EVALUATION OF THE INDOOR AIR VELOCITY OF A SIDEWALL INLET AND ROOF EXHAUST VENTILATED BROILER SHED USING COMPUTATIONAL FLUID DYNAMICS

*1Jongbo, A. O., 2Atta, A. T. and 3Moorcroft, I.

1 Department of Agricultural and Environmental Engineering, Federal University of Technology, Akure, Nigeria

2Institute of Agricultural Research and Training, Obafemi Awolowo University, Moor Plantation, Ibadan, Nigeria

3Engineering Department, Harper Adams University, Newport, TF10 8NB, United Kingdom.

ARTICLE HISTORY
Received: 02-06-21
Accepted: 15-07-21

KEYWORDS:
Airflow distribution; hot weather condition; simulation; inlet opening size; Climate control

Abstract
Fast-growing broiler chickens, bred for meat, find it difficult to adapt to warm conditions during hot weather periods in an enclosed environment. They tend to change their behavioural and physiological mechanisms to survive. This study was carried out to evaluate the air velocity distributions within a sidewall inlet and roof exhaust ventilated broiler shed using the computational fluid dynamics (CFD). The simulation was conducted using three turbulence models (standard k-ε, realisable k-ε, and SST k-ω) to determine the best predictive model for the hot weather ventilation of the broiler shed under consideration. The results predicted by the turbulence models were validated with the field experimental results. It was discovered that standard k-ε turbulence model predicted air velocity distributions, close to that of the air velocity distributions obtained during the experimental study except at the centre of the broiler shed where the CFD predicted higher air velocity. This shows that CFD could be adopted by Agricultural Engineers to create appropriate environments for animals before the structures are physically erected.

Correspondence: aojongbo@futa.edu.ng; aojongbo@gmail.com

1. Introduction
To provide a healthy animal environment, air flow must reach the animal occupied zones (Li et al., 2017; Bustamante et al., 2013). The sustainability of suitable climatic condition in the animal shed depends on the performances of the ventilation system (Norton et al., 2007). It is a difficult task to accurately quantify the indoor environmental conditions within the animal shed because of the complexity of the indoor conditions (Norton et al., 2007). However, significant progress has been achieved as to how computational fluid dynamics (CFD) could be used to quantify and analyse the indoor and outdoor environmental parameters. CFD has been used for simulating fluid flow in the agricultural facilities such as
2. MATERIALS AND METHODS

2.1. Experimental shed and measurements

The experimental work was carried out in a broiler shed (Plate 1), located at Harper Adams University, the United Kingdom within latitude and longitude 52.7795° N and 2.4271° W. The shed is 21.75 m long, 18.30 m wide, 2.36 m eave height and the inlets are 1.60 m above the floor. There are 24 bottom-hinged sidewall inlets (12 inlets on each sidewall) in the shed with dimension 0.52 m by 0.21 m and five roof fans of diameter 0.63 m which were all mechanically controlled by a CLIMATEC environmental control system.

Objectives of this study were, to (1) simulate the air velocity in the animal occupied zones inside the sidewall inlet and roof exhaust ventilated broiler shed; (2) determine suitable turbulence model for evaluating the indoor conditions of the broiler shed and; (3) validate the CFD simulation with the field experiments conducted in a broiler shed.
2.2. CFD modelling

2.2.1. Empty broiler shed

Figure 1 shows the geometries of broiler sheds with different inlet configurations and indoor obstacles. The geometries were developed with SolidWorks 2016. Figure 1a represents the broiler shed used in this study with twenty-four 0.52 m by 0.21 m sidewall inlets. Figure 1b shows a cross-section of the broiler shed with feeder and drinker lines raised to the height of 0.3 m above the floor with their locations from the sidewall as provided in the experimental broiler shed.

**Figure 1:** Broiler shed with (a) twenty-four sidewall inlets and (b) drinker and feeder. All dimensions are in metres.

2.2.2 Mesh generation

The geometries of the broiler shed (Figure 1) were imported into the CFD software package (Star CCM+ 12) for simulations. The internal parts of the geometry, which represents the computational domain, were extracted for surface and volume discretization.

For each of the shed’s geometries, three unstructured coarse mesh densities were generated to discretize the computational domain. For this study, the mesh with the highest meshing density (1,050,996) was selected and used in all the subsequent CFD simulations to ensure that CFD predictions were precise and accurate for all the turbulence models.
The standard $k-\varepsilon$ model is a semi-empirical turbulence model that is based on the transport equations for the turbulent kinetic energy ($k$) and rate of dissipation ($\varepsilon$) (Norton, 2010; Norton et al., 2007; Norton and Sun, 2009). Though it has wide industrial application, it is limited by its assumption that the turbulent energy, generated by the large eddies is equally distributed in the energy spectrum and the model requires wall function due to its inability to determine dissipation rate at the wall (ANSYS FLUENT, 2006). The model is generally represented by:

Turbulent kinetic energy ($k$)

$$\frac{\partial (\rho k)}{\partial t} + \sum_{i} \frac{\partial (\rho u_{i} k)}{\partial x_{i}} = \frac{\partial}{\partial x_{i}} \left( \frac{\partial k}{\partial x_{i}} \sigma_{k} \right) + 2\mu_{t} D_{ij} D_{ij} - \rho \varepsilon$$

(1)

Dissipation rate ($\varepsilon$)

$$\frac{\partial (\rho \varepsilon)}{\partial t} + \sum_{i} \frac{\partial (\rho u_{i} \varepsilon)}{\partial x_{i}} = \frac{\partial}{\partial x_{i}} \left( \frac{\partial \varepsilon}{\partial x_{i}} \sigma_{\varepsilon} \right) + C_{1\varepsilon} \frac{\varepsilon}{k} 2\mu_{t} D_{ij} D_{ij} - C_{2\varepsilon} \rho \frac{\varepsilon^{2}}{k}$$

(2)

where $u_{i}$ is the fluctuating component of velocity in $u_{i}$ direction; $\rho$ is the fluid density; the turbulent viscosity $\mu_{t} = \rho C_{\mu} \frac{k^{2}}{\varepsilon}$; $\sigma_{k} = 1.00, \sigma_{\varepsilon} = 1.30$ are Prandtl number for turbulent kinetic energy and dissipation rate equations respectively; $D_{ij}$ is the rate of deformation (strain rate); $C_{\mu}=0.09, C_{1\varepsilon}=1.44, C_{2\varepsilon}=1.92$ are turbulence model constants.

The realisable $k-\varepsilon$ model fulfils some mathematical constraints on the Reynolds stresses that are steady with the physics of turbulent flows such as positive standard Reynolds stress terms (Norton, 2010; Norton and Sun, 2009). In this model, the turbulence model constant $C_{i}$ is expressed as a function of mean flow and the properties turbulence and not continuous as in the instance of the standard $k-\varepsilon$ model (Norton, 2010; Norton & Sun, 2009). The model is represented by:

Turbulent kinetic energy

$$\frac{\partial (\rho k)}{\partial t} + \sum_{i} \frac{\partial (\rho u_{i} k)}{\partial x_{i}} = \frac{\partial}{\partial x_{i}} \left[ \frac{\partial k}{\partial x_{i}} \left( \mu + \frac{\mu_{t}}{\sigma_{k}} \right) \right] + P_{k} + P_{b} - \rho \varepsilon - Y_{M}$$

(3)

Dissipation rate

$$\frac{\partial (\rho \varepsilon)}{\partial t} + \sum_{i} \frac{\partial (\rho u_{i} \varepsilon)}{\partial x_{i}} = \frac{\partial}{\partial x_{i}} \left[ \frac{\partial \varepsilon}{\partial x_{i}} \left( \mu + \frac{\mu_{t}}{\sigma_{\varepsilon}} \right) \right] + \rho C_{1} D \varepsilon - \rho C_{2} \frac{\varepsilon^{2}}{k + \sqrt{\nu \varepsilon}}$$

(4)

where $C_{1} = max \left[ 0.43, \frac{\eta}{\eta + 5} \right]$; $\eta = D \frac{k}{\varepsilon}$; $D = 2D_{ij} D_{ij}$; $P_{k}, P_{b}$

are generated turbulent kinetic energies due to mean velocity gradient and buoyancy respectively; $C_{i}=1.90, \sigma_{i}=1.20$

The shear stress transport (SST) $k-\omega$ model is based on the transport equations for the turbulent kinetic energy ($k$) and the specific turbulent dissipation rate ($\omega$), which is the dissipation rate for every unit of turbulent kinetic energy (Norton, 2010; Norton & Sun, 2009).

Figure 2 shows the meshes of the broiler shed. During volume meshing, unstructured polyhedral grids were used to improve and optimise the overall quality of the cell surfaces and the volume mesh model. A prism layer mesh was used to generate prismatic cells near wall surfaces to improve the accuracy of the flow solution closer to all wall surfaces.
To verify the conditions under which indoor conditions of broiler shed were simulated, three turbulence models (standard (Std) $k-\varepsilon$, realisable (Re) $k-\varepsilon$, and shear stress transport (SST) $k-\omega$), widely adopted were selected and used (Li et al., 2016; Rong et al., 2016). They were selected to determine the most appropriate turbulence model for the indoor environment of a sidewall inlet and roof exhaust ventilated broiler shed.

For solution monitoring and convergence criteria, a global residual of 0.001 (0.1 %), for all fundamental equations was defined. The computations were not terminated until the residuals were lesser than 0.001 and the air velocity magnitudes in the broiler occupied zones were also stabilised. The air velocity magnitudes in the broiler occupied zones, where broiler chickens experience heat stress during hot weather periods, were only considered in this study.

2.2.3 Turbulence models, measurement planes and convergence criteria

To verify the conditions under which indoor conditions of broiler shed were simulated, three turbulence models (standard (Std) $k-\varepsilon$, realisable (Re) $k-\varepsilon$, and shear stress transport (SST) $k-\omega$), widely adopted were selected and used (Li et al., 2016; Rong et al., 2016). They were selected to determine the most appropriate turbulence model for the indoor environment of a sidewall inlet and roof exhaust ventilated broiler shed.

For solution monitoring and convergence criteria, a global residual of 0.001 (0.1 %), for all fundamental equations was defined. The computations were not terminated until the residuals were lesser than 0.001 and the air velocity magnitudes in the broiler occupied zones were also stabilised. The air velocity magnitudes in the broiler occupied zones, where broiler chickens experience heat stress during hot weather periods, were only considered in this study.

2.2.4 Boundary conditions

The boundary conditions specified in this study are shown in Table 1. These include the air velocity at the inlet, pressure at the outlet and shed wall surfaces. Air turbulence intensity of 0.10 was imposed at the inlet. The same turbulence intensity was obtained during the field
2.3 Occupied broiler shed

The experimental broiler shed, occupied with broiler models, was developed with SolidWorks 2016 (Figure 3) and imported into the Star CCM+ 12 for turbulence modelling. For CFD simulation purpose, the shed was filled with broiler models of a characteristic dimension of 0.18 m, similar to the size of the broiler models used during the field experiment (Jongbo et al., 2020). The fluid domain (internal part of the shed) was extracted and the surface and volume meshing were performed (Figure 4).

Table 1: Boundary conditions specifications

<table>
<thead>
<tr>
<th>Shed surfaces</th>
<th>Boundary conditions</th>
</tr>
</thead>
<tbody>
<tr>
<td>Inlet</td>
<td>Velocity inlet</td>
</tr>
<tr>
<td></td>
<td>Air velocity of $4.91 \text{ m s}^{-1}$ at the inlet</td>
</tr>
<tr>
<td></td>
<td>Inlet turbulence intensity of 0.10 (10 %)</td>
</tr>
<tr>
<td>Outlet</td>
<td>Pressure outlet</td>
</tr>
<tr>
<td></td>
<td>Pressure (0 Pa)</td>
</tr>
<tr>
<td>Shed walls, floor and roof</td>
<td>No-slip and smooth wall</td>
</tr>
</tbody>
</table>

Experimentation at the inlet of the broiler shed.

For CFD simulation purpose, the shed was filled with broiler models of a characteristic dimension of 0.18 m, similar to the size of the broiler models used during the field experiment (Jongbo et al., 2020). The fluid domain (internal part of the shed) was extracted and the surface and volume meshing were performed (Figure 4).

Figure 3: Experimental broiler shed occupied with broiler models.

All measurements are in metres.
In this section of the study, three meshing density refinements were performed based on the number of cells in the prism layers. The mesh densities were used for the simulations and were validated with the field experimental results to determine the appropriate volume mesh that would predict the airflow distributions in the broiler occupied zones using standard $k-\varepsilon$ turbulence model. Only standard $k-\varepsilon$ turbulence model was used for the occupied broiler shed due to its predictive capability in the simulated empty broiler shed. Air velocity magnitudes in the broiler occupied zones were obtained on measurement plane situated at 7.3 m from the front door.

2.4 Field experimentation set up

The field experiment comprised two studies. The first study was carried out in an empty experimental broiler shed while the second study was carried out in the broiler shed occupied with broiler models to represent live broiler chickens. The details of the field experiments have been reported by Jongbo et al. (2020).

3. RESULTS AND DISCUSSION

3.1 Validation of air velocity in the empty broiler shed

The air velocity predictions of the CFD simulation conducted with the broiler shed, having indoor obstacles such as feeding and drinking lines were validated using the field experiment results. Figure 5 shows the results of the validation of the turbulence models with the field experimental results.

As shown in Figure 5, the CFD turbulence model that predicted close air velocity to that
Figure 5: The validation of turbulence models [Std k-ε (red line); Re k – ε (green line); SST k – ω (purple line)] predictions with field experiment (blue line).

A t-test was conducted with SAS JMP 14 to analyse the differences between the predictions of the standard k-ε turbulence model and the field experiment at the distances 1 and 5m from the sidewall. The result of the analysis indicates that at 1m from the sidewall, there was a significant difference (p = 0.045) between the standard k-ε turbulence model prediction and the field experiment. However, at distance 5m, there is no significant difference (p = 0.245) between the standard k-ε turbulence model prediction.

obtained during the field experiment, in the broiler occupied zones, was Standard k-ε (red line). Standard k-ε turbulence model has been indicated as a good predictive turbulence model in some studies (Bustamante et al., 2013; Norton et al., 2013). In the study conducted by Blanes-Vidal et al. (2008), it was reported that there were higher discrepancies between the CFD predictions and field experiment. In this study, the discrepancies between the Standard k – ε (red line) and the field experiment were ±0.10 ms⁻¹ at all measurement locations. The CFD predictions of Standard k – ε could therefore be considered very good based on its predictions in this study. In the study carried out by Norton (2010, p.53) in the naturally ventilated calf shed, he indicated that standard k – ε and realisable k – ε were good turbulence models for predicting indoor environment of livestock. However, in this study, realisable k-ε has not shown to be a good turbulence model for simulating sidewall inlet and roof exhaust ventilated broiler shed because realisable k – ε turbulence model predicted that air velocity at the sidewall (1m from the inlet) was 0.46 ms⁻¹, higher than that at 5 m away from the sidewall (0.36 ms⁻¹).

A t-test was conducted with SAS JMP 14 to analyse the differences between the predictions of the standard k-ε turbulence model and the field experiment at the distances 1 and 5m from the sidewall. The result of the analysis indicates that at 1m from the sidewall, there was a significant difference (p = 0.045) between the standard k-ε turbulence model prediction and the field experiment. However, at distance 5m, there is no significant difference (p = 0.245) between the standard k-ε turbulence model prediction.
and the field experiment. Previous studies (Blanes-Vidal et al., 2008; Mostafa et al., 2012) have also shown that turbulence model may not predict the exact results as they were obtained during the field experiment. However, the prediction of CFD, though not exactly as a field experiment, could roughly follow a similar pattern of the field experiment.

3.2 Validation of air velocity in the broiler shed occupied with broiler models

For better understanding, the mesh density with 808,805 cells, was considered to be appropriate for further simulation processes. As shown in Figure 6, there was no significant difference (p < 0.05) between the prediction of standard \( k - \varepsilon \) and the field experiment at all locations except at the centre of the shed (9 m from the sidewall) where the standard \( k-\varepsilon \) turbulence model predicted higher air velocity in the broiler occupied zone. From the figure, it could be observed that the CFD predictions and that of the results of the field experiment followed similar pattern except at the centre of the building where CFD predicted higher air velocity. Similar to the previous study (Blanes-Vidal et al., 2008), though there could be slight discrepancy between the results of the field experiment and the CFD predictions, this study has shown that CFD could be used as an engineering tool to predict an estimation of indoor conditions of sidewall inlet and roof exhaust ventilated broiler shed which could direct further field experiments.

4 CONCLUSION

Airflow predictions of the CFD in the broiler occupied zones, when the broiler shed was empty, was validated with the field experiment.
The results of this study have shown that an estimation of the indoor air velocity of the sidewall inlet and roof exhaust ventilated broiler shed could be predicted by using CFD simulation and that standard $k-\varepsilon$ turbulence model showed better results compared to other turbulence models (realisable $k-\varepsilon$ and SST $k-\omega$) considered in this study. However, to simulate airflow in an empty broiler shed, it is advisable to increase the mesh density during the surface and volume discretisation to improve the prediction capacity of the CFD modelling.

Further study with an occupied “model” broiler shed was conducted to evaluate the impact of the current inlet opening technique used during the hot weather conditions by the commercial poultry farmers. The standard $k-\varepsilon$ turbulence model performed well in predicting the airflow distributions in the occupied zones of broiler chickens when validated with the results of the field experiment. However, comparing the mesh densities of occupied broiler shed with that of the empty broiler shed, this study has shown that the higher number of cells in the prism layer may be avoided due to the design complexity and longer computation time. With lesser design details of broiler shed occupied with broilers, the CFD could accurately predict the airflow distributions in the broiler occupied zones.

This study, in conjunction with the report findings of Albright (1990), has shown that the current method used in the sidewall inlet and roof exhaust ventilated broiler shed needs to be re-evaluated. It has clearly shown that the method may not provide better airflow in the broiler occupied zones where higher air movement is needed during hot weather periods. Therefore, this study suggests that ventilation engineers need to investigate other appropriate hot weather ventilation system for broiler production to alleviate the heat stress challenges faced by broiler chickens during hot weather periods.


Norton, T. (2010). *Using computational fluid dynamics to design naturally ventilated calf buildings that promote animal health and welfare*. University College Dublin, National University of Ireland, Dublin.


