet jet was predicted to deflect towards the outer side-wall. The contour and surface plots shown in figures 14a and 14b indicate this trend. As shown in figure 14a, the near-floor location demanding the largest energy from the animal is predicted to be located where the jet falls (X/L = 0.17) and near the outer side-wall. This trend was also indicated in figure 14b (treatment two) except at X/L = 0.48 corresponding to the falling jet location. This agrees with the overall trend that the inlet jet is deflected towards the outer side-wall as it falls to the floor.

**FURTHER WORK**

There is a great deal yet to learn about the subtle behavior of air motion in livestock confinement facilities. Future research should focus on the refinement and further verification of models that describe air motion. Once accomplished, the nature and fate of contaminants, dust, moisture, and bacterial colonies as related to fresh-air distribution can be investigated.

Additionally, modeling techniques should be pursued that capture the main features of turbulent, three-dimensional behavior without the computational overhead currently present with the model used for this research project. Modeling provides the research engineer with an unmatched investigative tool, one that should not be overlooked.

The extent and nature of three-dimensional distributions of airspeed and temperature need further work. The results presented here indicate a substantial three-dimensional behavior at moderate and highly buoyancy affected flows. To the extent that this trend persists in full-scale production facilities needs further analysis.

The results presented here are based on a nice, uniform distribution of animals at floor level. Work needs to be done on the localized buoyant effect of a non-uniform distribution of animals. It is envisioned that localized plumes of natural convection patterns will develop that could greatly effect the overall distribution of fresh air, especially during minimum, winter-time ventilation arrangements.

Finally, the cross-stream recirculation patterns found due to Benard Convection indicate a possible weakness in using the Ar as the non-isothermal jet stability parameter. Benard Convection is highly dependent upon the height and width of the building being ventilated. As the height-to-width ratio decreases, the cross-stream recirculation patterns increase in intensity and numbers. Thus, it is conceivable that one could ventilate two buildings with the same Ar and have substantially different airflows provided one of the buildings had a lower height-to-width ratio. Future research should investigate the effect of building size on non-isothermal jet stability and the ultimate effect on mass, momentum and energy distributions in livestock facilities.
CONCLUSIONS

The effects of animal-generated buoyancy forces on the distributions of airspeed and temperature for winter-time ventilation of a swine growing/finishing facility were investigated. Both numerical and experimental investigations were conducted using a 1/5 scale-model facility. Numerically, the problem was analyzed with a high-level turbulence model that predicts turbulent airflow behavior, if it in fact exists. From this study, a number of conclusions can be drawn:

- For ventilation design and analysis purposes, the LBLR model adequately predicted airspeed and temperature trends for all treatment levels investigated.
- A significant cross-stream airflow pattern developed which affected the overall airspeed and temperature distributions. This cross-stream behavior was attributed to Benard convection. The cross-stream behavior resulted in a double-helix airflow pattern from inlet to exhaust.
- Three-dimensional effects were important contributors to the airspeed and temperature distributions especially for moderate and highly buoyancy affected flows.
Airflow patterns were adequately predicted for $\text{Ar}_e > 40$ and $J$ values < 0.00053. For $\text{Ar}_e < 40$ and $J$ values > 0.00053, the visualized patterns indicated that the jet separated from the ceiling before the opposing end-wall. This discrepancy was attributed to variations in the experimental and numerical inlet flow development assumptions.

REFERENCES


Kuehn, T. H. 1990. Personal communications based on past numerical modeling experiences. Mechanical Engineering Department, University of Minnesota, Minneapolis.


——. 1986. The use of archimedes number in the design of ventilation systems for animal housing. Conf. on Agricultural Engineering, Adelaide. 24-28 August.


NOMENCLATURE
$\text{Ar}_e =$ inlet corrected Archimedes Number (dimensionless)
$\mu =$ constant used in LBLR model (= 0.0165)
$\nu =$ constant used in LBLR model (= 20.5)
$\chi =$ constant used in LBLR model (= 0.05)
$A_s =$ actual inlet slot area ($m^2$)
$c_v =$ constant used for turbulent viscosity (= 0.09)
\( c_1 \) = constant used in k-equation (= 1.44)
\( c_2 \) = constant used in e-equation (= 1.92)
\( c_p \) = specific heat (J/kg·C)
\( c_d \) = coefficient of discharge (= 0.60)
\( F_b \) = vertical buoyant force (upward positive)
\( F_{\mu} \) = LBLR damping function for turbulent viscosity
\( f_1 \) = viscous dissipation auxiliary relation
\( f_2 \) = viscous dissipation auxiliary relation
\( g \) = gravitational constant (9.81 m/s²)
\( G_b \) = turbulent production due to buoyancy
\( H \) = building height (= 0.64 m)
\( J \) = Jet Momentum Number (dimensionless)
\( k \) = turbulent kinetic energy (m²/s²)
\( L \) = length scale of turbulence (m)
\( l \) = Prandtl mixing length (m)
\( p \) = calculated pressure (Pa)
\( p \) = static pressure (Pa)
\( Q \) = inlet ventilation rate (m³/s)
\( R_k \) = turbulent Reynold’s Number (dimensionless)
\( R_t \) = turbulent Reynold’s Number (dimensionless)
\( R_{AH} \) = Raleigh Number based on building height (dimensionless)
\( S_\phi \) = generalized source term (per unit volume-time)
\( T \) = temperature (C)
\( T_i \) = inlet air temperature (C)
\( T_s \) = simulated animal surface temperature (C)
\( T_t \) = chamber top temperature (C)
\( u_0 \) = average inlet velocity (m/s)
\( V \) = building volume (= 3.07 m³)
\( V_t \) = generalized velocity scale of turbulence (m/s)
\( W \) = building (and inlet slot) width (= 2.0 m)
\( y^+ \) = dimensionless distance from a solid boundary
\( y_p \) = generalized normal distance from a solid boundary (m)
\( u, v, w \) = velocity components in x, y, and z direction (m/s)
\( x, y, z \) = coordinate directions (m)

**GREEK SYMBOLS**

\( \rho \) = density (kg/m³)
\( \varepsilon \) = viscous dissipation of turbulent energy (m²/s³)
\( \delta_{ij} \) = kronecker delta (dimensionless)
\( \mu_1 \) = laminar viscosity (kg/m·s)
\( \mu_t \) = turbulent viscosity (kg/m·s)
\( \mu_{eff} \) = effective viscosity (= \( \mu_1 + \mu_t \)) (kg/m·s)
\( \nu \) = kinematic viscosity (m²/s)
\( \Gamma_\phi \) = generalized diffusion coefficient (per m·s)
\( \sigma \) = Prandtl or Schmidt Number (dimensionless)
\( \beta \) = thermal coefficient of volumetric expansion (1/K)
\( \kappa \) = Von Karman’s constant (= 0.40)

**SUBSCRIPTS**

- **actual** = property evaluated using actual environment
- **ave** = property evaluated using average spatial conditions
- **eff** = effective property (laminar plus turbulent)
- **i, j, k** = cartersian-tensor notation (1 = x, 2 = y, and 3 = z)
- **l** = laminar component
- **ref** = property evaluated using reference (inlet) conditions
- **t** = turbulent component
- **∞** = property evaluated at the free-stream conditions

**SUPERSCRIPTS**

- / = denotes fluctuating component
- \(-\) = denotes average component