Journal of Engineering and Engineering Technology

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

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

[1] Department of Agricultural and Environmental Engineering,
Federal University of Technology, Akure, Nigeria
[2]Institute of Agricultural Research and Training, Obafemi Awolowo
University, Moor Plantation, Ibadan, Nigeria
[3]Engineering Department, Harper Adams University, Newport,
TF10 8NB, United Kingdom.

ABSTRACT

Fast-growing broiler chickens, bred for meat, find it difficult to adapt to warm condition 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

greenhouses (Benni *et al.*, 2016), broiler sheds (Bustamante *et al*., 2013), cattle sheds (Norton *et al*., 2010) and pig houses (Li *et al*., 2017). Notwithstanding, the information on its application in the sidewall inlets and roof exhaust ventilated broiler shed, during hot weather periods, is hardly available. As indicated by Albright (1990) poultry keepers generally open the sidewall inlets 100 % to increase the air movement into the broiler shed with the intention of increasing air movement in the animal occupied zones during hot summer times. For the proper understanding of the effect of the 100 % opening of inlets on air velocity distribution in the animal occupied zones, it is vital to numerically simulate the indoor conditions using CFD. This would enable the ventilation engineers to provide appropriate hot weather ventilation system for sidewall inlet and roof exhaust ventilated broiler sheds.

Therefore, this study was carried out to evaluate the effect of 100 % opening of sidewall inlets on the air velocity distribution in the animal occupied zones using CFD. The 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. 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 º N and 2.4271 º 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.

Plate 1: A sidewall inlet and roof exhaust ventilated 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.
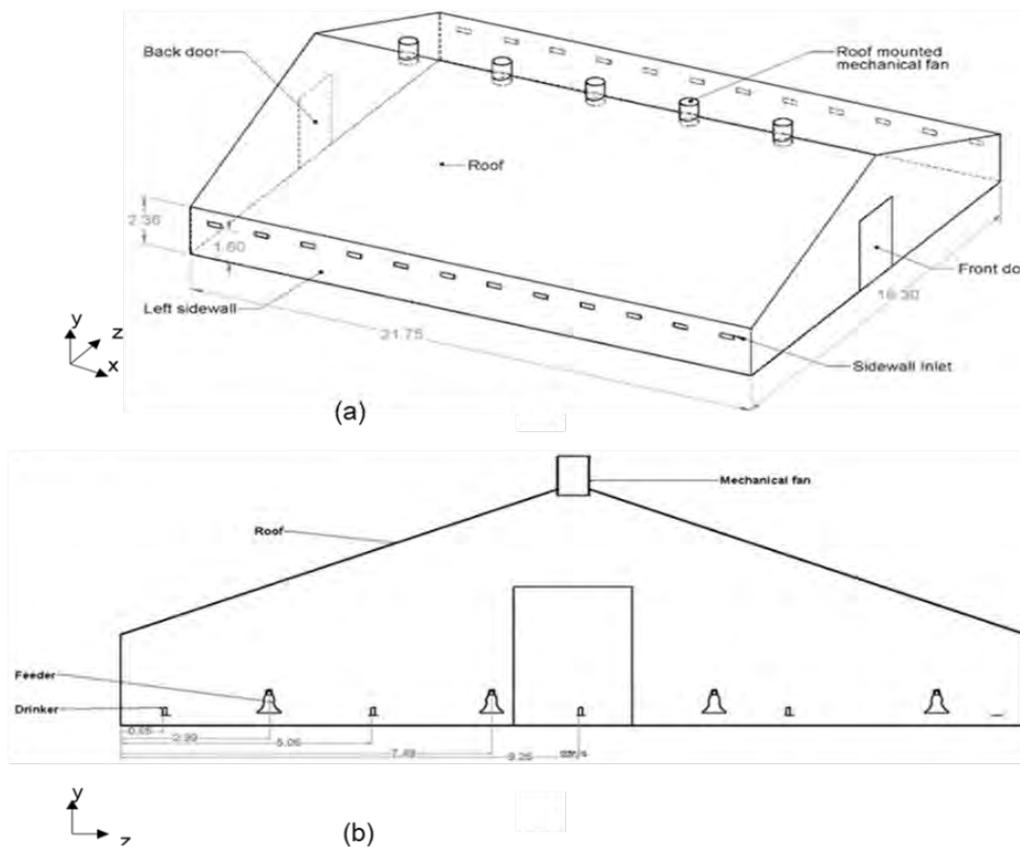

Figure 1: Broiler shed with (a) twenty-four sidewall inlets and (b) drinker and feeder. All dimensions are in metres.

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 (ε) (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)

$$\frac{\partial(\rho k)}{\partial t} + \frac{\partial(\rho u_i k)}{\partial x_i} = \frac{\partial}{\partial x_i}\left(\frac{\partial k}{\partial x_i}.\frac{\mu_t}{\sigma_k}\right) + 2\mu_t D_{ij} D_{ij} - \rho\varepsilon \qquad (1)$$

Dissipation rate (ε)

$$\frac{\partial(\rho\varepsilon)}{\partial t} + \frac{\partial(\rho u_i\varepsilon)}{\partial x_i} = \frac{\partial}{\partial x_i}\left(\frac{\partial\varepsilon}{\partial x_i}.\frac{\mu_t}{\sigma_\varepsilon}\right) + C_{1\varepsilon}\frac{\varepsilon}{k}2\mu_t D_{ij} D_{ij} - C_{2\varepsilon}\rho\frac{\varepsilon^2}{k} \qquad (2)$$

where u_i is the fluctuating component of velocity in u_i direction; ρ 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_μ=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_μ is expressed as a function of mean flow and the properties turbulence and not continuous as in the instance of the standard k-ε model (Norton, 2010; Norton & Sun, 2009). The model is represented by; Turbulent kinetic energy (k)

$$\frac{\partial(\rho k)}{\partial t} + \frac{\partial(\rho u_i k)}{\partial x_i} = \frac{\partial}{\partial x_i}\left[\frac{\partial k}{\partial x_i}\left(\mu + \frac{\mu_t}{\sigma_k}\right)\right] + P_k + P_b - \rho\varepsilon - Y_M \qquad (3)$$

Dissipation rate (ε)

$$\frac{\partial(\rho\varepsilon)}{\partial t} + \frac{\partial(\rho u_i\varepsilon)}{\partial x_i} = \frac{\partial}{\partial x_i}\left[\frac{\partial k}{\partial x_i}\left(\mu + \frac{\mu_t}{\sigma_\varepsilon}\right)\right] + \rho C_1 D\varepsilon - \rho C_2 \frac{\varepsilon^2}{k + \sqrt{\nu\varepsilon}} \qquad (4)$$

where $C_1 = max\left[0.43, \frac{\eta}{\eta+5}\right]$;

$\eta = D\frac{k}{\varepsilon}$; $D = \sqrt{2D_{ij}D_{ij}}$; P_k, P_b

are generated turbulent kinetic energies due to mean velocity gradient and buoyancy respectively; C_2 = 1.90, σ_ε = 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 (ω), 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.
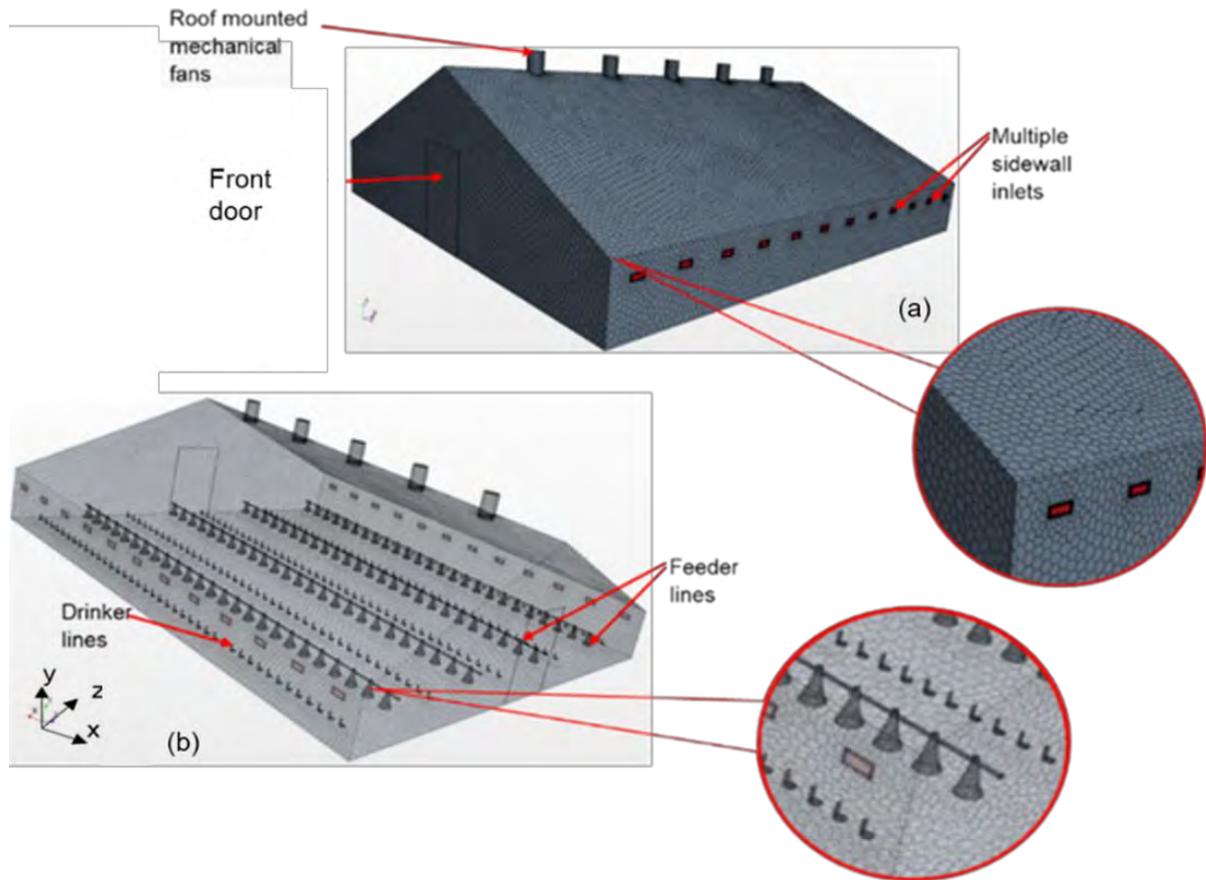
Figure 2: Polyhedral meshes of the broiler shed with; (a) twenty-four bottom-hinged inlets and (b) internal obstacles.

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–ω), 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

Table 1: Boundary conditions specifications

Shed surfaces	Boundary conditions
Inlet	Velocity inlet
	Air velocity of 4.91 m s^{-1} at the inlet
	Inlet turbulence intensity of 0.10 (10 %)
Outlet	Pressure outlet
	Pressure (0 Pa)
Shed walls, floor and roof	No-slip and smooth wall

experimentation at the inlet of the broiler shed.

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).
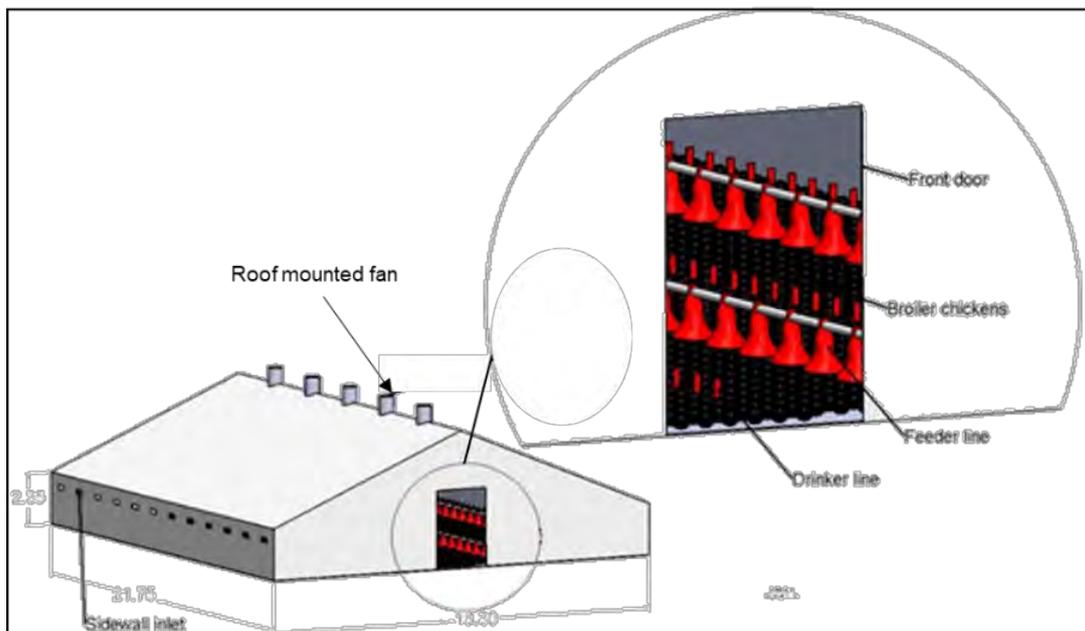

*Figure 3: Experimental broiler shed occupied with broiler models.
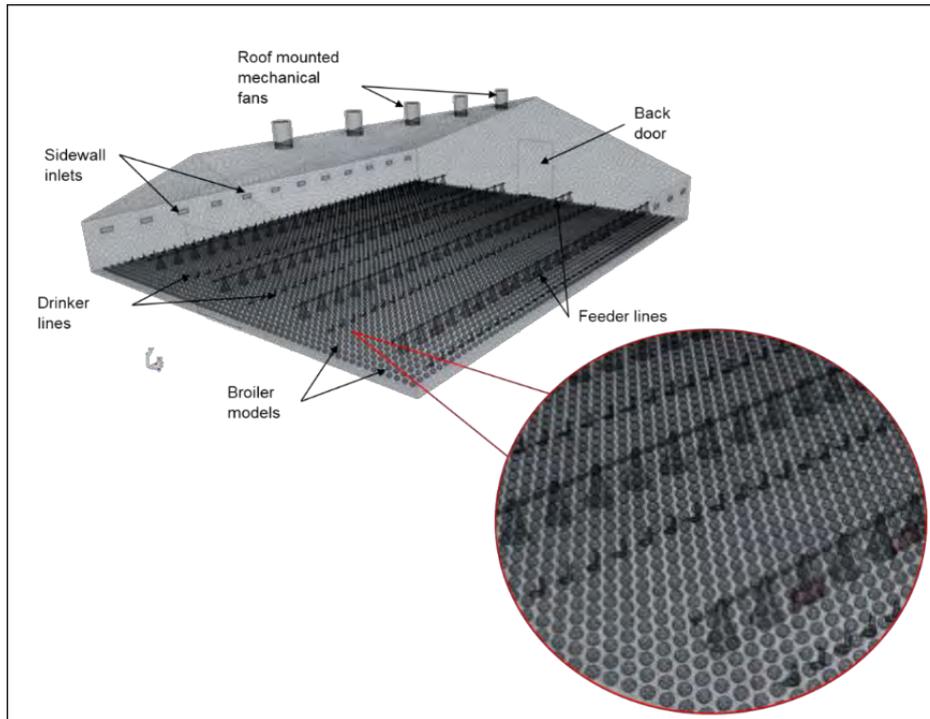All measurements are in metres.*

Figure 4: Surface and volume meshes of the broiler shed

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
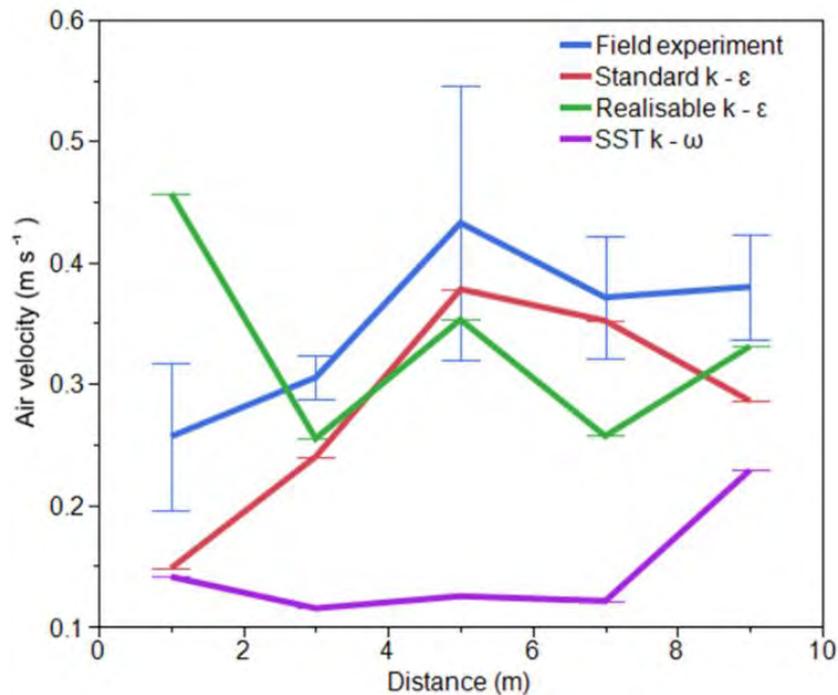
Figure 5: The validation of turbulence models [Std k-ε (red line); Re $k-\varepsilon$ (green line); SST $k-\omega$ (purple line)] predictions with field experiment (blue line).

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 $k-\varepsilon$ (red line) and the field experiment were ±0.10 ms⁻¹ at all measurement locations. The CFD predictions of Standard $k-\varepsilon$ 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 $k-\varepsilon$ and realisable $k-\varepsilon$ 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 $k-\varepsilon$ 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-ε 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
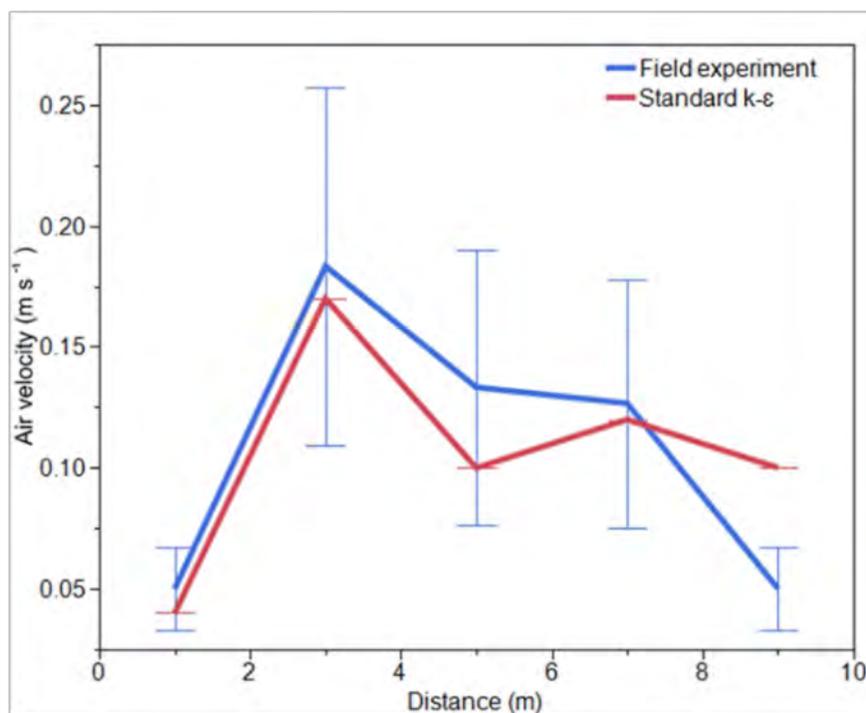

Figure 6: Validation of air velocity predictions of standard k-ε turbulence model (red line) with the field experiment (blue line).

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–ε turbulence model showed better results compared to other turbulence models (realisable k-ε 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-ε 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.

Albright, L. D. (1990). *Environment control for animals and plants*. American Society of Agricultural Engineers, St. Joseph, Michigan.

ANSYS FLUENT. (2006). Modeling turbulent flows: introductory FLUENT training. *ANSYS, Inc. Proprietary*.

Benni, S., Tassinari, P., Bonora, F., Barbaresi, A., & Torreggiani, D. (2016). Efficacy of greenhouse natural ventilation: Environmental monitoring and CFD simulations of a study case. *Energy and Buildings*, *125*, 276–286. https://doi.org/10.1016/j.enbuild.2016.05.014

Blanes-Vidal, V., Guijarro, E., Balasch, S., & Torres, A. G. (2008). Application of computational fluid dynamics to the prediction of airflow in a mechanically ventilated commercial poultry building. *Biosystems Engineering*, *100*(1), 105–116. https://doi.org/10.1016/j.biosystemseng.2008.02.004

Bustamante, E., García-Diego, F. J., Calvet, S., Estellés, F., Beltrán, P., Hospitaler, A., & Torres, A. G. (2013). Exploring ventilation efficiency in poultry buildings: The validation of computational fluid dynamics (CFD) in a cross-mechanically ventilated broiler farm. *Energies*, *6*(5), 2605–2623. https://doi.org/10.3390/en6052605

Jongbo, A. O., Moorcroft, I., White, D., Norton, T., & Okunola, A. A. (2020). Evaluation of airflow movement within a broiler shed with roof ventilation system during summer. *IOP Conference Series: Earth and Environmental Science*, *4 4 5* (0 1 2 0 2 8) . https://doi.org/10.1088/1755-1315/445/1/012028

Li, H., Rong, L., & Zhang, G. (2016). Study on convective heat transfer from pig models by CFD in a virtual wind tunnel. *Computers and Electronics in Agriculture*, *123*(April), 203–210. https://doi.org/10.1016/j.compag.2016.02.027

Li, H., Rong, L., & Zhang, G. (2017). Reliability of turbulence models and mesh types for CFD simulations of a mechanically ventilated pig house containing animals. *Biosystems Engineering*, *161*, 37–52. https://doi.org/10.1016/j.biosystemseng.2017.06.012

Mostafa, E., Lee, I. B., Song, S. H., Kwon, K. S., Seo, I. H., Hong, S. W., … Han, H. T. (2012). Computational fluid dynamics simulation of air temperature distribution inside broiler building fitted with duct ventilation system. *Biosystems Engineering*, *112*(4), 293–303. https://doi.org/10.1016/j.biosystemseng.2012.05.001

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.

Norton, T., Grant, J., Fallon, R., & Sun, D.- W. (2009). Assessing the ventilation effectiveness of naturally ventilated livestock buildings under wind dominated conditions using computational fluid dynamics. *Biosystems Engineering*, *103*(1), 78–99. https://doi.org/10.1016/j.biosystemseng.2009.02.007

Norton, T., Grant, J., Fallon, R., & Sun, D.- W. (2010). Assessing the ventilation performance of a naturally ventilated livestock building with different eave opening conditions. *Computers and Electronics in Agriculture*, *71*(1), 7–21. https://doi.org/10.1016/j.compag.2009.11.003

Norton, T., Kettlewell, P., & Mitchell, M. (2013). A computational analysis of a fully-stocked dual-mode ventilated livestock vehicle during ferry transportation. *Computers and Electronics in Agriculture*, *93*, 217–228. https://doi.org/10.1016/j.compag.2013.02.005

Norton, T., & Sun, D.-W. (2009). Computational fluid dynamics in thermal processing. In R. Simpson (Ed.), *Engineering aspects of thermal food* (pp. 317–363). New York: CRC Press- Taylor & Francis Group.

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. https://doi.org/10.1016/j.biortech.2006.11.025

Rong, L., Nielsen, P. V., Bjerg, B., & Zhang,

G. (2016). Summary of best guidelines and validation of CFD modeling in livestock buildings to ensure prediction quality. *Computers and Electronics in* *Agriculture, 121,* 180–190. https://doi.org/10.1016/j.compag.2015.12.005