v. 3, n. 21, 2023

Journal of
Engineering
Research

VALIDATION OF A CFD METHODOLOGY APPLIED TO A H-DARRIEUS VERTICAL AXIS WIND TURBINE

Celso Antonio Bittencourt Sales Junior
Universidade Federal de Itajubá, Instituto de Engenharia Mecânica – IEM/UNIFEI
Itajubá – Minas Gerais - Brasil
Lattes: http://lattes.cnpq.br/1826220299079724

Angie Lizeth Espinosa Sarmiento
Universidade Federal de Itajubá, Instituto de Engenharia Mecânica – IEM/UNIFEI
Itajubá – Minas Gerais - Brasil
Lattes: http://lattes.cnpq.br/9086096367189031

Yina Faizully Quintero Gamboa
Universidad Santo Tomas, Facultad de Ingeniería Mecánica
Tunja – Boyacá - Colômbia
Lattes: http://lattes.cnpq.br/2041413756720459

Abstract: Studies and research in wind energy have received great attention in recent years, mainly in the decentralized generation of electric energy, through small-size wind turbines. In this sense, the present work aims to validate a Computational Fluid Dynamics methodology for a case study of an H-Darrieus Vertical Axis Wind Turbine. The main recommendations of a robust reference methodology were replicated, which carried out a multivariate sensitivity analysis, using a transient two-dimensional simulation model and the sliding mesh technique to account for the turbine's rotational motion. Comparing the results from the simulation of the present work with the results of the reference methodology, satisfactory values were obtained, considering that the maximum difference in the final value of the Coefficient of Power was less than 9%. Furthermore, the torque profile of both studies was virtually identical. Based on these results, it was possible to conclude that the studied methodology was validated and, therefore, can be applied to other small-size H-Darrieus Wind Turbines, operating under similar operating conditions.
Keywords: H-Darrieus simulation, VAWT, CFD, CFD Validation, Wind Energy.

INTRODUCTION

Computational fluid dynamics (CFD) simulations are valuable tools for predicting wind turbine performance (MAYA et al., 2017; SARMIENTO et al., 2015). In this sense, this work consists of developing a methodology of CFD simulation of an H-Darrieus Vertical Axis Wind Turbine using the software ANSYS Fluent® based on Balduzzi et al., 2014, 2016a, 2016b work. In Balduzzi et al., 2016a work, the approach taken was an extensive sensitivity analysis of main simulation settings for this case type, which generated suitable parameters and minimally sufficient to get at least reasonable results to estimation the power generation of H-Darrieus wind turbine of small-scale. The study case consists in a H-Darrieus with three blades, 850 mm of radius (R_{turb}) and 246 mm of chord length (c). Basically, the main parameters and configurations investigated and established by Balduzzi et al., 2016a were:

- Computational domain: consists of a 2D model, containing a circular rotative zone (R.Z.), with diameter (Dr) equal to twice turbine's diameter (D_{turb}), enclosed by a rectangular stationary zone (S.Z.) both with adequate dimensions to simulation case. It was necessary to use some construction lines to control both element sizes around the blades and H-Darrieus wake region. The number of blades is equal to three with NACA0018 shape cambered by turbine radius (to compensate the flow curvature effects according to Balduzzi et al., 2014);
- Degree of refinement of the spatial and temporal discretization (mesh and time-step): all mesh was function of the number of division of blades walls once that the growth of the elements goes since the airfoils to the more external zones. In boundary layer region was utilized the inflation technique with adequate number of quadrilateral rows to capture all stages/zones of transition of flow. A special attention was given to interface between stationary and rotative zones and was found a suitable size to avoid any decompensate. With respect of time-step, this must be able to satisfy the condition of Courant number (Co) between 5 and 10 in proximities of blades and approximately 0.15 in interface;
- Turbulence model: k-ω SST model presented the best results;
- Solver type: pressure-based;
- Solution Algorithm: coupled;

- Convergence criteria: 0.1% of the CP value between two subsequent revolutions, that required until 90 revolutions of turbine.

Moreover, the authors created criteria and dimensionless numbers to measure the spatial/time discretization and numerical calculations stability (BALDUZZI et al., 2016b). Thereby it was stablished:

- The average dimensionless wall distance (y^+) of a blade must be approximately 1 to satisfy the turbulence model;

- Co < 10 in most regions, being acceptable some values until 50 if was in a little zone. At interface Co can vary a little bit around 0.50;

- It was created a dimensionless number called Grid Reduced Vorticity (GRV) that gives an estimate of the velocity variation within a single element. GRV is equal 1 when a 50% velocity variation between two adjacent cells occurs, therefore its average must be less than 0.01 in a region surrounding the blade within a distance equal to 50% of blade chord length from the blade wall.

Based on these criteria, several simulations were carried out, in order to replicate this methodology and learn a reference model to H-Darrieus simulations, which totaled around 200 hours of simulation time in a CPU with 3,40 GHz and 4 cores, on 64 GB RAM.

METHODOLOGY

Throughout this work, the following steps were carried out: Obtain the coordinate files of cambered NACA 0018; geometry creation; mesh generation; set solver configurations; to extract and to interpretate the results. In detail below:

COORDINATES POINTS OF CAMBERED NACA 0018

In Airfoiltools web site (Airfoil Tools, n.d.) is possible to get the coordinates points of several airfoil shapes through a download of a .txt file generated based on inputs given by user. In present case it was selected NACA0018 airfoil with a chord length equal to 246 mm and a radius camber equal to the wind turbine's radius (i.e., Camber = 850 mm) with origin at 25% of chord from trailing edge.

GEOMETRY

The geometry was initially developed in AutoCad® software, first entering the coordinate points of the NACA0018 obtained in the previous section. Once this is done, the software allows you to easily create a trailing edge rounding and construction lines around the blades with a water drop shape with a diameter equal to twice the length of the chord. In addition, a control line was created at about 15 mm from the airfoil wall in order to monitor element sizes in the region of the boundary layer (Fig. 1). The next step was to copy and rotate twice by 120° the first blade created and its control lines, which gives rise to the three blades of the wind turbine. Finally, a circumference was created, originating from the center of the turbine, with a diameter equal to twice the diameter of the turbine, thus completing the creation of the rotating zone. The stationary zone was created in ANSYS Design Modeler ® software and it consists basically in a rectangle with dimensions equal to 238 m x 102 m (length x width) containing a 'hole' of 3.4 m of diameter with center localized at 68 m from left edge of rectangle and 51 m from both bottom and top edges of rectangle, where will the rotating zone be located. Importing the rotating zone from the AutoCad® through a .iges file and joining to the stationary zone the geometry was finished (Fig. 2).

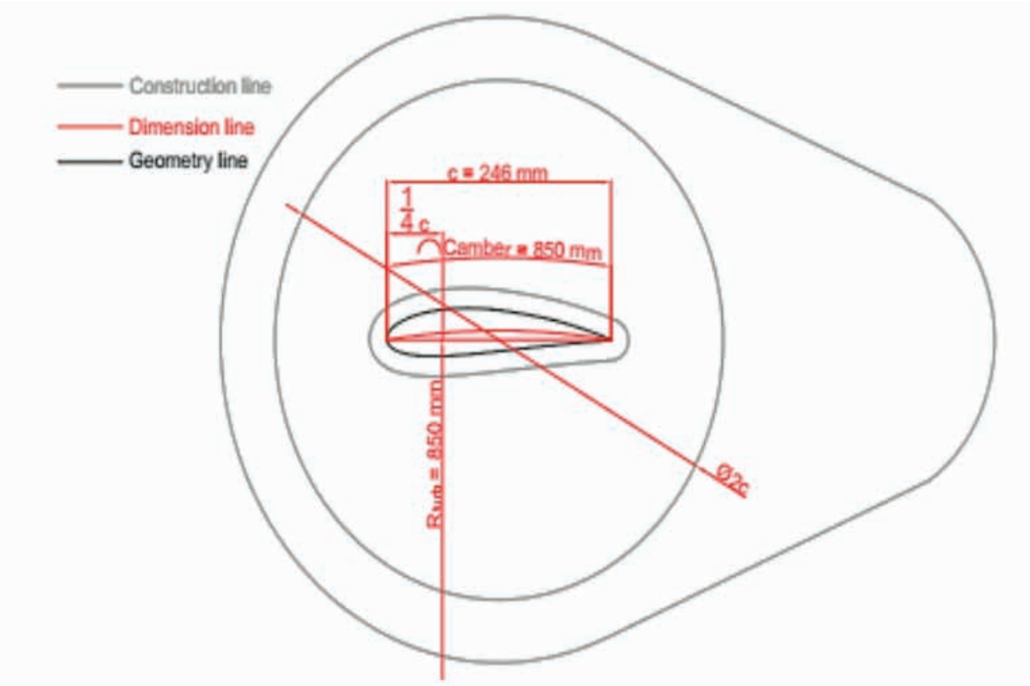

Figure 1 – Blade zone.

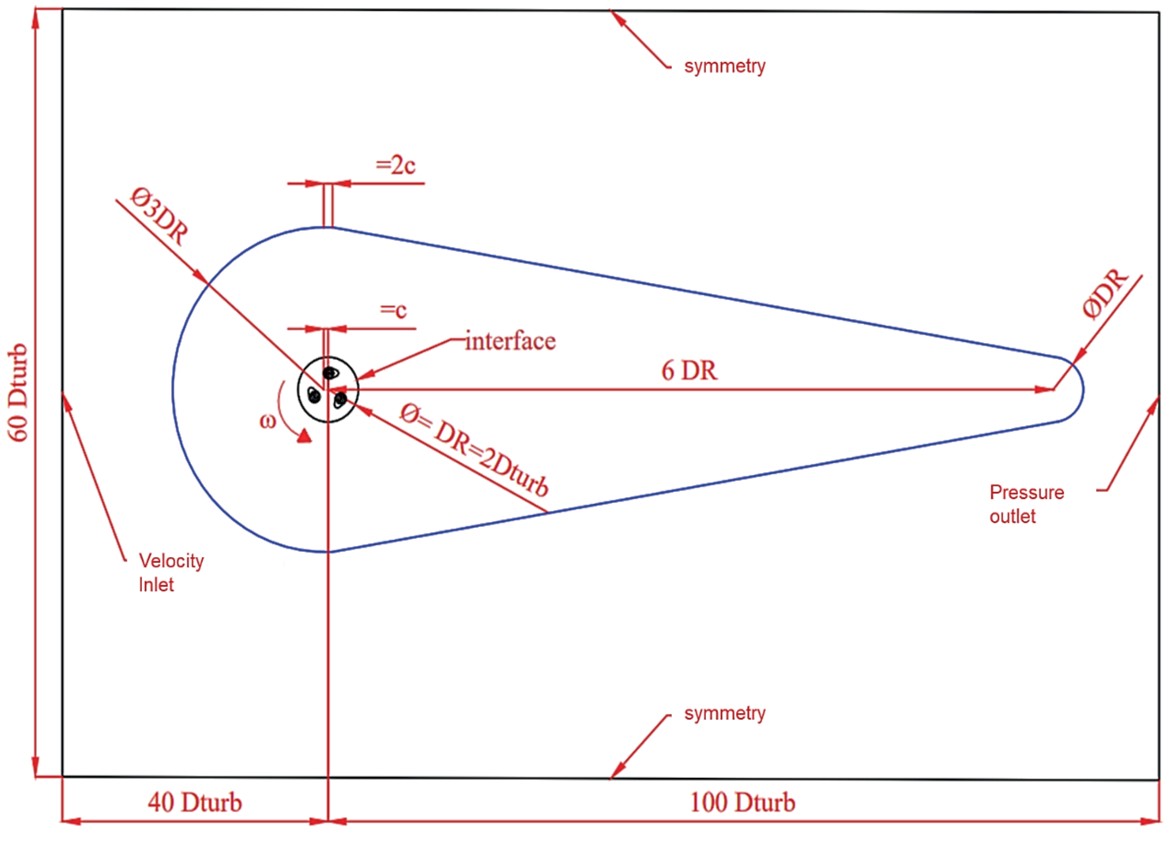

Figure 2 – Geometry and boundary conditions locations.

Journal of Engineering Research ISSN 2764-1317 DOI 10.22533/at.ed.3173212313062

MESH

The software utilized was ANSYS Meshing ® and the main mesh parameters provided and adopted was:

- Number of nodes on the airfoil: 523
- Number of quadrilaterals rows: 50
- First row thickness: 0,03 mm
- Interface elements sizes: 20 m

With these settings the mesh resulted in 353,006 elements on rotating domain and 165,669 elements on stationary domain, totalizing 518,675 elements. Figure 3 show the mesh details.

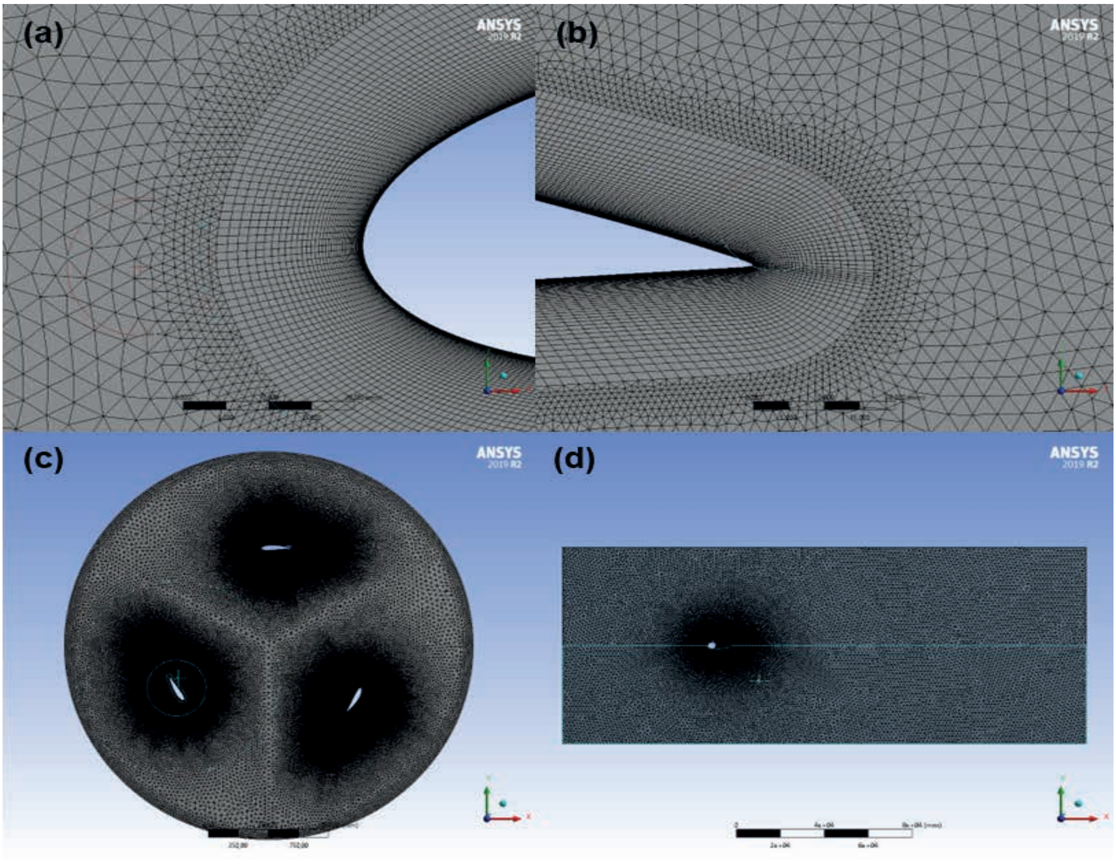

Figure 3 – Mesh details: (a) Leading edge, (b) Trailing edge, (c) RZ, (d) SZ.

SOLVER SETTINGS

The ANSYS Fluent ® software was used as a solver in transient case. The 2nd order upwind scheme was used for the spatial discretization of all the equations including a transport term (i.e., momentum, energy, and turbulence), as well as the bounded 2nd order implicit for the time differencing, to achieve a good resolution. The turbulence model utilized was k-ω SST, in line with Espinosa Sarmiento et al., 2021; Espinosa Sarmiento; Ramirez Camacho; de Oliveira, 2020; Sarmiento; de Oliveira; Camacho, 2017, along pressure-based solver-type, within coupled solution algorithm. It was necessary 30 iterations per time-step that in turn was equal to $2{,}228 \times 10^{-4}$ s. The convergence criteria equal to 0,1% of variation in Coefficient of Power (Cp) between two subsequent revolutions is achieved at approximately 90 revolutions, so it was necessary 28880 time-steps. Residuals of each quantity it was equal to 10^{-5}. Air like ideal-gas was used, with reference density equal to 1.222 kg/m³. At rotating zone was attribute the mesh motion with rotational velocity equal to 20.9412 rad/s, while the blade's wall is stationary relative to the rotation of mesh. It is very important to set the mesh interface correctly, selecting the respective inner and outer edges of the circular interface.

The boundary conditions pictured in Figure 2 were:

- Velocity Inlet: 8 m/s on X direction;
- Pressure Outlet: 0 Pa (Gauge);
- Symmetry on lateral boundaries;
- Interface between rotating and stationary zone;
- Wall with no slip condition on stationary (relative to rotating zone) blade surfaces.

RESULTS

The following main results were obtained:

- All meshes metrics (mainly element quality, skewness – min 0.00; max 0.60 - and orthogonal quality – min 0.50; max 1) were according to the recommendations of meshing software;
- The maximum average value of y+ was equal to approximately 1.25;
- The average value of Co at rotating zone was equal to approximately 0.9 and 0.5 on interface. The maximum value in rotating zone was equal to 10;
- The velocity contour is physically consistent indicating the development of the wake of the turbine (see Fig. 4);
- The torque profile was very similar to Balduzzi et al., 2016a results (see Fig. 5);
- All Cp relative errors compared with Balduzzi et al., 2016a results were below 10% (see Fig. 6).

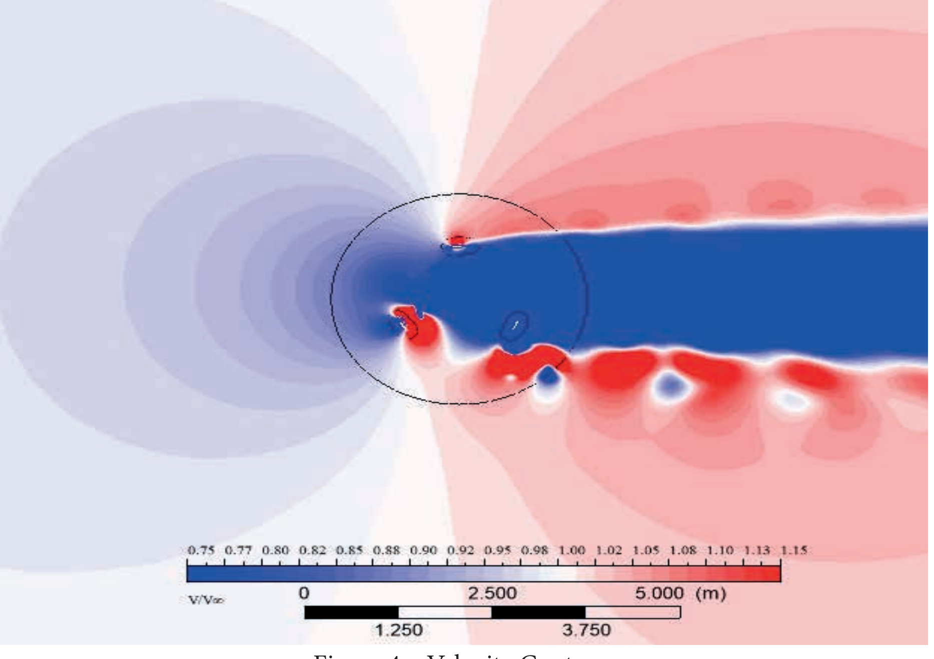

Figure 4 – Velocity Contour.

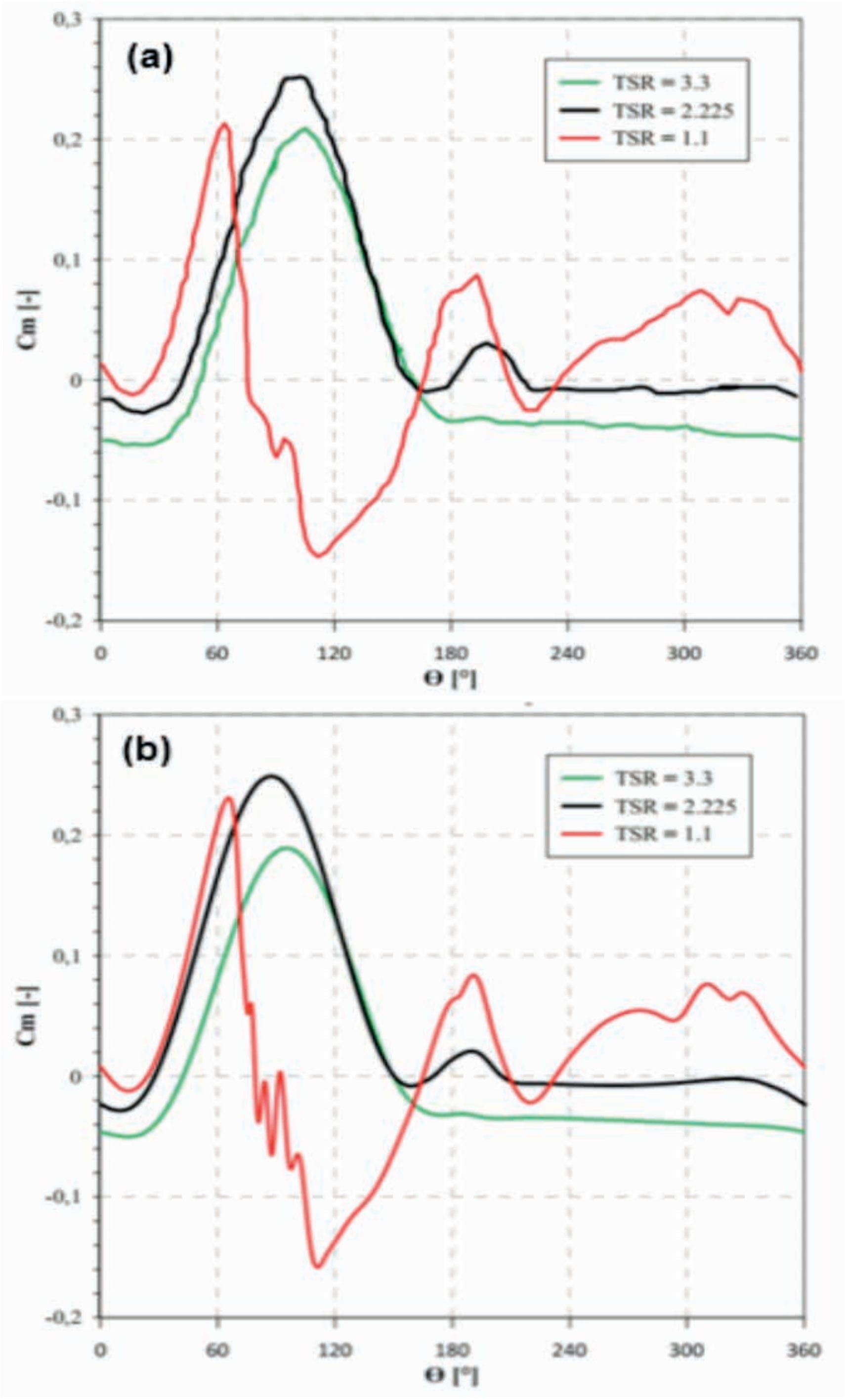

Figure 5 – Torque Profile. (a) Balduzzi et al., 2016a, (b) Present work.

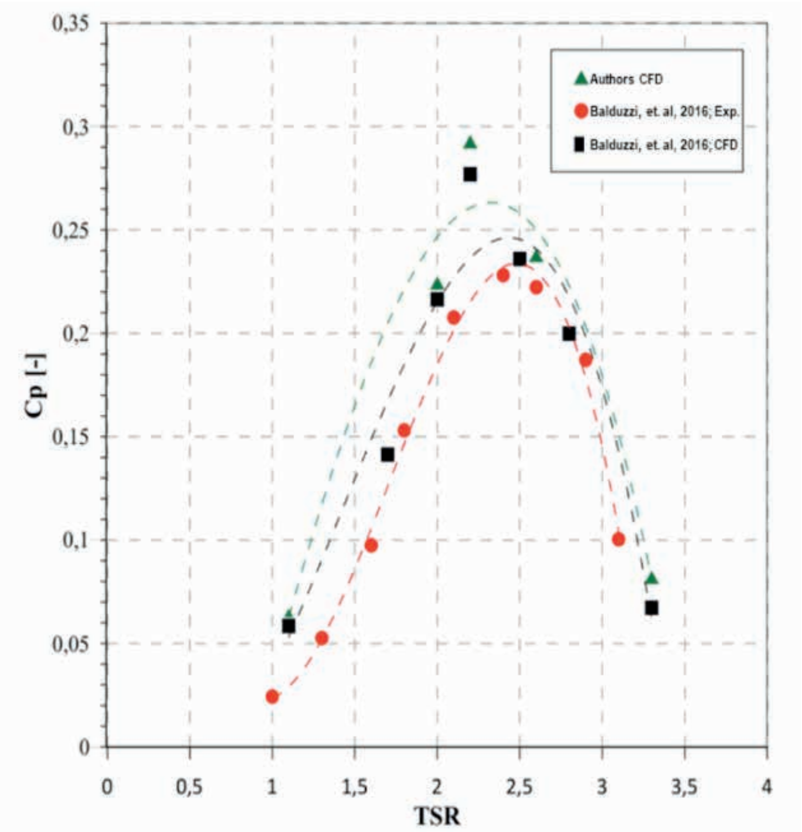

Figure 6 – Characteristic curves.

CONCLUSIONS

Based on the results, it is possible to infer that:

- In general, the mesh was adequate, except about the maximum average y+ that exceeded around 10% from Balduzzi et al., 2016a, however it was acceptable considering the final results;
- The average and maximum Courant number in the rotating zone suggests that in general the time-step was sufficient. However, in the interface, despite being low (0.5), it was still well above the recommended (0.15);
- The torque profile (torque coefficient versus azimuthal position of a blade) was very similar to Balduzzi et al., 2016a results (Fig. 5) as well as the characteristic curve (Fig.6) which presented a Cp difference of less than 9% for any operating point comparing with results of Balduzzi et al., 2016a.

Derived these observations it was possible to conclude that the studied methodology could be validated and therefore can be applied to 2D CFD simulations of other similar H-Darrieus small wind turbines.

ACKNOWLEDGMENTS

Special thanks to Fluid Dynamics and Turbomachinery (FDT - UNIFEI) group and Prof. Francesco Balduzzi for his support. Thanks to Santo Tomás University (USTA) for partner in project "Diseño aerodinámico de una turbina eólica de eje vertical de baja potencia para aplicaciones en una region con importante potencial eólico en Boyacá". Finally, thanks to National Council for Scientific and Technological Development (CNPq) and to the Research Support Foundation of the State of Minas Gerais (FAPEMIG) (processes APQ-00653-22 referring to the Projects: “Numerical and Experimental Analysis of Wind Microgenerators for Applications in Remote Regions in Brazil” - call 001/2022, registration at DPI UNIFEI No: PVDI297-2022) for funding this research.

REFERENCES

Airfoil Tools. Disponível em: <http://airfoiltools.com/>. Acesso em: 16 maio. 2023.

BALDUZZI, F. et al. **Blade design criteria to compensate the flow curvature effects in h-darrieus wind turbines**. Journal of Turbomachinery, v. 137, n. 1, p. 1–10, 2014.

BALDUZZI, F. et al. **Critical issues in the CFD simulation of Darrieus wind turbines**. Renewable Energy, v. 85, p. 419–435, 1 jan. 2016a.

BALDUZZI, F. et al. **Dimensionless numbers for the assessment of mesh and timestep requirements in CFD simulations of Darrieus wind turbines**. Energy, v. 97, p. 246–261, 15 fev. 2016b.

ESPINOSA SARMIENTO, A. L. et al. **Design and off-design performance improvement of a radial-inflow turbine for ORC applications using metamodels and genetic algorithm optimization**. Applied Thermal Engineering, v. 183, p. 116197, 25 jan. 2021.

ESPINOSA SARMIENTO, A. L.; RAMIREZ CAMACHO, R. G.; DE OLIVEIRA, W. **Performance analysis of radial-inflow turbine of ORC: New combined approach of preliminary design and 3D CFD study**. Journal of Mechanical Science and Technology, v. 34, n. 6, p. 2403–2422, 1 jun. 2020.

MAYA, D. et al. **BIOMASS GASIFICATION IN DOWNDRAFT DUAL STAGE REACTOR BY EXPERIMENTAL ANALYSIS AND SIMULATION WITH CFD TOOLS**. PAPERS OF THE 25TH EUROPEAN BIOMASS CONFERENCE, p. 808–816, 2017.

SARMIENTO, A. L. E. et al. **Gasification of Agro-Industrial Wastes for Electricity Cogeneration**. Proceedings of the ASME Turbo Expo, v. 3, 12 ago. 2015.

SARMIENTO, A. L. E.; DE OLIVEIRA, W.; CAMACHO, R. G. R. **Influence of the vortex design method in the performance characteristics of reversible axial rotors**. Journal of the Brazilian Society of Mechanical Sciences and Engineering, v. 39, n. 5, p. 1575–1588, 1 maio 2017.