

ENC-2022-0643

DIFFUSER STUDY FOR ENHANCING THE EFFICIENCY OF WIND TURBINES

Ricardo Victal Moreira Leite

Turan Dias Oliveira

Centro Universitário SENAI CIMATEC – Avenida Orlando Gomes, 1845 – Piatã, Salvador – BA, CEP: 41.650-010
ricardo.leite@aln.senaicimatec.edu.br; turan.oliveira@fieb.org.br

Alex Álisson Bandeira Santos

Centro Universitário SENAI CIMATEC – Avenida Orlando Gomes, 1845 – Piatã, Salvador – BA, CEP: 41.650-010
alex.santos@fieb.org.br

Luzia Aparecida Tofaneli

Centro Universitário SENAI CIMATEC – Avenida Orlando Gomes, 1845 – Piatã, Salvador – BA, CEP: 41.650-010
luzia.tofaneli@fieb.org.br

Abstract. *The wind energy is becoming more relevant in the Brazilian energy matrix; however, the traditional geometry of wind turbines has a power coefficient limited to approximately 59.3 %. In spite of this, there are evidences that the use of a diffuser encapsulating the turbine allows to the surpassing of said limit by causing an acceleration profile, which actuates in the flow through the turbine. Therefore, it is relevant the use of Computational Fluid Dynamics, notorious tool for reducing research costs in this field, to study the impact of different geometry characteristic in said profile. In order to achieve this, it was studied the influence of the height of the flange and the diffuser angle in a flanged diffuser in order to determine the optimal characteristics for the increase in efficiency in the wind turbine. Thus, the more effective design was found to be the one with the height of the flange equal to 0.25 D and the with a 5° diffuser angle, in accordance to what was expected from literature.*

Keywords: *Wind Turbine, Aerodynamics, Diffusers, DAWT.*

1. INTRODUCTION

The wind energy in Brazil had a significant growth between 2019 and 2020, reaching 14.98% increase in the capacity, going from 15.45 GW in 2019 to 17.75 GW of installed capacity in 2020. This data puts Brazil as the seventh biggest installed capacity of wind energy in the world and third in growth of said capacity (ABEEÓLICA, 2021). However, the classic momentum theory of Betz states that, for the conventional, not encapsulated, wind turbine design, the power coefficient is limited to 16/27 (approximately 59.3%). Burton et al. (2001) has shown that for a real conventional turbine, this value is usually below 50%, showing the restrictions of this design.

In this context, the use of a diffuser, encapsulating the wind turbine allows for, according to Dick (1984), surpassing the Betz limit, by generating an acceleration of the airflow before the passage in the turbine. Hansen et al. (2000) added that said acceleration is intimately dependent to the geometry of said encapsulation, while the results gathered by Abe and Ohya (2004) indicates that the velocity profile depends only on the variation in the radius (distance to the center of the rotor).

Among the various designs, the flanged diffusers stand out as it, according to Agha et al. (2018), allows for a smaller equipment, reducing loads, as well as, reducing the time needed for the vortices generated deteriorate, being one of the more studied types of diffusers. As such, according to Lipian et al. (2015) the maximum acceleration occurs when the height of the flange is around 0.3D and the angle of the diffuser is around 4° to 6°.

In light of this, the present study becomes relevant; having the general objective of analyzing two different parameters, that influence the geometry of flanged diffusers (flange height and diffuser angle) through Computational Fluid Dynamics (CFD), in order to determine the optimal diffuser geometry for enhancing the efficiency of the wind turbine.

2. METHODOLOGY

2.1 Computational modeling

In this study, a numerical approach was used in order to solve the continuity and Navier-Stokes equations, Eq. (1) through Eq. (4), by the finite volume method, using the ANSYS CFX™ software, in order to describe the airflow, enabling the analysis of said flow.

$$\frac{\partial \rho}{\partial t} + \text{div}(\rho \mathbf{u}) = 0 \quad (1)$$

$$\frac{\partial(\rho u)}{\partial t} + \text{div}(\rho u \mathbf{u}) = -\frac{\partial \rho}{\partial x} + \text{div}[\mu \text{grad}(u)] + S_{Mx} \quad (2)$$

$$\frac{\partial(\rho v)}{\partial t} + \text{div}(\rho v \mathbf{u}) = -\frac{\partial \rho}{\partial y} + \text{div}[\mu \text{grad}(v)] + S_{My} \quad (3)$$

$$\frac{\partial(\rho w)}{\partial t} + \text{div}(\rho w \mathbf{u}) = -\frac{\partial \rho}{\partial z} + \text{div}[\mu \text{grad}(w)] + S_{Mz} \quad (4)$$

where ρ stands for specific mass; p stands for pressure; t stands for time; x , y and z are the three Cartesian coordinates; u , v and w are, respectively, the velocities in the x , y and z directions; \mathbf{u} stands for the three-dimensional velocity vector, μ is the fluid viscosity and S_{Mx} , S_{My} , and S_{Mz} are the momentum sources terms in the directions expressed in the subscripts.

2.2 Mathematical and physical validation

In order to validate the initial conditions of the simulation and so guarantee the correspondence to the natural phenomenon, it was necessary compare the results produced by the simulation to one produced by Abe and Ohya (2004). Therefore, it was compared three different turbulence models, two different meshes and two different values of the Reynolds numbers, in order to determine the most precise conditions to be used in the subsequent analysis of the diffusers.

The parsing of said conditions was based on the comparison of the behavior of velocity and pressure in during the flow through the diffuser. The velocity was compared by the graph between the relative position along the x -axis, turned dimensionless by dividing by the diameter of the diffuser inlet, and the velocity, turned dimensionless by dividing by the initial velocity (5 m/s). Moreover, pressure behavior was compared, by the graph between said position variable and the dimensionless pressure, obtained from the pressure coefficient (C_p).

At this point, the models of turbulence used, were the k - ϵ , RNG k - ϵ and SST (Shear Stress Transport). The first two are very similar to the one use by Abe and Ohya (2004), whereas the last one was considered the most adequate to an evaluation of the regions nearest to the walls of the experiment.

In addition, the boundary conditions set were based on symmetries, walls, inlet velocity, and relative pressures at the outlet. In the front and back planes, symmetry was admitted, whereas at the top, it was admitted a free slip wall and, in the diffuser, a no slip wall. For the inlet, it was set an entry speed of 5 m/s and in the outlet; it was defined by the use of a relative pressure of 0 Pa with the inlet.

To complete this part of the study it was analyzed the influence of the Reynolds number in the results by comparing two different values for such measure of the turbulence of the flow. The value of 64,718.7 which came from the data defined and the value of 20,000, set by adopting an entry speed of 1.54 m/s, in order to get an identical value to which adopted by Abe and Ohya (2004).

The geometry chosen for this study was of a flanged diffuser, with the same characteristics adopted by Abe and Ohya (2004), as shown in Fig. 1.

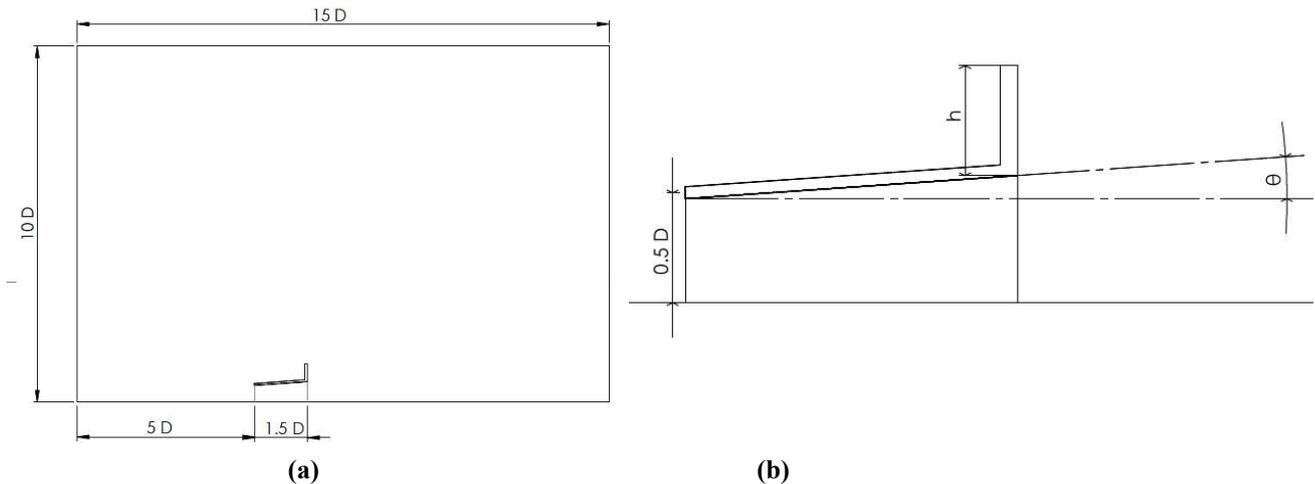


Figure 1. Geometry used in the mathematical and physical validation

The geometry was constructed in the SolidWorksTM software, using the diameter of the diffuser inlet (D), set as 0.2 m, as the basis for the construction of the part. In addition, h , was defined as the height of the diffuser flange, measured by its ratio with the cited diameter (h/d), set at this point as 0.5, and the angle between the diffuser and the x -axis was defined as theta (θ) and set at this point as 4° . The geometry was made three-dimensional by the revolution around the bottom edge with an angle of 0.5° along the middle plane.

Based on this geometry, two different meshes were constructed, by dividing the model previously constructed in finite control volumes through the definitions of a set of conditions of points around which the volumes would be created. To enhance the precision of the study, the limit of one element in width was adopted, as well as an increase in the refinement of the meshes around the diffuser and the bottom edge, responsible for the representation of the radial symmetry of the system. This resulted in two different meshes, the first (mesh 1) with 17,987 elements and the second (mesh 2), more refined, with 176,386 elements, in order to progressively enhance the precision of the study as well as analyze the impact of the mesh in the experiment. The second mesh, therefore, had an increase in refinement in the diffuser and bottom edges using elements of 2 mm in size, as well as an inflation in the diffuser edges with the smaller element having a size of 0.63 mm as shown in Fig. 2.

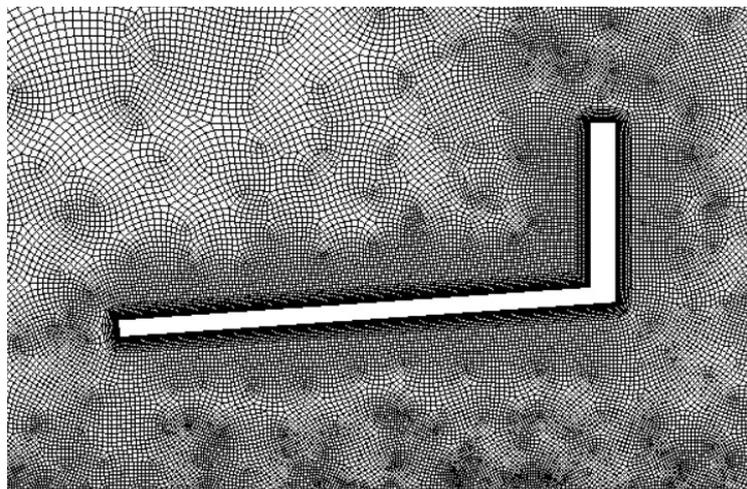


Figure 2. Mesh 2 in the vicinities of the diffuser

2.3 Diffuser evaluation

Using the conditions for the simulations defined in the validation as well as the same style of geometry and mesh, some parameters that affect the geometry of the diffuser, and consequently the characteristics of the airflow inside it, were evaluated. Therefore, it was chosen four different flange heights and four different angles to be simulated and compared in order to define the best combination of said factors between them.

For the flange height, once again defined by the ratio with the diameter of the diffuser inlet (h/d), it was chosen the values of 0, 0.125, 0.25 and 0.5 to be analyzed. As for the angles, it was picked four equally spaced angles in the interval between 0° and 15° , which resulted in the analysis of the angles of 0° , 5° , 10° and 15° .

These diffusers were compared using the profiles and contours of velocity and pressure, both turned dimensionless in the same way as in the validation, the maximum velocity ratio, along with its position in the x-axis, and the variation of said velocity ratio increase in the vertical plane at this point.

3. RESULTS AND DISCUSSION

3.1 Mathematical and physical validation

At this point, only the velocity ratio and pressure coefficient profiles were analyzed, shown in Fig.3, comparing to the results of Abe and Ohya (2004) in which the closest behavior was chosen as the more appropriated one.

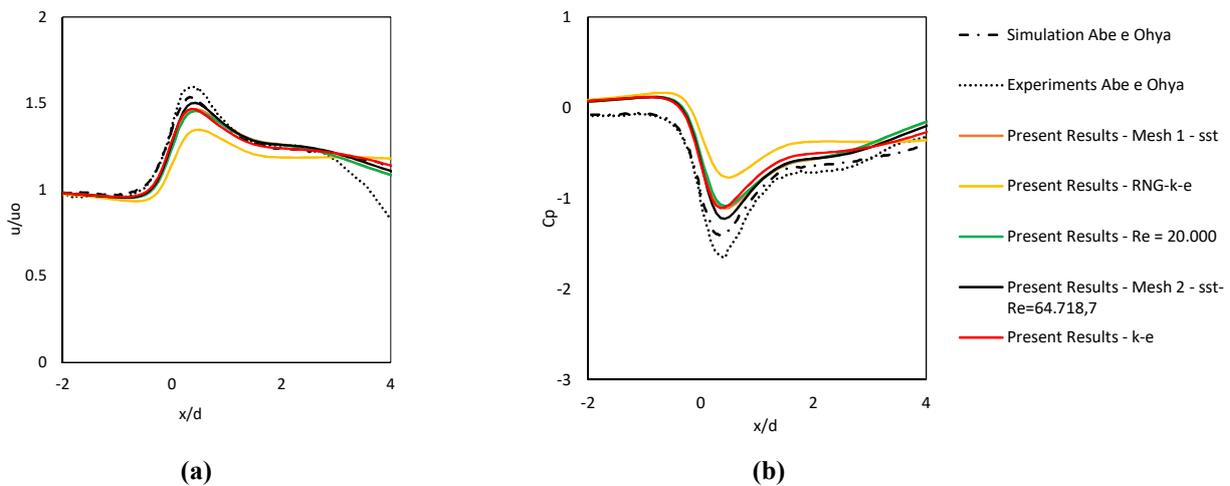


Figure 3. Velocity ratios (a) and pressure coefficient (b) profiles from the mathematical and physical validation simulations

Through the graphs, it is noticeable that all the simulations were relatively close to the desired behavior, the closest behavior to the one observed by Abe and Ohya (2004) being the one with SST as its turbulence model and the mesh 2, more refined, as its mesh. That indicates that this condition is the most adequate to be used in the next stages of this research as it is the one closest to the real behavior of the phenomenon.

In addition, it is clear that the value of the Reynolds number does not influence the airflow inside the diffuser, as predicted by Abe and Ohya (2004), shown in the graphs by the superposition of the profiles in which this was studied. The greatest difference was due to the turbulence model, in which the SST was shown to be the most precise, and the refinement of the mesh was shown to have little influence at the analysis, but still the closest were mesh 2.

It is important to note, that despite these were the closest conditions, the results did not end up being equal to the literature. This is more relevant when comparing the peak of the velocity graph, as this was the focus of the analysis of the subsequent simulations. Said difference was measured by the relative discrepancy at this point, both in the value of x/d and u/u_0 , being 8.4% and 5.8% respectively when comparing with the experimental results of Abe and Ohya (2004). Those values are considered acceptable to validate said simulations conditions, once there are known sources for them, such as the process of acquiring the data from Abe and Ohya (2004) as well as the difficulty of establishing some dimensions of the model used in their experiment.

3.2 Diffuser evaluation

The first item of comparison between the geometries studied were the profiles of the velocity ratio and pressure coefficient taken in the centerline of the diffuser, shown in Fig. 4, in the same way as studied in the validation. Each graph constructed was the comparison between the different angles for the same flange height value, being used to determine the optimal angle for each case.

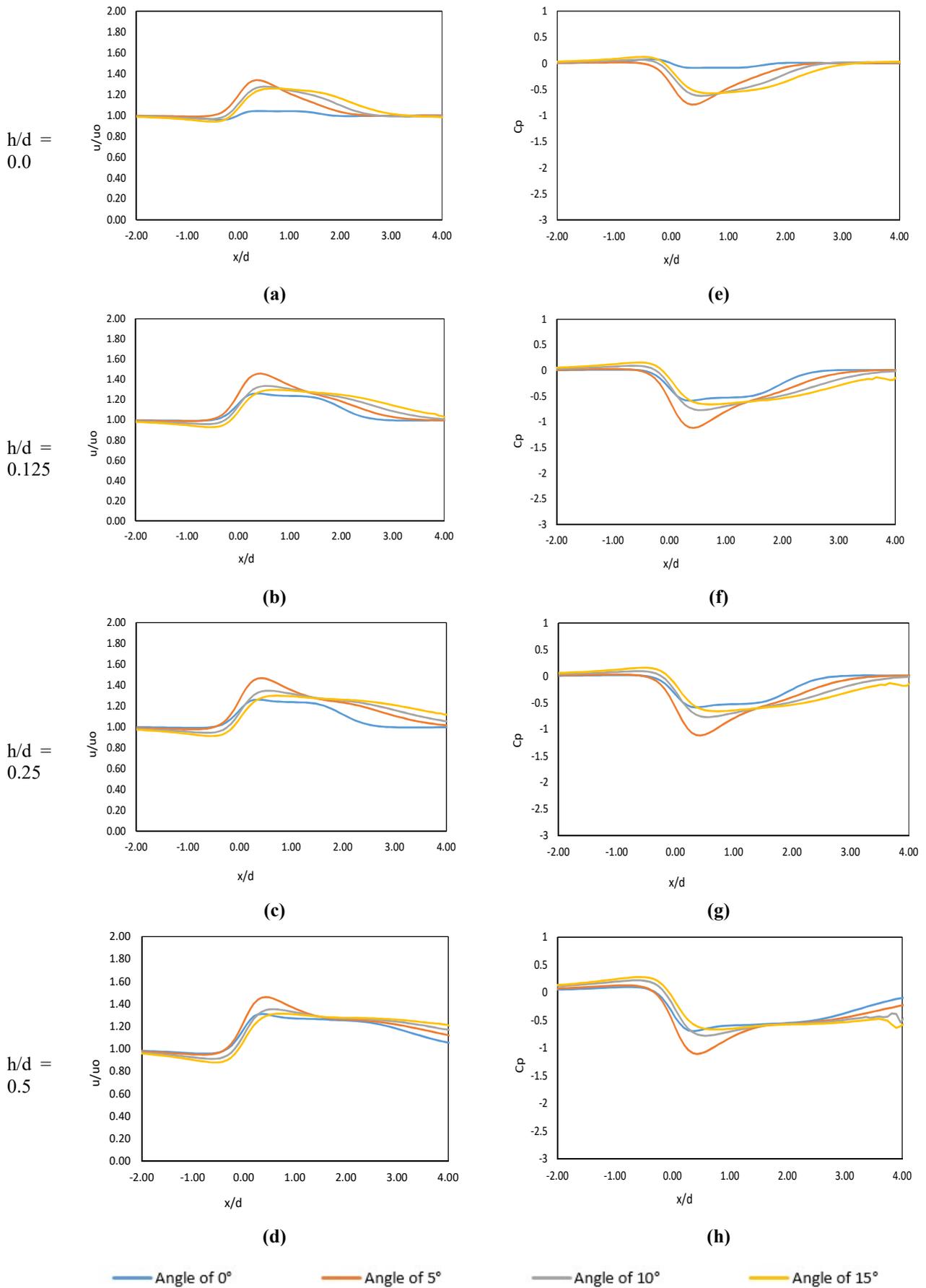


Figure 4. Profiles of velocity ratios (a - d) and pressure coefficient (e - h) along the x-axis

Analyzing the profiles shown in the Fig. 4 above, it is clear that the acceleration generated in each point is dependent of the position along the x-axis as well as the parameters initially studied. From those, it is evident that the profiles generated by using the angle of 5° are better, since the peaks are higher in every case, showing that the pressure reduction generated by the diffuser was greater, which in turn, generated a greater acceleration in the airflow through the diffuser. Between those, the variation between different flange height ratios appeared to have a smaller influence in the airflow, as the peaks appear to have similar height when comparing the results.

In addition to these graphs, the contours of velocity ratio for each diffuser was collected, as shown in Fig. 5 through Fig.8. These provided another way to visualize and so compare the influence of each diffuser on the airflow.

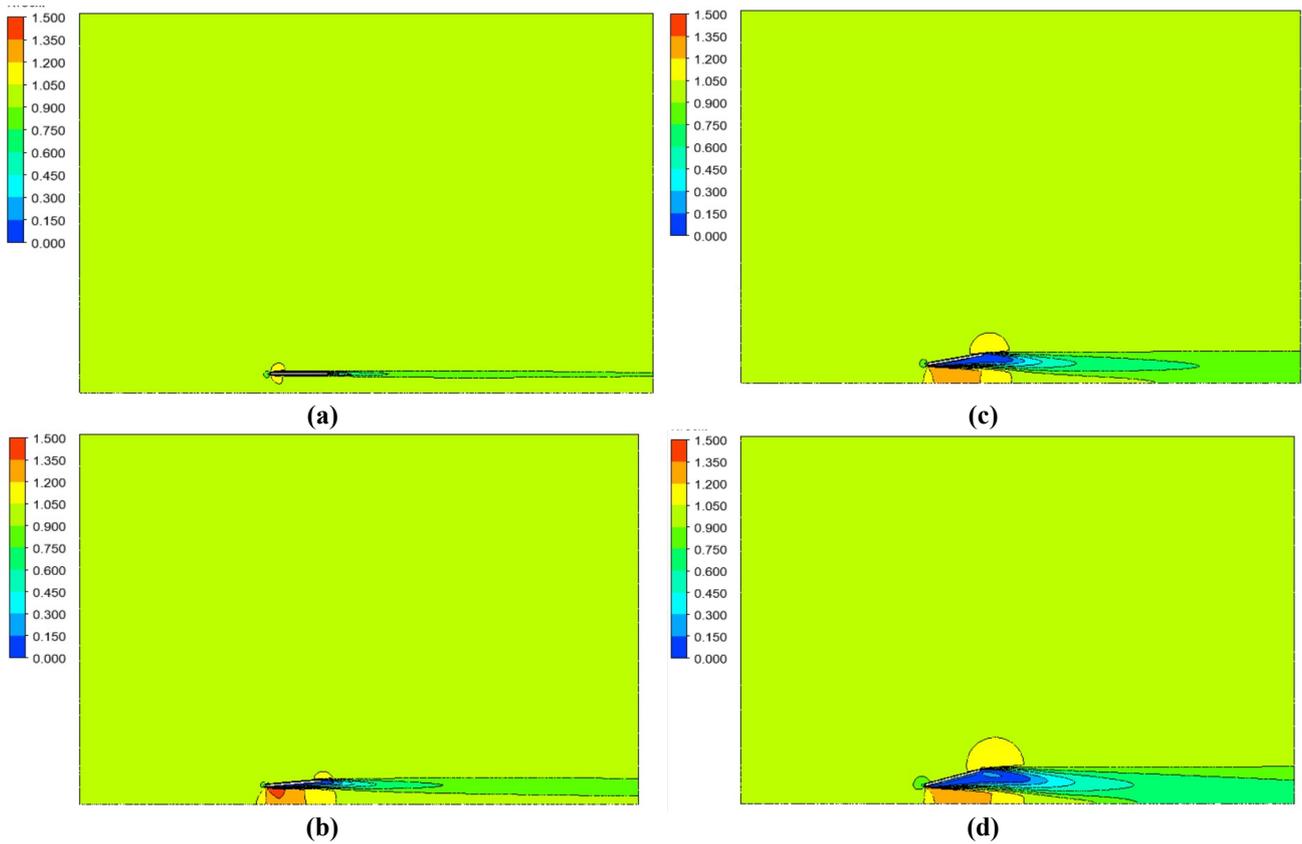
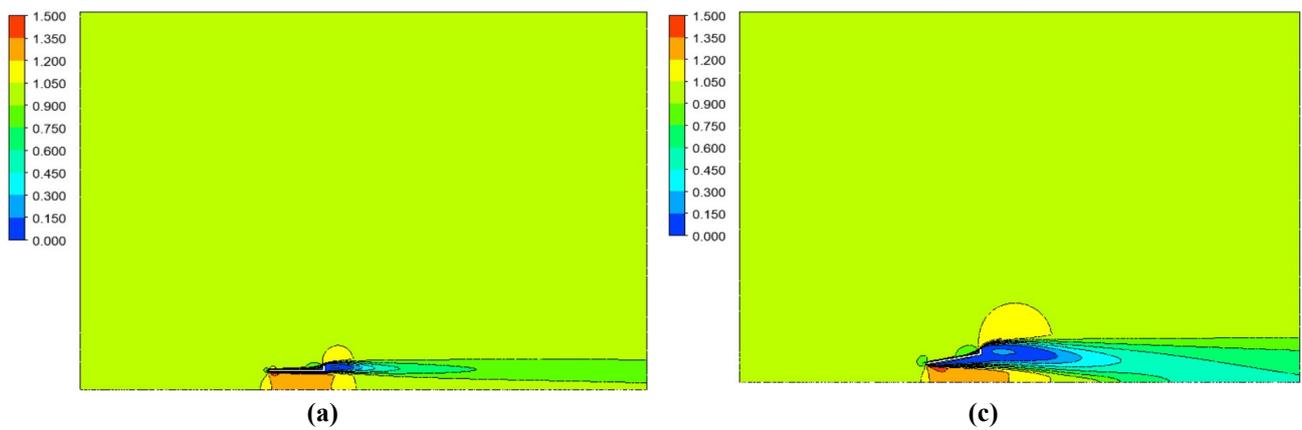


Figure 5. Contours of velocity ratio (a-d) for h/d ratio of 0



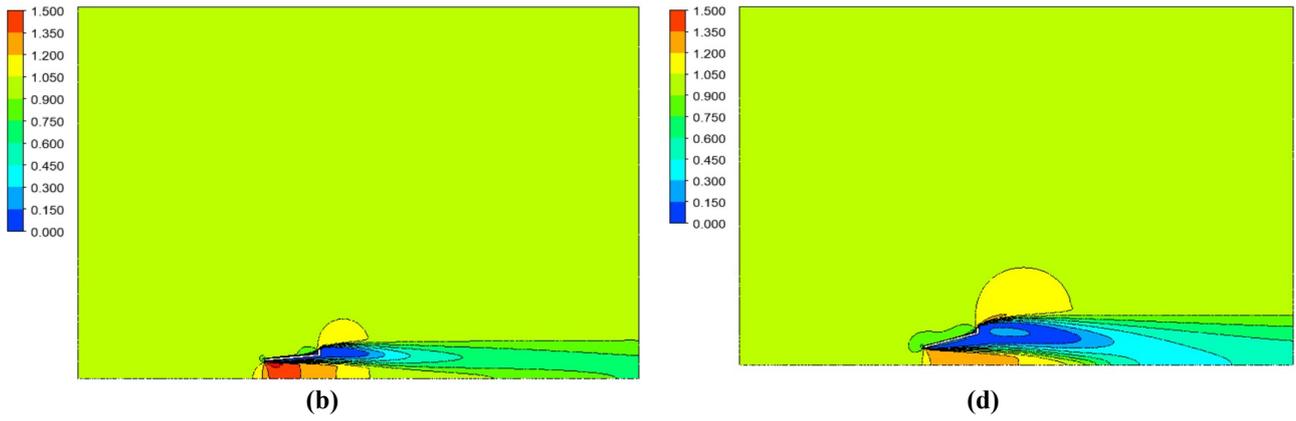


Figure 6. Contours of velocity ratio (a-d) for h/d ratio of 0.125

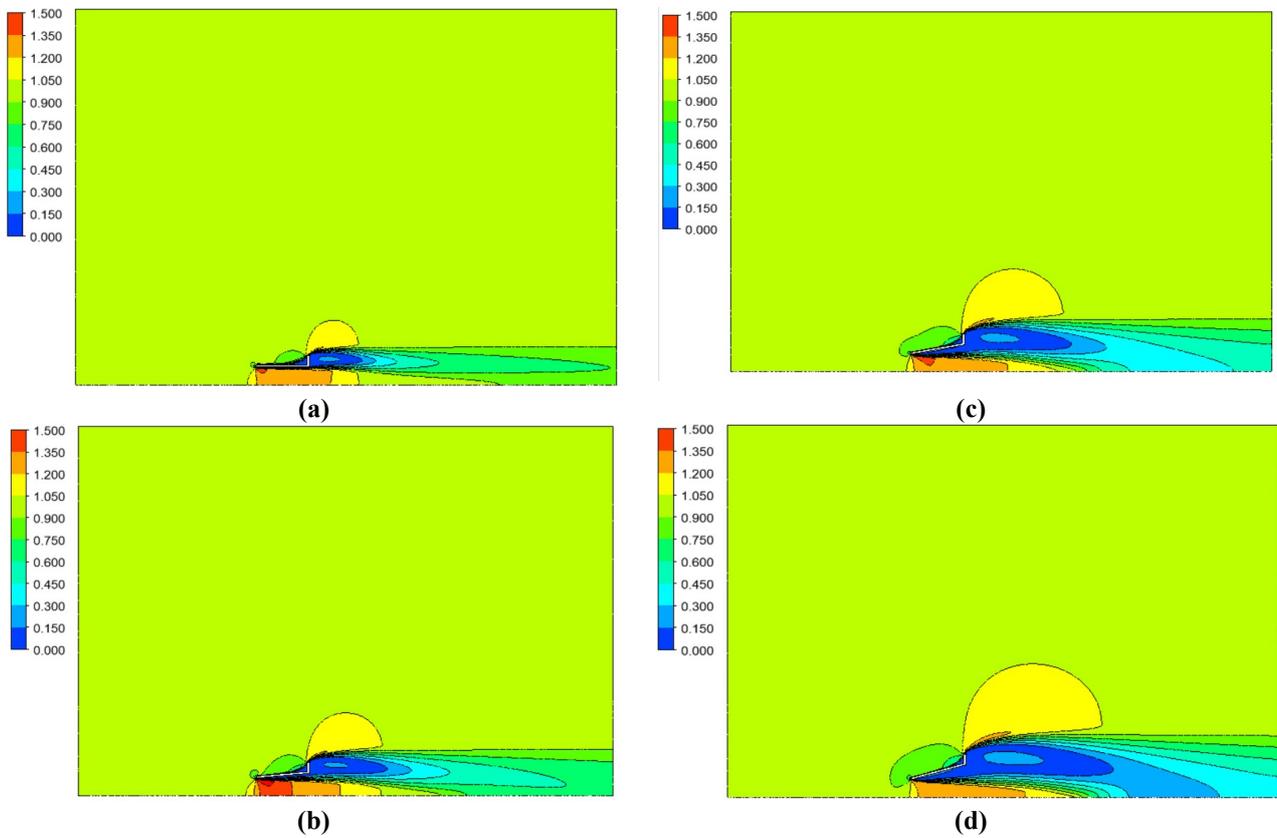
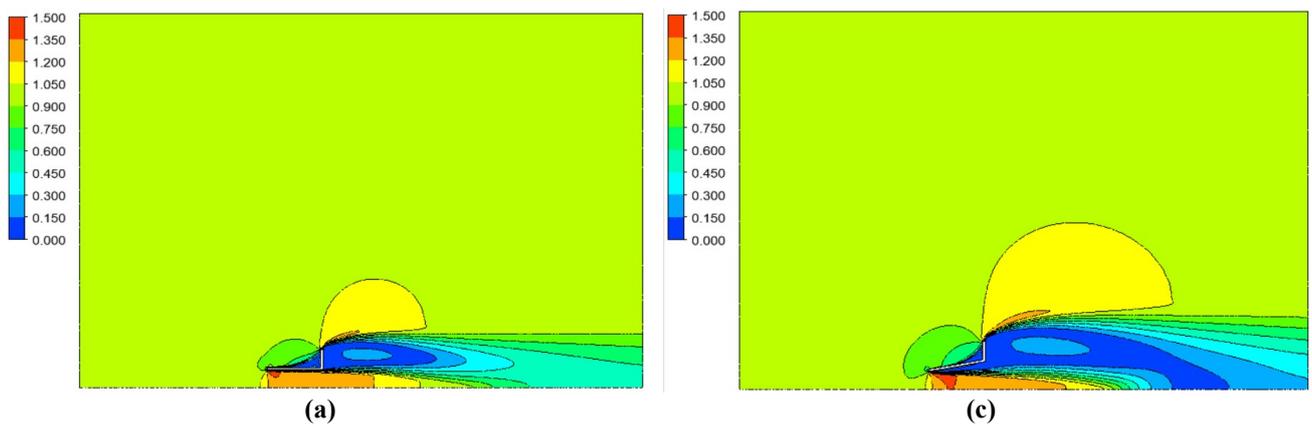


Figure 7. Contours of velocity ratio (a-d) for h/d ratio of 0.25



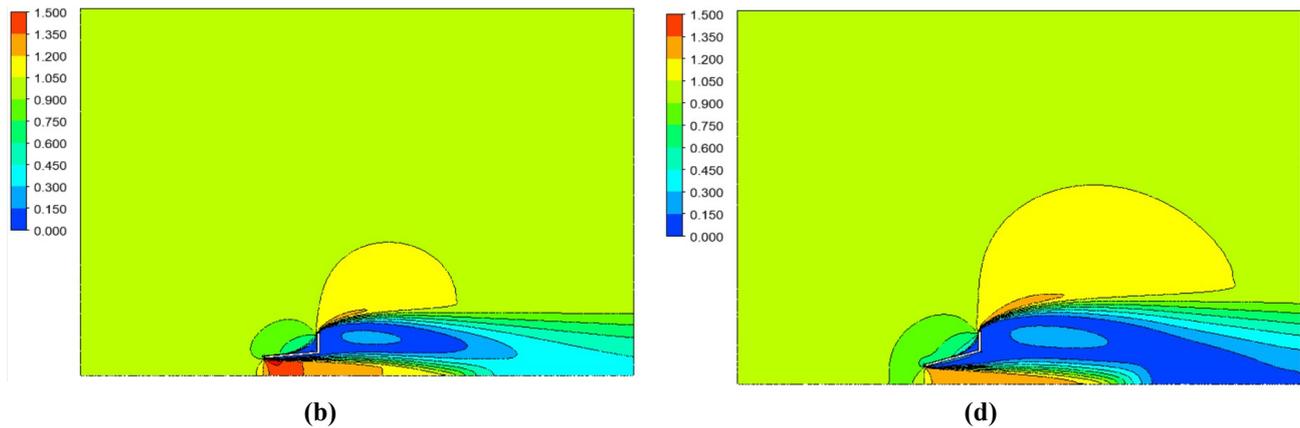


Figure 8. Contours of velocity ratio (a-d) for h/d ratio of 0.5

From the analysis of the contours, presented in Fig. 5 through Fig. 8, it is possible to further confirm what was shown in the velocity profiles shown in Fig. 4, as in the contours where the diffuser angle was 5° (images b in every Figure) the red area, which indicates the higher velocity ratio, is bigger in every case. Between those, it is noticeable that for the flange height ratio of 0.125, the region appears to be slightly bigger, suggesting that this ratio is slightly better than the other ratios.

Moreover, from these contours, it is made clear that the velocity ratio also varies in the vertical plane, which is in the direction of the radii of the diffuser, as stated by Abe and Ohya (2004). This is shown as for each position on the x-axis, there is a color variation present which indicates a variation in velocity between the lower limit of the geometry and the diffuser itself. From the analysis of said variation, it is noticeable that it consists in a reduction in the velocity of the airflow, thus reducing the effectiveness of the diffuser, mainly due to the boundary layer formed along the diffuser walls, but also due to an increase of the turbulence generated by a recirculation of the flow in these areas. Furthermore, it is shown that the region of smaller velocity (represented by the colors closer to blue in the contours), increases with the diffuser angle, suggesting an explanation for the better performance of the diffusers with this angle in the profiles, as discussed previously. Beyond that, these regions tend to extend after the diffuser with the increase in the flange height ratio, showing a much smaller influence in the flow inside the equipment.

In addition, in order to provide a more direct comparison between the sixteen cases studied, it was constructed two three-dimensional graphs. The first one, shown in Fig. 9, compiled the position in relation to the origin located at the diffuser inlet, along the x-axis of the maximum value of the velocity ratio, used to determine the optimal position for the rotor, in order to be able to extract the most amount of power. The second graph, shown in Fig.9, compiled the values of such highest values, in order to facilitate the optimal value.

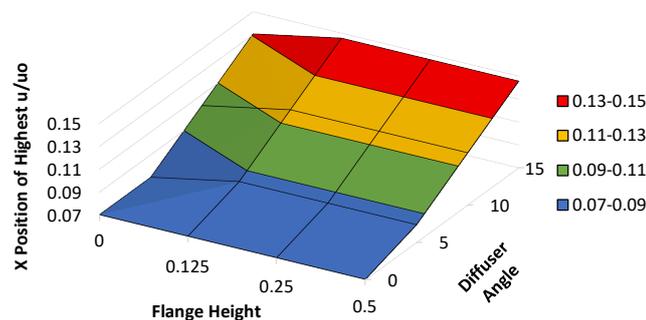


Figure 9. Position along the x-axis of the highest value of the velocity ratio as function of diffuser angle and flange height

The position of the rotor is most dependent of the angle of the diffuser than the height of the flange. This is easily noticeable in the Fig.9, by the equality in most values for the same angle, whereas having an amplitude of almost 50% between the lowest and highest angle value.

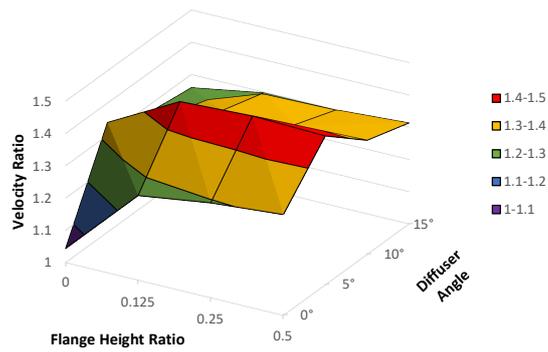


Figure 10. Highest value of the velocity ratio as function of diffuser angle and flange height.

Figure 10, therefore, shows clearly, that the angle of diffuser that provided the highest acceleration was 5°, and as for the flange height, the values for its ratio with the diameter of 0.125, 0.25 and 0.5 had very similar results, with the highest value being 1.47 for the ratio of 0.25.

In order to develop the study of the variation of the velocity ratio along the y-axis noted when analyzing the contours above, the profile of said ratio along the y-axis where compiled in the Fig. 11 presented below.

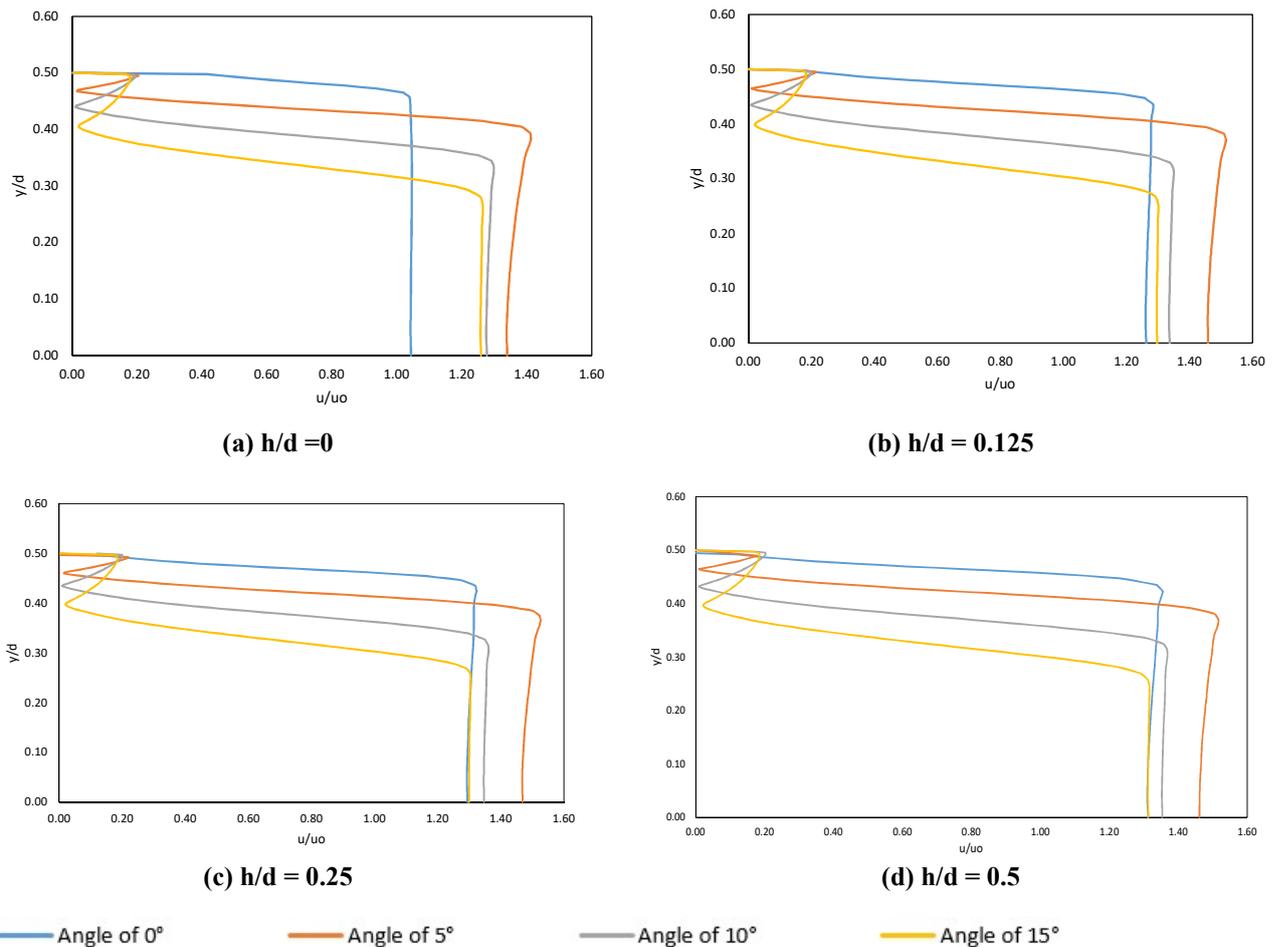


Figure 11. Velocity ratio profiles along the y – axis

Through the analysis of these profiles, it is clear that with the increase in the angle, the variation in the velocity profiles also increase. All the profiles present the expected constant reduction due to the boundary layer, however, by the end of the profiles, located near the diffuser wall, the velocity suddenly increased again, and soon decreased,

showing the existence of a recirculation in the airflow. This recirculation is prejudicial to the efficiency of the diffuser, as it is a form of turbulence, which consumes a portion of the wind's kinetic energy, which would otherwise be transformed into electric power. Said recirculation, while nonexistent for the angle of 0° , increases with the increase in the angle, shown by the increase of the height of the variation. Besides this, it is noticeable that the increase in velocity is similar between all the diffusers, except by those that have the angle of 5° , in which, such profiles have a higher value, shown by the distance of its curve from the rest.

From all this data it is possible to conclude that the most effective diffuser was the one with flange height of $0.25D$ and angle of 5° . This is made clear as it has the highest value for velocity ratio, demonstrating that the air was more accelerated when passing through, allowing more power to be extracted, shown by the highest peak in the profiles and the highest value in the three-dimensional graph, further confirmed by the profiles along the y-axis, while having the smallest recirculation. This confirms what Lipian et al. (2015) proposed in terms of flange height and the optimal angle.

4. FINAL CONSIDERATIONS

To sum up, it is possible to conclude that the present study was able to analyze the influence of the flange height and diffuser angle on the airflow in a wind turbine. It is important to note, however, that those are not the only parameters that affect such geometry, nor are the values analyzed the only ones possible. Therefore, in future studies it is sought to analyze some other values of the parameters studied near the optimal values defined at this point in order to refine the optimal values found. In parallel, it is sought to study other concepts of diffusers, such as the ones with an airfoil profile, in order to determine if the flanged diffuser is indeed the best diffuser to enhance the performance of wind turbines.

5. REFERENCES

- Abe, Ken-ichi, and Yuji Ohya. "An Investigation of Flow Fields around Flanged Diffusers Using CFD." *Journal of Wind Engineering and Industrial Aerodynamics*, vol. 92, no. 3-4, Mar. 2004, pp. 315–330., <https://doi.org/10.1016/j.jweia.2003.12.003>.
- ABEEÓLICA. *Boletim Anual de Geração Eólica 2020*. São Paulo: Associação Brasileira de Energia Eólica, 2021.
- Agha, Arouge, et al. "Diffuser Augmented Wind Turbine (DAWT) Technologies: A Review." *International Journal of Renewable Energy Research*, vol. 8, no. 3, Sept. 2018, pp. 1369–1385., <https://doi.org/10.20508/ijrer.v8i3.7794.g7436>.
- BURTON, Tony; SHARPE, David; JENKINS, Nick; BOSSANYI, Ervin. *Wind Energy Handbook*. Chichester: John Wiley & Sons, 2001.
- Dick, E. "Momentum Analysis of Wind Energy Concentrator Systems." *Energy Conversion and Management*, vol. 24, no. 1, Jan. 1984, pp. 19–25., [https://doi.org/10.1016/0196-8904\(84\)90047-5](https://doi.org/10.1016/0196-8904(84)90047-5).
- Hansen, M. O., et al. "Effect of Placing a Diffuser around a Wind Turbine." *Wind Energy*, vol. 3, no. 4, Oct. 2000, pp. 207–213., <https://doi.org/10.1002/we.37>.
- Lipian, Michał, et al. "Sensitivity Study of Diffuser Angle and Brim Height Parameters for the Design of 3 KW Diffuser Augmented Wind Turbine." *Open Engineering*, vol. 5, no. 1, 18 May 2015, pp. 280–286., <https://doi.org/10.1515/eng-2015-0034>.

6. RESPONSIBILITY NOTICE

The authors are the only responsible for the printed material included in this paper.