COMPUTER SIMULATIONS OF FLOW OVER WIND TURBINE BLADES USING OPEN-SOURCE CODE

Isadora Venancio Bastos[1*] and Felipe Pamplona Mariano[1]

[1]*Laboratório de Engenharia Térmica e de Fluidos (LATEF); Escola de Engenharia Elétrica, Mecânica e de Computação (UFG), Universidade Federal de Goiás (UFG), Goiânia, Brazil.*
*Corresponding author. *E-mail: venanciobastos@discente.ufg.br*

ABSTRACT

The drag and lift coefficients are dimensionless numbers that describe the force interactions between a fluid flow and solid objects, and they are essential for designing and optimizing wind turbine blades to improve efficiency. The computational approach enables the study of different aerodynamic profiles, allowing for a detailed analysis of their behavior under various flow conditions. To investigate these coefficients, a geometric model and three-dimensional complex meshes were developed to study airflow over a wind turbine blade based on a NACA 4415 profile, with a 10° angle of attack, unit length, and chord, and a Reynolds number of 750,000. For this purpose, the OpenFOAM software was chosen, as it is a free, open-source tool dedicated to solving Computational Fluid Dynamics problems. The main result is the development of a process for conducting computer simulations of three-dimensional turbulent flows over complex geometries, specifically wind turbine blades.

Keywords: Computational fluid dynamic; OpenFOAM; Wind turbine blade.

INTRODUCTION

Wind turbine blades are the devices responsible for capturing the kinetic energy of the wind and converting it into rotational energy, by producing work. The blades need optimized the aerodynamic shape to obtain the maximum efficiency. Then to provide the optimized design the present paper uses the Computational Fluid Dynamic techniques [1].

In CFD, partial differential equations, that model fluid dynamics, are solved using numerical methods in conjunction with advanced computational techniques. This approach allows the computational simulation of fluid flows, resulting in velocity, pressure, temperature, force fields, and other fundamental variables essential for analyzing, designing, and optimizing equipment and systems [1] and [2].

The main objective of the present work is to establish the necessary procedures (a methodology) for conducting flow simulations over wind turbine blades. To achieve this, it is proposed to use the OpenFOAM software, as it is free and open source.

METHOD

The Reynolds Average Navier-Stokes and continuity equations, Eqs. (1) and (2), are used to model the fluid dynamic to three-dimensional incompressible flow, in steady state and the Spallart-Almaras turbulence model, Eq. (3), is used to solve the turbulent stress tensor:

$$\frac{\partial(u_i u_j)}{\partial x_j} = -\frac{1}{\rho}\frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j}\left[(\nu + \nu_t)\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} - \frac{2}{3}\delta_{ij}\frac{\partial u_k}{\partial x_k}\right)\right], \qquad Eq.\ (1)$$

$$\frac{\partial u_j}{\partial x_j} = 0, \qquad Eq.\ (2)$$

$$u_j\frac{\partial \tilde{\nu}}{\partial x_j} = C_{b1}\tilde{S}\tilde{\nu} + \frac{1}{\sigma}\left[\frac{\partial}{\partial x_j}\left((\nu + \tilde{\nu})\frac{\partial \tilde{\nu}}{\partial x_j}\right) + C_{b2}\frac{\partial \tilde{\nu}}{\partial x_j}\frac{\partial \tilde{\nu}}{\partial x_j}\right] - C_{\omega 1} f_w \frac{\widetilde{\nu^2}}{d^2}, \qquad Eq.\ (3)$$

where u_j is the velocity component, p is the pressure, ρ is the density, ν is the molecular viscosity, ν_t is the turbulence viscosity e $\tilde{\nu}$ is the turbulence viscosity modified, $\tilde{S}$ is the strain rate, $C_{b1}, C_{b2}, C_{\omega 1}$, are the model's constant given in [3] and [4], f_w is the buffer function and d is the wall distance.

OpenFOAM [3] was employed to solve Eqs. (1), (2) and (3) using second order Finite Volume Method [1]. During the post-processing stage, the three-dimensional geometry of the wind turbine blade is generated, followed by mesh generation over the fluid domain using HyperHexMesh. The fluid properties and those required by the pressure-velocity coupling algorithm are then specified [5], with Semi-Implicit Method for Pressure-Linked Equations (SimpleFOAM) being specifically utilized.

The process then proceeds to the computational simulation phase, and finally, the simulation results of velocity and pressure fields are prepared and the lift (C_L) and drag (C_D) coefficients are calculated by using Eq. (4), during the post-processing stage using ParaView.

$$C_{L,D} = \frac{2F_{L,D}}{\rho U_\infty^2 A_p}, \quad \textit{Eq. (4)}$$

when $F_{L,D}$ is the lift or drag force, U_∞ is the free stream velocity and A_p is the planar area of the blade given by $A_p = C \cdot b$, where C is the chord and b is the blade span.

RESULTS AND DISCUSSION

Flow simulations over a wind turbine blade formed by a NACA 4415 profile, to compare with [1, 5], with $C = 1.0$ m, $b = 1.0$ m, $U_\infty = 1.0$ m/s and $\nu = 1.3E(-6)$ m²/s, resulting in a Reynolds number of 7.5E(+5) were conducted [5]. The three-dimensional mesh obtained using HyperHexMesh is shown in Fig. 1.

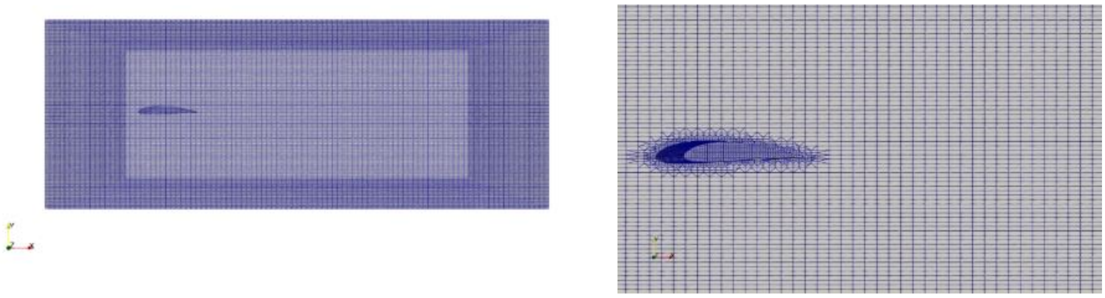

Figure 1. Lateral (left) and zoom (right) views of the mesh over the blade.

In Fig. 2(left), it is possible to identify a region of velocity in blue, close to the trailing edge of the wing, which indicates the presence of recirculation. In Fig. 2(right) is shown the pressure distribution along the cross-section of the blade. There is positive pressure in the region close to the leading edge, and negative pressure (in relation to atmospheric pressure) on the blade, that define a high lift force.

In Fig. 3 is shown the streamlines over the blade, highlighting more explicitly the phenomenon wing tip vortices, that define the efficiency of a blade. It should be understood that this phenomenon is inherent in the flow over a blade and, as a consequence, reduces the lift force.

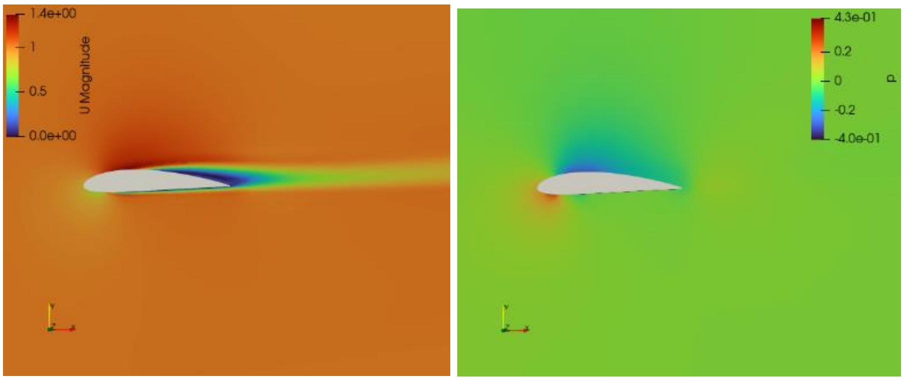

Figure 2. Velocity (left) and pressure (right) fields in a cross-section plane located in the center of the blade.

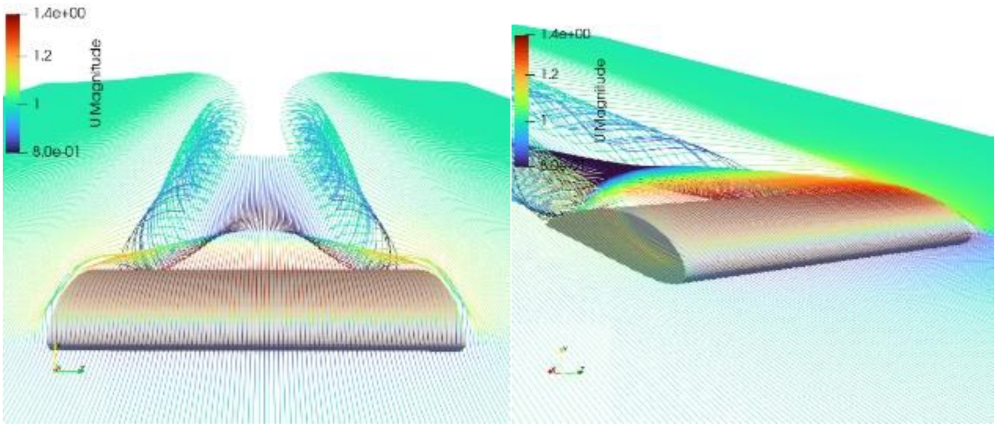

Figure 3. Streamlines over the blade.

The coefficients of lift and drag Eq. (4) were obtained in post processing step. The flow takes about 200 iterations to reach stability, and the force coefficients are obtained in the last interaction, as can be seen in Figs. (4) and (5). This gives a value of $C_L = 3.52E(-1)$ and $C_D = 1.03E(-1)$.

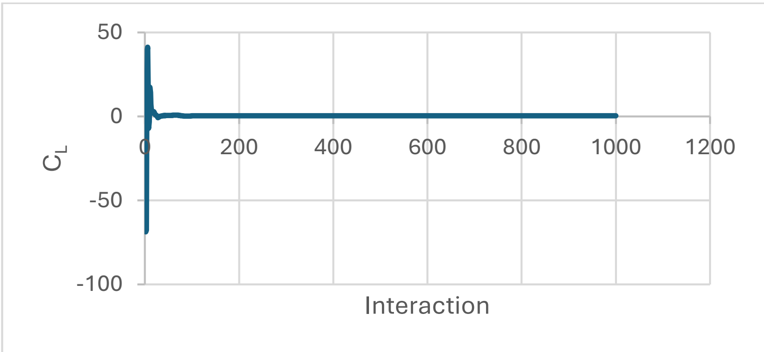

Figure 4. Lift coefficient C_L.

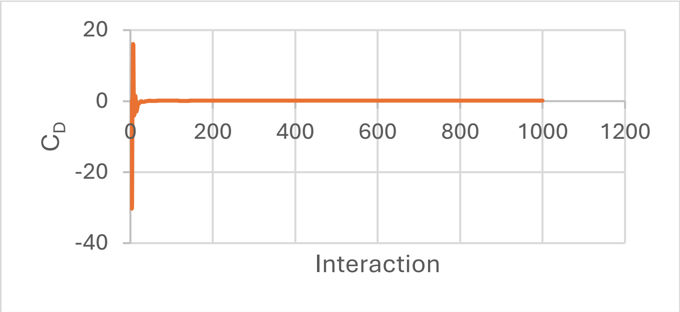

Figure 5. Drag coefficient, C_D.

CONCLUSION

The results obtained can be considered an efficient way of studying the behavior of flow over a blade. By using the images, it is possible to analyze the effects that influence the aerodynamic coefficients and their characteristics. In addition, the convergence of the results suggests that the simulations were successful.

However, it is necessary to improve the results obtained by carrying out simulations with more refined meshes and changing the design of the wing geometry in order to obtain more efficient wings with high aspect ratios.

ACKNOWLEDGMENT

The authors thank Eletrobras/FURNAS and the Research and Technological Development Program (P&D) of ANEEL for the infrastructure and financial support.

REFERENCES

[1] Anderson, JD. Computational Fluid Dynamics: The basis with applications. McGraw Hill, 1995.

[2] White FM. Fluid Mechanics. McGraw Hill, 2007; 6 ed.

[3] OpenFOAM. Available at: https://www.openfoam.com/documentation/guides/latest/doc/index.html. 2024.

[4] Spalart PR, Allmaras SR. A One-Equation Turbulence Model for Aerodynamic Flows. Recherche Aerospatiale, 1994; 1, 5-21.

[5] Calixtrato MP. Simulação de escoamentos sobre aerofólios utilizando malha não-estruturada no OpenFOAM®. Universidade Federal de Goiás, 2022.