STUDY OF CEILING DEFLECTOR APPLICATION IN A POULTRY HOUSE

Marcos Batistella Lopes*, Viviana Cocco Mariani, Kátia Cordeiro Mendonça
*Author for correspondence
Department of Mechanical Engineering
Pontifical Catholic University of Paraná
Imaculada Conceição, 1155, Curitiba, Brazil, 80215-901
E-mail: marcos.batistella@pucpr.edu.br

ABSTRACT

This present work proposes to apply the Computational Fluid Dynamics (CFD) by using Ansys® CFX version 14.5 to evaluate the impact of roof deflectors on the airflow patterns inside a tunnel ventilation poultry building located at Montenegro city (Brazil). Four different cases have been considered: roof with 24° of inclination with and without deflectors, with 6° of inclination and ceiling without inclination. For each case, it has generated three meshes (coarse, intermediate and fine) by using ICEM CFD 14.5 verifying mesh quality by aspect ratio and y^+, and the verification has done by comparing velocity and temperature profiles. It was used the k-ε standard turbulent model and Reynolds number at inlet is above 10^5, isothermal walls, null relative pressure at outlets and variation velocity at inlet according pressure drop. Results obtained show the higher pressure drop at roof with 24° of inclination with deflector.

INTRODUCTION

Probably the most consumed meat over the next five years will be chicken [1] because it is the cheapest and accessible source of meat in the world. Chicken meat is more environmental friend than other kind of meat [2] since chicken footprint is a half that of pork and a quarter that of beef. Moreover, to produce by grazing system 1 kg of chicken meat it is necessary average 3.5 m³ of water, while pork needs 4.5 m³ and beef 15 m³ [3].

Brazil is the largest exporter and the third largest producer, just after USA and China [4]. Thus, investing in research and technology in poultry industry is necessary to Brazil become a global reference in this area. The principal goal of research with broiler chickens is reduce slaughter age and loss of production, and increases the average weight, feed conversion and density of birds.

A good ventilation system must guarantee the thermal comfort of broilers (influenced by parameters such as temperature, humidity and air velocity) and remove toxic gases from broiler cycle life (NH_3, CO_2, CH_4 and H_2S). Natural ventilation cannot always achieve these needs. Moreover, climate changes require a better control over poultry buildings.

NOMENCLATURE

A_0	[m^2]	Inlet area
c_p	[Jkg^{-1}K^{-1}]	Specific heat at constant pressure
c_μ	[-]	Empirical constant
D	[m]	Fan diameter
F	[N]	Bulk force
H	[m]	Building height
$Hmax$	[m]	Distance of floor to the ridge
k	[m^{2}s^{-2}]	Turbulent kinetic energy
l	[m]	Characteristics dimension
L	[m]	Building length
p	[Pa]	Thermodynamic pressure
Pr_t	[-]	Turbulent Prandt number
Q	[m^{3}s^{-1}]	Volumetric flow
T	[K]	Thermodynamic temperature
U_n	[ms^{-1}]	Velocity component at n direction
u'	[ms^{-1}]	Velocity fluctuation at x direction
$V0$	[ms^{-1}]	Inlet normal velocity
v'	[ms^{-1}]	Velocity fluctuation at y direction
w'	[ms^{-1}]	Velocity fluctuation at z direction
W	[m]	Building width
y^+	[-]	Sub layer scaled distance to the wall
z,y,x	[m]	Cartesian coordinates

Special characters

α	[rad]	Inclination angle
Δp	[Pa]	Pressure drop
ε	[m^{2}s^{-3}]	Dissipation of turbulent kinematic energy
ρ	[kgm^{-3}]	Specific mass
ν	[m^{2}s^{-1}]	Kinematic viscosity
ν_t	[m^{2}s^{-1}]	Kinematic turbulent viscosity
τ_w	[Pa]	Shear stress
$\tau_{T,eff}$	[m^{2}s^{-1}]	Turbulent diffusion coefficient

Subscripts

max	Maximum
t	Turbulence
w	wall
0	Inlet

Norton *et al.* [5] have made a review about the application of Computational Fluid Dynamics (CFD) research on the agriculture industry. These authors have found three principal papers concerning poultry houses studies ([6], [7] and [8]). That means there were not many papers about this subject because computational capacity was limited and/or there are no interest in investigating the airflow in that kind of building.

Blanes-Vidal *et al.* [9] have validated CFD simulations of the airflow inside a Spanish commercial poultry building by comparing the velocity at some points of building with experimental results. The authors have used Fluent 6.0 to simulation with k-ε standard turbulence model and Gambit 1.3 to build 3D unstructured meshes with 217.000 nodes.

Seo *et al.* [10] have analyzed both quantitatively and qualitatively natural ventilated broiler houses in cold season. They have observed an optimal result of simulations with diffuser beneath the chimney inlet of aviary to mix warm and cold air. The authors have chosen the k-ε standard turbulence model in a simulation using Fluent 6.2 and Gambit 2.3 to build 3D meshes.

Saraz [11] has investigated the ammonia concentration in a Brazilian poultry house by validation numeric results using CFD tools with experimental data. Saraz *et al.* [12] have elaborated the state of art of the application of CFD in environment of broilers facilities and their limitations. The authors have found that there are a few number of CFD research applied to poultry buildings, the most part of them uses k-ε turbulence model, and the quadric meshes are more adequate to this kind of application.

Mostafa *et al.*[13] have investigated the broiler thermal comfort using CFD to analyze four ventilation systems to prevent cold air drafts during winter season. These authors have provided results of air temperature distribution and ammonia concentrations considering the ventilation on and off. Naturally, air temperature and ammonia concentration increase after turning off the ventilation system.

Bustamante *et al.*[14] have made an experimental validation of a CFD analysis by comparing velocities values at some regions of domain. They have showed the relative error of velocity was less than 7%.

Fawaz *et al.* [15] have investigated the performance of a solar-assisted localized heating and ventilation system for chicken brooding by using Ansys® CFX version 14.0. The authors have been constructed meshes with more the 10^6 elements and simulated than using k-ε standard turbulence model. Their outcomes have showed them systems saved 74% of the energy, however this systems cover 84% of the load required and the payback is more than 4 years.

In this context CFD techniques are useful to provide a good approximation of air flow within aviaries. Hence, the principal objective of this work is to analyze four cases by using Ansys® CFX version 14.5 and to compare velocities profiles.

MATHEMATICAL MODELING

The poultry barn employed in this study is located in Montenegro city (Brazil). The problem domain is presented in Fig 1 where air enters through the sidewalls and it is extracted out by fans, known as tunnel ventilation.

A simplified representation of the poultry building is presented at Fig. 2 where the values of distances are: L = 80 m, W = 12 m, H = 2.7 m, Hmax = 5.4 m and D = 1.3 m.

The cases that have been studied are presented in Fig. 3.

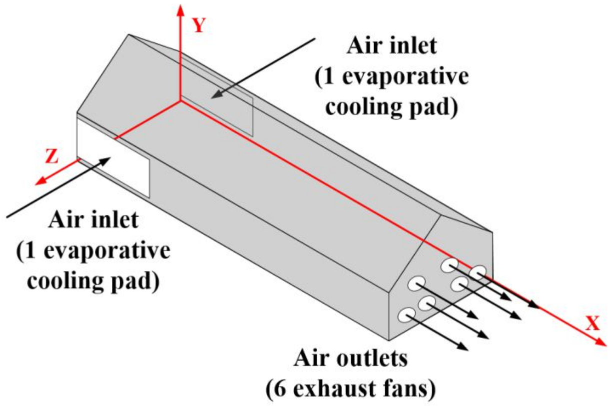
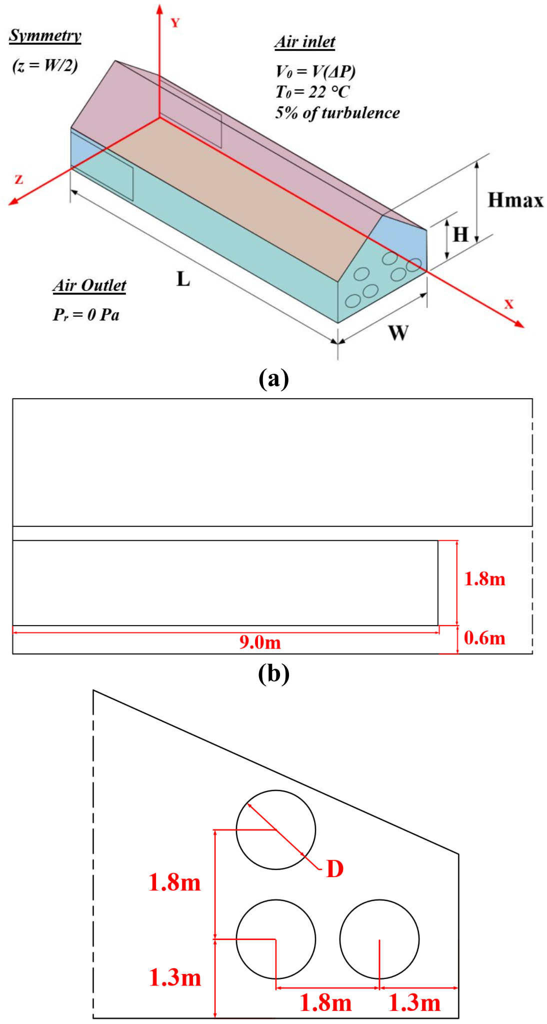


Figure 1 Tunnel ventilation.

Temperature (°C)	
Floor ($y = 0$)	25
Roof ($0 \le z \le W/2$ and $H \le y \le Hmax$)	25
Walls ($z = 0$; $x = 0$ and $x = L$)	30



(c)

Figure 2 Geometric information of **(a)** domain with boundary conditions, **(b)** inlet and **(c)** outlets regions.

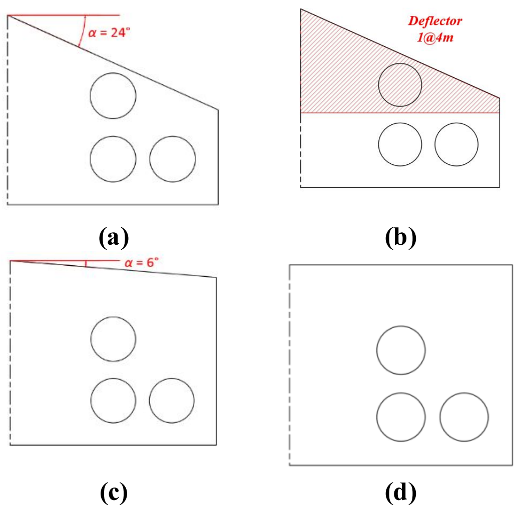


Figure 3 Variations of building **(a)** 24° inclination, **(b)** with deflector, **(c)** 6° inclination and **(d)** ceiling.

The hypotheses of this work are:

i. Steady state;
ii. Fluid: dry air (ideal gas);
iii. Reference pressure: 1 atm;
iv. Incompressible fluid: $\rho = 1.166$ kg/m³;
v. Sources of heat generation are negligible;
vi. Isothermal and smooth walls.

Thus, equations of mass conservation, Navier-Stokes and energy (applied in a differential volume and with Einstein notation) are, respectively:

$$\frac{\partial U_i}{\partial x_i} = 0\,, \tag{1}$$

$$\frac{\partial (U_i U_j)}{\partial x_j} = -\frac{1}{\rho}\frac{\partial P}{\partial x_i} + \frac{\partial}{\partial x_j}\left(\nu\frac{\partial U_i}{\partial x_j} - \overline{u_i u_j}\right) + F_i\,, \tag{2}$$

$$\frac{\partial (U_j T)}{\partial x_j} = \frac{\partial}{\partial x_j}\left(\tau_{T,eff}\frac{\partial T}{\partial x_j}\right), \tag{3}$$

where subscribes i and j means the components vectors of the three Cartesians positions (x, y and z), U_i and U_j are the components of the velocity vector (m/s), ρ is the density (kg/m³), P is the pressure (Pa), ν is the kinematic viscosity (m²/s), F_i are the bulk forces (N), T is the temperature (K) and $\tau_{T,eff}$ is the turbulent diffusion coefficient (m²/s), defined as $\tau_{T,eff} = \nu + \frac{\nu_t}{Pr_t}$. Pr_t is the turbulent Prandt number and ν_t is the kinematic turbulent viscosity, defined as $\nu_t = c_\mu \frac{k^2}{\varepsilon}$ (k-ε standard turbulent model); with k being the turbulent kinematic energy (m²/s²), ε is dissipation of turbulent kinetic energy (m²/s³) and c_μ.an empirical constant equals to 0,09. The term $\overline{u_i u_j}$ (m²/s²) of Eq. (2) is known as Reynolds tensor. The principal goal of turbulence models is to find this term.

In this study, the k-ε standard has been used. Proposed by Launder and Spalding [16], the k-ε standard turbulence model uses Navier-Stokes equations, Eq. (2), to obtain equations for k and ε, respectively, as:

$$U_j\frac{\partial k}{\partial x_j} = \frac{\partial}{\partial x_j}\left[\left(\nu + \frac{\nu_t}{\sigma_k}\right)\frac{\partial k}{\partial x_j}\right] + \nu_t\left(\frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i}\right)\left(\frac{\partial U_i}{\partial x_j}\right) - \varepsilon, \tag{4}$$

$$U_j\frac{\partial \varepsilon}{\partial x_j} = \frac{\partial}{\partial x_j}\left[\left(\nu + \frac{\nu_t}{\sigma_\varepsilon}\right)\frac{\partial \varepsilon}{\partial x_j}\right] + c_1\nu_t\frac{\varepsilon}{k}\left(\frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i}\right)\left(\frac{\partial U_i}{\partial x_j}\right) - c_2\frac{\varepsilon^2}{k}, \tag{5}$$

where empirical constants are: $c_1=1,42$, $c_2=1,92$, $\sigma_k=1$ and $\sigma_\varepsilon=1,22$.

The parameters k and ε are defined by equations (6) and (7), respectively:

$$k = \frac{1}{2}(u'^2 + v'^2 + w'^2), \tag{6}$$

$$\varepsilon = \frac{k^{\frac{3}{2}}}{l}\,, \tag{7}$$

where l is a characteristic dimension (m) and u', v' and w', are components of velocity fluctuations (m/s).

Boundary conditions

The boundary conditions are presented at Fig. 2a. Note that the inlet velocity depends of pressure drop and first it has been used the follow to obtain de volumetric flow:

$$Q = -6{,}39.10^{-4}\Delta p^2 - 3{,}67.10^{-2}\Delta p + 9{,}94, \tag{8}$$

where Q is the volumetric flow (m³/s) and Δp is the pressure drop (Pa). Eq. (8) has been obtained by quadric curve adjust of Munters EM50 fan pressure drop chart. Then, velocity is calculated by the follow equation:

$$V_0 = Q{A_0}^{-1}, \tag{9}$$

where V_0 is the inlet velocity (m/s) and A_0 is the inlet area (m²).

Mesh grid

According to Versteeg and Malalasekera [17] the implementation of wall boundary conditions for turbulent flows starts evaluating the y^+ value defined as:

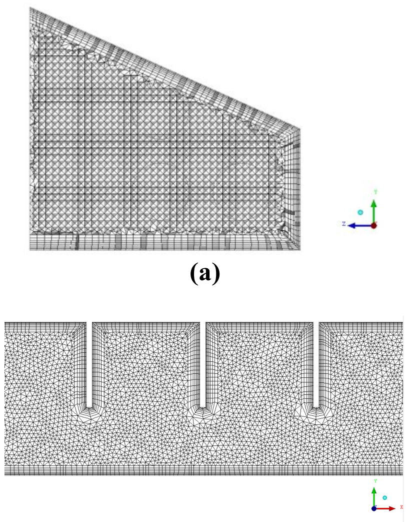
$$y^+ = \frac{y}{\nu}\sqrt{\frac{\tau_w}{\rho}}, \tag{10}$$

where y is the distance between wall and first mesh node (m) and τ_w is the shear stress (Pa). For k-ε standard turbulent model y^+ values must be at the range of 20 to 100 [18].

Tab. 1 shows the number of element of each mesh. To refine then it has been used a ratio factor of 8. The maximum value of y^+ observed was proximally 120 with y^+ of 50 as standard mean value. All of tetra elements had an aspect ratio above 1/5 and it has been considerate acceptable.

Table 1 Meshes grid.

Case	Mesh	Elements number
24° inclination	Coarse	102 848
	Intermediate	761 629
	Fine	5 879 906
Deflector	Coarse	320 948
	Intermediate	1 028 232
	Fine	7 070 139
6° inclination	Coarse	122 476
	Intermediate	935 328
	Fine	7 394 880
Ceiling	Coarse	117 039
	Intermediate	966 430
	Fine	7 646 347



(a) / **(b)**

Figure 4 Details of deflector fine mesh at planes **(a)** x = 42m and **(b)** z = 6m.

Some planes of deflector fine mesh case are show in Fig.4. All the meshes are very similar with prism layers at the wall and tetra elements inside the fluid domain. The worst quality elements are localized at the transaction between prism and tetra elements.

The chosen convergence criteria was the RMS [18] (Root Mean Square) $\leq 10^{-6}$, except for the energy equation that has achieved RMS $\leq 10^{-5}$. To achieve convergence it has been used an autotimescale factor [18] of 1/10 for energy equation, 1/100 for turbulence equations and 1/20 for mass and momentum equations. The advection scheme that has been applied was High Resolution [18].

For each simulation it have been considered the Thermal Energy [18] heat transfer method (appropriate for low velocities) and the buoyance effect (with $g_y = -9.81 m/s^2$ because Montenegro city has an elevation of 34m).

RESULTS

To save time, it has been used parallel processing and the results of the coarse mesh have used as initial solution for the intermediate mesh, whose results, have been, in turn, used as initial solution for the fine mesh.

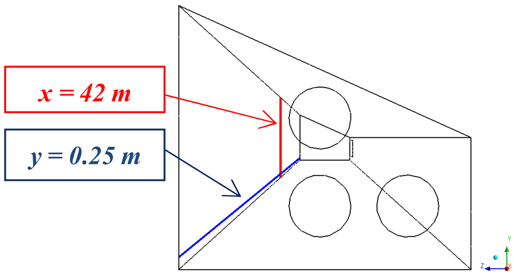


Figure 5 Lines of data analysis located at symmetry plane (z = W/2) at y = 0.25 m and x = 42 m.

The follow results have been taken on a line on symmetric plane (z = W/2) at high y = 0.25m (medium high of broiler chicken) and a vertical line at x = 42 m as show at Fig. 5.

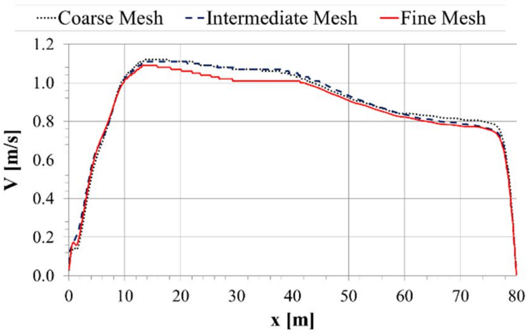
Fig.6 presents the mesh approach of deflector case to guarantee a mesh grid independence (the others cases follow the same pattern). Observe the coarse mesh have good results with less number of elements regarding fine and intermediate meshes.

The regions near of walls (x = 0 and x = 80 m) velocity assume values close to 0, respecting the boundary layer principle as showed at Fig. 7a. The same principle can be observed at Fig. 7b where velocity is close to 0 at the floor (y = 0) and roof (y = 5.4 m).

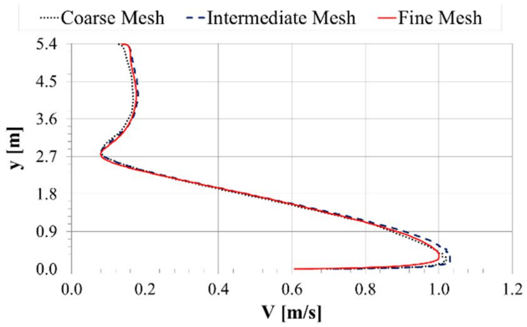
For all cases velocity reaches a maximum value just after the inlet (x ≈ 12 m) and it decrease until the outlets (x = 80 m) because of pressure drop as showed at Fig. 7a. The maximum velocity profile value at vertical line (Fig. 7b) is located near the broiler chicken level.

Temperature is almost constant through domain (the variation of temperature is proximally of 1°C) because the prevalent temperature is the inlet temperature in this model and it will not show at this work.

Contours of velocity at y = 0.25 m are exposed at Fig 8. Observe that for all cases there is some dead zones near to the entrance and a few ones near to the exit. For 24° inclination and deflector cases (Fig. 8a and 8b, respectively) there a velocity peak just after air inlet and the velocity decrease toward the air outlet.

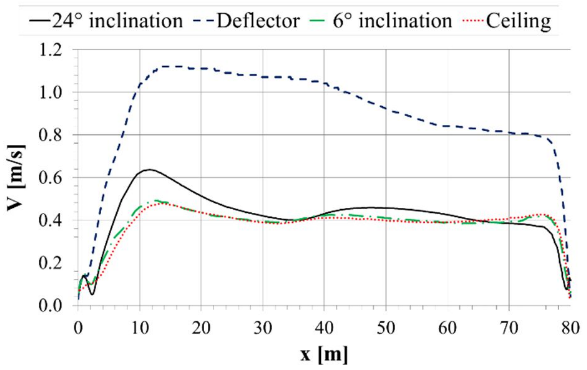


(a) y = 0.25m

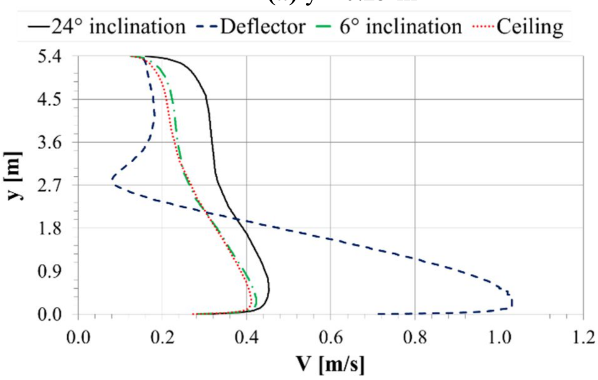


(b) x = 42 m

Figure 6 Mesh independence study.



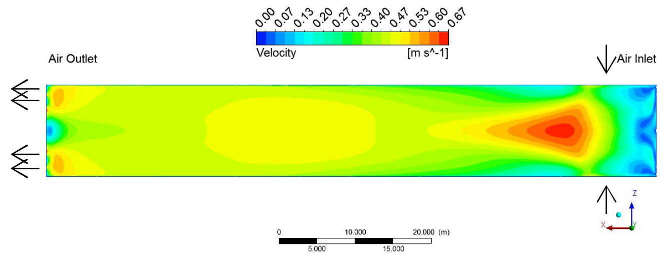
(a) y = 0.25 m



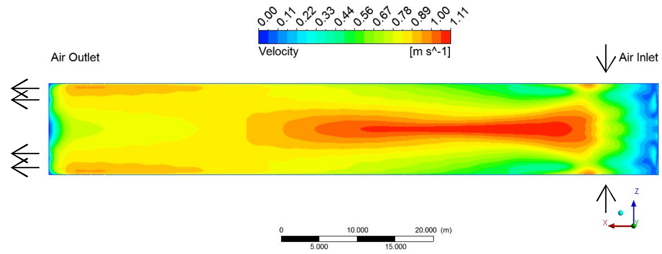
(b) x = 42 m

Figure 7 Comparative profiles of velocity.

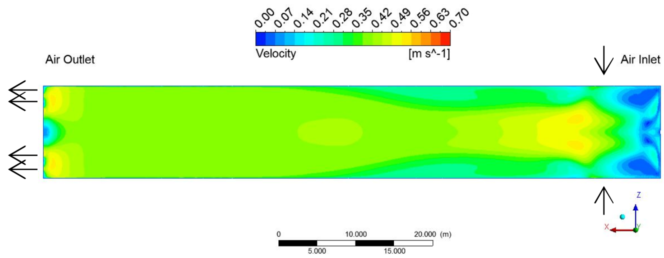
(a) 24° inclination



(b) Deflector



(c) 6° of inclination



(d) Ceiling

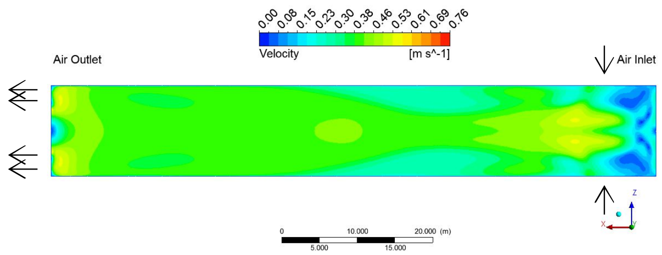


Figure 8 Velocity contour at plane y=0,25m.

The pressure drop with respectively volumetric flow is presented at Tab. 2. All cases have showed a low pressure drop and the highest pressure drop was 7.24 Pa (deflector case) because of the higher velocity values on its domain (Fig. 8b).

Table 2 Pressure drop.

Case	Δp [Pa]	Q [1.000 m^3h^{-1}]
24° inclination	5.88	34,93
Deflector	7.24	34,71
6° inclination	5.91	34,92
Ceiling	6.04	34,90

CONCLUSIONS

This work presented a CFD analysis using Ansys® CFX version 14.5 of a poultry building located at Montenegro city (Brazil). It has been used the fluid air as ideal gas and the k-ε standard turbulence model to predict the airflow through the poultry building.

Four cases have been studied: 24° inclination, deflector, 6° inclination and ceiling. Curves of velocity and temperature contour have been presented to compare the cases.

The k-ε standard turbulence model have been used based on literature about CFD application on poultry houses. However, it is possible that SST k-ω could be more adequate for this kind of flow pattern because of many separated flow regions in this geometry.

Results have been showed the highest pressure drop occurs at deflector. However, the values are very closely and velocity profiles outcomes have showed a higher velocity with application of deflector. In addition, velocity contour at chicken level have showed less dead zones next to entrance regarding other cases. Hence, the application of deflectors can be a good solution.

A real situation is pretty more complex than the model of this study. A poultry house can have side inlets, heaters, fountains, feeders and luminaire. All of these objects have an influence of flow pattern. Future works could analyze some of these aspects.

REFERENCES

[1]Organisation for Economic Co-operation and Development (OECD), Agricultural Outlook 2014-2023, *OECD/Food and Agriculture Organization of the United Nations*, 2014.

[2] K. Hamerschlag, Meat eater's guide, *Environmental Working Group. (EWG)*, 2011.

[3] P.W. Gerbens-Leenes, M.M. Mekonnen, A.Y. Hoekstra, The water footprint of poultry, pork and beef: A comparative study in different countries and production systems, *Water Resources and Industry* 1-2 (2013) 25 – 36.

[4] *Brazilian Poultry Association* (UBABEF), Annual Report 2014, 2014.

[5] Nórton, T., Sun, Da-Wen, Grant, J., Fallon, R., Dodd, V., Applications of computational fluid dynamics (CFD) in the modelling and design of ventilation systems in the agricultural industry: A review, *Bioresource Technology 98*, 2007, pp. 2386-2414

[6] van Ouwerkerk, E.N.J., Voskamp, J.P., Aliskan, Y., Climate simulation and validation for an aviary systems for laying hens, *Institute of Agricultural and Environmental Engineering*, AgENG, 1994, Report N 94-C-063

[7] Mistriotis, A., de Jong, T., Wagemans, M.J.M., Bot, G.P.A., Computational fluid dynamics as a tool for the analysis of ventilation and indoor microclimate in agricultural buildings, *Netherland Journal of Agricultural Science 45*, 1997, pp. 81-96

[8] Worley, M.S., Manbeck, H.B., Modelling particle transport and air flow in ceiling inlet ventilation systems, Transactions of the *ASAE 38* (1), 1995, pp. 231-239

[9] Blanes-Vidal, V., Guijarro, E., Balasch, S., Torres, A.G., Application of computational fluid dynamics to the prediction of airflow in a mechanically ventilated commercial poultry building, *Biosystems Engineering*, 2008, pp. 105 -116

[10] Seo, I.H., Lee, I.B., Moon, O.K., Kim, H.T., Hwang, H.S., Hong, S.W., Bitog, J.P., Yoo, J.I., Kwon, K.S., Kim, Y.H., Han, J.W., Improvement of the ventilation system of a naturally ventilated broiler house in the cold season using computational simulations, *Biosystems Engineering 104*, 2009, pp. 106-117

[11] Saraz, J.A.O., Measurement and CFD Modelling of Ammonia Concentration Flux and Thermal Environment Variables in Open Inside Broiler Housing, *PhD. thesis*, Universidade Federal de Viçosa, Viçosa, Brazil, 2010

[12] Saraz, J.A.O., Martins, M.A., Marín, O.L.Z., Damasceno, F.A., Velasquez, H.J.C., Una revisón acerca de la dinamica de fluidos computacionales (CFD) em instalaciones avícolas, *Dyna, año 79*, 2012, pp. 142-149

[13] Mostafa, E., Lee, I.B., Song, S.H., Known, K.S., Seo, I.H., Hong, S.W., Hwang, H.S., Biotg, J.P., Han, H.T.,Computational fluid dynamics simulation of air temperaturedistribution inside broiler building fitted with duct ventilation system, *Biosystem Engineering 112*, 2012, pp. 293-303

[14] Bustamante, E., García-Diego, F.J., Calvet, S., Estellés, F., Beltrán, P., Hopitaler, A., Torres, A.G.,Exploring ventilation efficiency in poultry buildings: the validation of computational fluid dynamics (CFD) in cross-mechanically ventilated broiler farm, *Energies 6*, 2013, pp. 2605-2623

[15] Fawaz, H., Abiad, M.G., Ghaddar, N., Ghali, K., Solar-assited localized ventilation system for poultry brooding, *Energy and Buildings 71*, 2014, pp. 142-154

[16] Launder, B.E. and Spalding, D.B., The numerical computation of turbulent flows, *Computer Methods in Applied Mechanics and Engineering 3*, 1974, pp. 269-289.

[17] Versteeg, H. K. e Malalasekera, W., An Introduction to computational fluid dynamics, *Longman Scientific & Technical*, 2007, 503p.

[18] User's Manual Version 14.5, 2012. Copyright 1996-2012 ANSYS Europe Ltd.