Modeling of fluid dynamics of airflow in the breeding livestock using COMSOL Multiphysics software

Guilherme de Souza Dourado [a]

[a] Center for Engineering, Modeling and Applied Social Sciences, Federal University of ABC - UFABC, São Bernardo do Campo, São Paulo, Brazil, g.dourado@aluno.ufabc.edu

Abstract. This study aimed to investigate the airflow patterns and velocity fields inside two stables for livestock breedings located in the Czech Republic. Their 3D models were constructed to post discretization and simulation by Computational Fluid Dynamics. The measurements in both stables were used as inputs for the simulations. The results shown by the velocity magnitudes and streamlines indicate that the fields are well-ventilated.

Keywords. NH3, CFD, livestock, barn, airflow.

1. Introduction

Livestock plays a crucial role in the world's food supply and the economy of agriculture. It contributes 10-15% of the total food calories and is responsible for 25% of the dietary protein consumed globally. Additionally, it plays a significant role in the world's agricultural output, contributing to 36% of its value. Moreover, around 1.3 billion individuals depend on livestock for their livelihood [1].

Several research studies have emphasized the adverse effects of the expanding livestock industry, especially concerning the discharge of ammonia (NH_3). According to research findings, livestock rearing and production in the European Union (EU-28) were responsible for two-thirds of the total agricultural NH_3 emissions in 2014 [2]. Rearing of pigs globally is responsible for 15% of ammonia emissions from livestock, for example [3].

The most significant sources of NH_3 emissions stem from manure management. Emissions related to manure management include those from animal housing systems, handling and storing manure, manure deposition during grazing, and the application of manure as fertilizer on soils [2]. Also, the overpopulation of pigs in stables leads to accumulating urine and feces and generating these metabolic gas.

The release of significant ammonia compounds can lead to degraded air quality and ecosystem damage. NH_3 in the atmosphere forms fine particulate matter that can lead to respiratory issues in humans or worsen existing respiratory problems [4]. Moreover, inside the stables, this release harms the animal's welfare. Factors such as the size and material of the stable, as well as good ventilation and drainage, need to be considered to prevent these issues.

The ventilation in the stables is essential for controlling the temperature inside them and, consequently, the ammonia concentration. However, it must be done carefully to avoid decreasing the temperature too much in cold climates or causing discomfort for the animals with high air speeds. Therefore, this study aims to map the airflow patterns inside two barns in the Czech Republic, the first one is located in the Hodětín district, and the second one is in the Záhoří district and it is going to be evaluated their ventilation for possible post-ammonia emission studies.

2. Methodology

2.1 Computational Fluid Dynamics (CFD)

The Navier-Stokes equations describe the motion of fluids based on the fundamental conservation laws of mass, momentum, and energy. They are differential equations that describe the time evolution of velocity, pressure, and density of a fluid motion and can be applied, in general, to any Newtonian fluid particle. However, these equations are highly complex and challenging to solve in their complete form once it has nonlinearities.

Computational Fluid Dynamics (CFD) is the application of numerical methods for calculating those Navier-Stokes equations without simplifications in an approximated solution once it discretizes and solves them, typically using iterative methods. An approach to the solution involves discretizing the fluid domain into a set of control volumes and then solving the equations for each volume using a finite difference or finite element approach. This strategy is very common in CFD softwares and is called the finite volume method.

Applying the CFD to this work would be advantageous since the objectives include obtaining the air velocity fields and streamline patterns. And the analytical solution would not be as detailed as a numerical solution.

A simple workflow could be done to make a CFD simulation, and it consists in:

1. Implementing engineering hypothesis to analyze the problem for possible simplifications of the governing equations and post-verification of the results;
2. Modeling the geometry in one or more dimensions to make the computational solution available;
3. Meshing this geometry to discretize the model for the calculations;
4. Executing the simulation itself;
5. Making a post-analysis of the results.

2.2 Engineering Hypothesis

To implement the engineering hypothesis, first, it is necessary to observe the data shown in Table 1.

Tab. 1 – Simulation inputs.

	Hodětín	Záhoří	Units
T_i	288.09	291.85	K
T_0	265.88	286.95	K
P	961.73	960.00	hPa
V	0.443	2.36	m s^{-1}

It has the measurements taken in each stable, where T_i is the inside average temperature in Kelvins (K), T_0 is the outside average temperature also in Kelvins, P is the average pressure in hectoPascal (hPa), and V is the maximum velocity in meters per second (m s^{-1}).

Observing these data, which will be the inputs for the simulations, some conclusions are made:

- The fluid could be considered at a low Mach number;
- Thus, it is incompressible (with constant density);
- And with a high Reynolds number.

Also, for the simulations, it is assumed that the wall of the geometries has the no-slip condition, the fluid is turbulent in a single-phase flow, and the analysis was for a stationary regime.

2.3 3D Model

After that, it was necessary to make the 3D models. For this, the Autodesk Inventor software [5] was used.

The Hodětín barn, shown in Figure 1, is 9.7 meters long, 9.2 meters wide, and 3 meters high. The ventilation system consists of three windows where the air enters and two fans where it goes out.

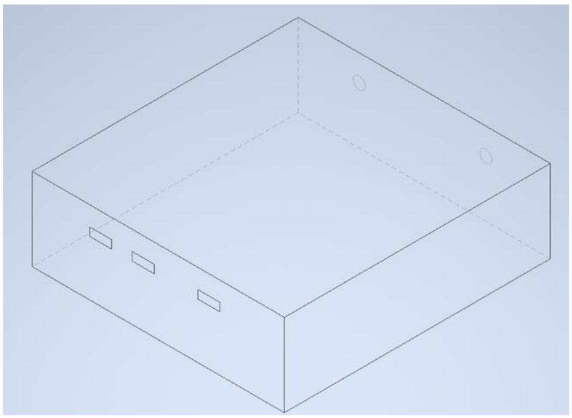

Fig. 1 – Geometry of the Hodětín barn.

The Záhoří barn, shown in Figure 2, is 11.4 meters long, 7 meters wide, and 2.5 meters high. The ventilation system consists of six windows, where two of them (in the front of the image) are inlets, and the other four and the fan at the top work as outlets for the air.

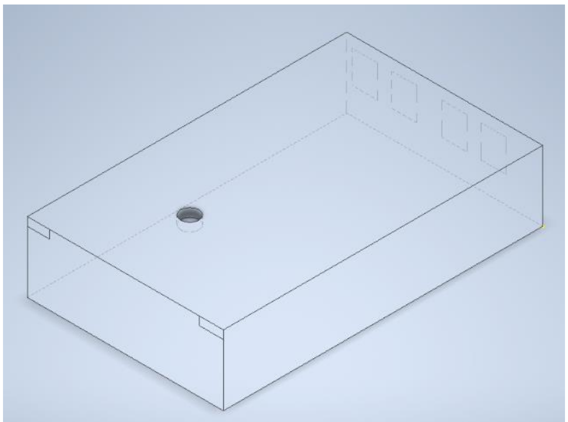

Fig. 2 - Geometry of the Záhoří barn.

2.4 Mesh

With the 3D models, it was possible to discretize them, and the COMSOL Multiphysics [6] software was used for it in version 6.1.

The meshing of each field was unstructured due to the circular forms of the fans. The number of elements of the Hodětín barn was 355,777, and of the Záhoří barn was 478,385, with the average quality of the elements for both of them being approximately

0.7. They are shown in Figures 3 and 4.

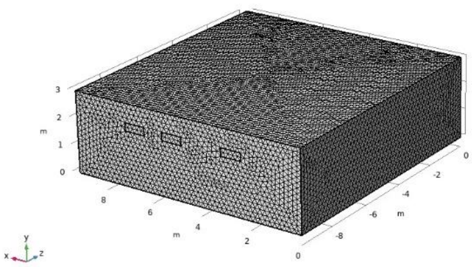

Fig. 3 – Mesh of the Hodětín barn.

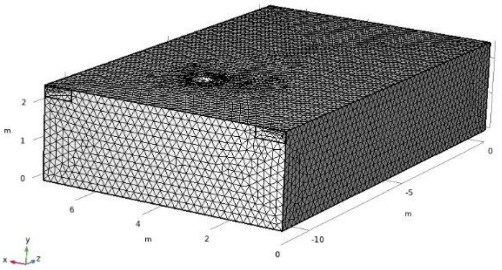

Fig. 4 - Mesh of the Záhoří barn.

2.5 Simulation Considerations

Then, for the simulation, it is necessary to determine a strategy to solve the turbulent flow. Of the available options, the Reynolds-Averaged Navier-Stokes (RANS) model is the best option, where additional transport equations are included to represent the turbulent effects. The k-ε turbulence model was selected in the RANS models because of its uncomplicated structure, reliable effectiveness, and advantageous ability to converge and deliver reasonably accurate results. This model resorts to transport equations to the kinetic energy (k) and the dissipation rate (ε) to describe the turbulent flow [7].

Other simulation setups are that it was considered an automatic pseudo-time stepping from physics and an automatic CFL number. Both simulations had 1000 iterations of limit for converging and used the smoothed aggregation AMG solver for all the variables. For accelerating the convergence time, the multigrid technique was used with the V-cycle with five levels of mesh.

3. Results

3.1 Hodětín geometry

In order to see the results, 3D plots of the streamlines were generated. Figure 5 shows the air behavior in the Hodětín geometry. Figure 6, with fewer streamlines plotted and an upper perspective, shows with more clarity some recirculation regions of the air.

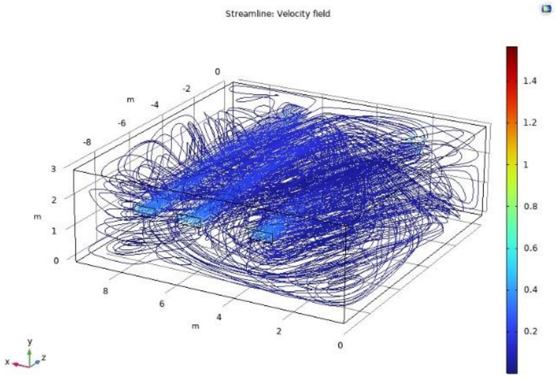

Fig. 5 – Isometric view of the airflow streamlines of the Hodětín barn.

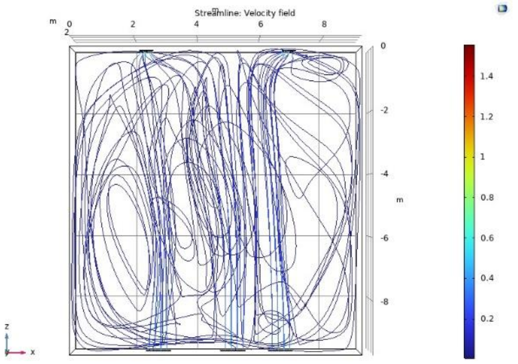

Fig. 6 - Upper view of the airflow streamlines of the Hodětín barn.

Figure 7 shows the velocity magnitude by slices on the X-Y plane. The maximum velocity inside the field is approximately 0.35 m s^{-1} and is shown in red, right in front of the inlets. Moreover, the zones with less velocity magnitude are in blue, with 0 m s^{-1}. Also, it is possible to observe in Figure 8 that in the outlet fans, the velocity increases to approximately 1,4 ms^{-1} since the fans generate a low-pressure region that sucks the air, accelerating him away.

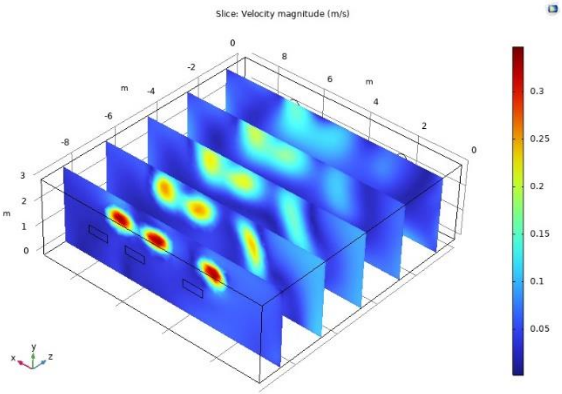

Fig. 7 – Slices View Velocity Magnitude of the Hodětín barn.

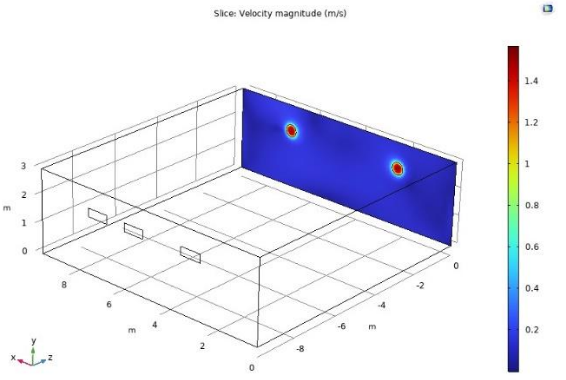

Fig. 8 - Slices View for Velocity Magnitude in outlet fans of the Hodětín barn.

3.2 Záhoří geometry

In the Záhoří geometry, as shown in Figures 9 and 10, the inlet velocity streamlines are more targeted for the outlet big windows. Furthermore, unlike the other simulation, the field does not have many recirculation areas.

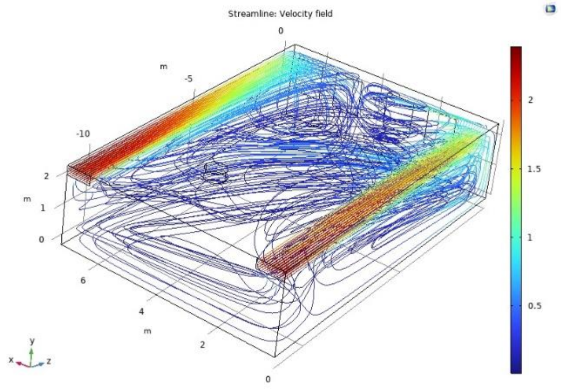

Fig. 9 - Isometric view of the airflow streamlines of the Záhoří barn.

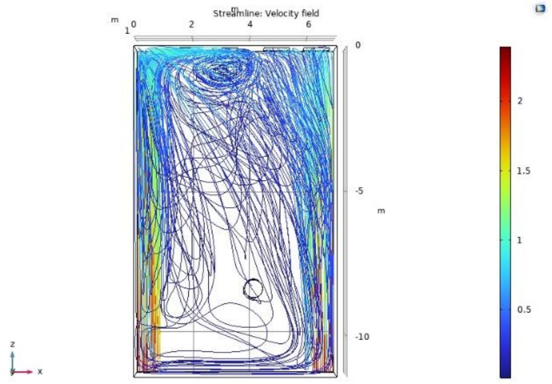

Fig. 10 - Upper view of the airflow streamlines of the Záhoří barn.

The velocity magnitude is maximum in the inlets, as shown in Figure 11, and is approximately 2.3 m s^{-1}. Also, it has low values for velocity in the central area. For the outlet windows, the velocity magnitude is approximately 1.3 m s^{-1} (Figure 12). This makes sense because the windows' area is more significant than the inlet, and consequently, the velocity is minor.

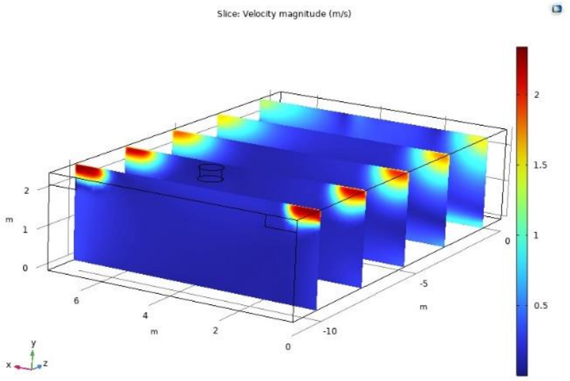

Fig. 21 - Slices View Velocity Magnitude of the Záhoří barn.

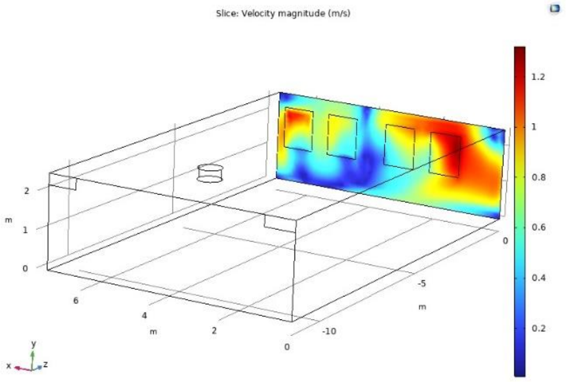

Fig. 12 - Slices View for Velocity Magnitude in outlet windows of the Záhoří barn.

4. Conclusion

In this work, a CFD workflow was performed to see the airflow patterns inside two barns located in the Czech Republic for analyzing their velocity fields. For this, firstly, 3D models were made with the Inventor software. After that, the discretization of these models was executed in COMSOL Multiphysics software by generating. Finally, with the determined engineering hypothesis, the CFD simulation was made in the same software, and the airflow streamlines were plotted, such as the velocity magnitude planes.

The results presented in this work show that the velocity fields are coherent with the physics laws. Furthermore, it is concluded that the streamlines showed good ventilation points, which could reduce the adverse effects of ammonia emission. Additionally, the air speeds do not increase much, not creating any risk for the animals.
Further studies could be done to analyze the temperature behavior due to ammonia gasses and join these results with an airflow analysis. Moreover, an ammonia emission modeling could be made and validated with empirical data.

5. References

[1] FAO. Food and Agricultural Organization of the United Nations: Statistics Division. [Online].; 2011 [cited 2023 March 09. Available from: https://www.fao.org/statistics/en/.

[2] Sajeev E.P.M., Amon B., Ammon C., *et al*. Evaluating the potential of dietary crude protein manipulation in reducing ammonia emissions from cattle and pig manure: A meta-analysis. Nutr Cycl Agroecosyst. 2017 November;(110): 161–175.

[3] Olivier J.G.J., Bouwman A.F., Van der Hoek K.W., Berdowski J.J.M. Global air emission inventories for anthropogenic sources of NOx, NH3 and N2O in 1900. Environmental Pollution. 1998;(102): 135-148.

[4] Kampa M, Castanas E. Human Health Effects of Air Pollution. Environmental Pollution. 2008;(151): 362-367.

[5] Autodesk. Autodesk Inventor. [Online]. [cited 2023 February 20. Available from: https://www.autodesk.com.br/products/inventor/overview.

[6] Comsol. Comsol 6.1. [Online].; 2022 [cited 2023 February 08. Available from: https://www.comsol.com/product-download.

[7] Tomasello N, Valenti F, Cascone G, Porto SMC. Development of a CFD Model to Simulate Natural Ventilation in a Semi-Open Free-Stall Barn for Dairy Cows. Buildings. 2019 August; 9(8): 183.