ERURJ **2024**, *3*, 2, 1007-1018

(Review)

Review of Improving the Performance of Horizontal Axis Wind Turbine Using Passive Flow Control

Wafaa Mostafa[1, *]

[1] *Mechatronics Department, Faculty of Engineering, Egyptian Russian University, Egypt*

* Corresponding author: Wafaa Mostafa, E-mail: wafaa-mostafa@eru.edu.eg

Received 16th November 2023, revised 10th March 2024, Accepted 10th March 2024

DOI: 10.21608/ERURJ.2024.229504.1071

ABSTRACT

Controlling rotor blade flow separation and dynamic stall is required to improve the aerodynamic performance of horizontal axis wind turbine (HAWT) blades. Active and passive flow controls are the two basic methods for controlling the flow state. The use of passive and/or active flow management strategies to improve the aerodynamic performance of the blade is common. Many publications are currently being conducted to determine the impact of various passive and active strategies on the aerodynamics of blades. Active flow control is limited since it requires an extra controller, actuators, and power source to control the flow. Passive flow control, on the other hand, is more straightforward and less expensive. As a result, passive flow control is more beneficial in real-world applications, but only under certain conditions. Researchers are developing improved designs for HAWT blades, the main component of the turbine responsible for power generation. CFD is a versatile and powerful tool for simulating and analyzing airflow over wind turbines. It can provide detailed information about the flow field around the turbine, including velocity, pressure, and turbulence. This information can be used to understand the aerodynamic forces acting on the turbine and to identify areas for improvement. CFD has been used to develop a variety of design and optimization strategies for wind turbines, leading to significant improvements in aerodynamic performance.

Keywords: Power improvement; HAWT Wind turbine blade; Passive flow

1. **Introduction**

There is a great interest in using renewable energy as an alternative energy source. Recently, the globe has begun to transition away from chemicals derived from petroleum, which are harmful to the environment, and moving more toward renewable energy sources. There are different resources of renewable energy like solar, hydro, geothermal, hydrogen storage, biomass, and wind energies. Wind power is predicted to have a significant share of electricity generation in the coming years. The wind will provide more than a third of all energy by 2050, according to overall electricity demand [1], [2]. Wind energy has numerous advantages, which is why it is one of the world's most rapidly expanding sources of power and the most promising renewable source because it is reliable, clean, and has a minimal carbon footprint. In 2021, an estimated 102 gigawatts (GW) of wind generating capacity were assumed added globally, comprising more than 83 GW onshore and about 19 GW offshore. Total additions increased by 7% compared to 2020, reaching their greatest level yet, with yearly offshore installations nearly double their previous peak. By the conclusion of the year, total global wind power capacity had risen 13.5% over 2020 to exceed 845 GW (791 GW onshore and the rest offshore). Wind power capacity in operation worldwide is expected to provide 7% of total energy output in 2021 [3].

According to the International Renewable Energy Agency (IRENA), global renewable energy (GRE) capacity reached 2537 GW by the end of 2019. Providing about 34.7% of the Net Global Energy (NGE) capacity and wind energy contributed about 2.5%. Global wind power increased by 19% with a new capacity of 60 GW in 2019 [4]. The Global Wind Energy Council (GWEC) [5] claims that, in 2020, New wind power installations in the world are approaching 90 GW, an increase of 53% over 2019. Egypt is the leading country in Africa for new wind power installations by 262 MW. Egypt works on the first single privately owned wind farm and simultaneously has a short-term target of 7 GW by 2022 [4]. According to the European Wind Energy Association (EWEA), wind power could produce more than 2,500 TWh by 2050 which will present about 50% of Europe’s expected electricity demand [6]. The two typical designs for wind turbines are HAWT and VAWT. Vertical-axis wind turbines (VAWT) represent a small portion of the wind turbine market today due to their inferior wind power conversion [7].

2. **Active and passive wind turbine control**

Based on the scope of flow modifications and energy expenditure, flow-control devices can be broadly classified as passive or active. To accomplish the desired impact, Active control techniques include the regulation of flow fields in specific sections of the wind turbine blade. Active flow control relies on external power to alter the airflow around an aerodynamic body, such as a wind turbine blade. The energy savings achieved through this method must outweigh the energy required to operate the actuators. Active flow control systems often incorporate moving parts that are integrated into the existing blade structure. These components are activated or deployed based on specific criteria. In recent years, there has been significant progress in adopting active flow control systems for wind turbines, primarily for the purpose of vibration control, which addresses issues arising from unstable aerodynamic blade loads. This is achieved by adjusting the airflow locally along the blade. Active flow control devices, such as flaps, flexible T.E. flaps, flexible L.E. flaps, split flaps, stall ribs, spoilers, and boundary layer suction and blowing, have also been extensively employed in aeronautical applications and hence have the potential to have a comparable influence on the wind power sector.

The effectiveness and energy efficiency of passive control systems have been evaluated by numerous researchers using a variety of numerical and experimental techniques (vortex generators, micro tabs, fixed leading-edge slat, flow vane, L.E. protuberances). The flow characteristics in the boundary layer are changed in passive techniques to control the flow over an airfoil. By manipulating the separated shear layer above the blade, passive flow control methods can effectively regulate the airflow. These methods achieve flow control without requiring external energy or additional power input by employing a combination of high and low-momentum fluid particles. Passive geometric modifications that regulate the pressure gradient, fixed mechanical devices that manage flow separation, and devices on the flow surface that aid in drag reduction are all examples of how this is accomplished. During normal operation, these devices usually have fixed positions or geometrical orientations that are not controllable. Active flow control systems employ external energy inputs to manipulate the airflow and can dynamically adjust their configuration during operation. In contrast, passive flow control systems consist of fixed components attached to the airfoil frame, eliminating the need for external energy and reducing maintenance requirements. Due to their cost-effectiveness and technological feasibility, various aerodynamic experts have promoted the use of passive flow control systems.

3. Aerodynamic analysis over a 2D airfoil

Kumar et al. [8] compared the aerodynamic performance of **NACA 4412** with **S809** for wind turbine purposes [8]. An evaluation of various turbulence models (Reazible, k-ε, Spalart Allamaras, and k-ω (SST) Shear stress transport) against experimental data revealed the **k-ω** shear stress transport (SST) model to be the most accurate. The simulation was conducted at $\mathbf{Re=10^6}$ at different angles of attack. The lift and drag coefficient are compared for both airfoils and the lift coefficient of NACA 4412 is higher than S809 which leads to selecting NACA 4412 for wind turbine applications.

Researchers are investigating the usage of micro-cylinders around airfoil sections or wind turbine blades to improve aerodynamic performance. At a $Re= 6x10^6$, Luo et al. [9] performed numerical research to analyze the aerodynamic performance of a stalled NACA 0012 airfoil with and without a micro-cylinder placed at the leading edge. To find the best control parameter, Reynolds-averaged Navier-Stokes equations (RANS) simulations were used. Furthermore, Delayed Detached Eddy Simulations (DDES) were used to capture transient vortical patterns inside the large separation zone. The impact of various micro-cylinder diameters and placements was investigated. Overall, the addition of micro-cylinders resulted in an improved lift coefficient and lift-to-drag ratio following stall. The micro-cylinder's position had a significant influence on the total aerodynamic force generated. As demonstrated by DDES and RANS, the addition of a micro-cylinder late stall and reduced the size of the separation region. Of the 15 micro cylinder locations examined in this study, the Placement of micro cylinders near the pressure surface of the airfoil was found to be ineffective for stall control. When positioned near the suction surface, the optimal spacing between the micro cylinder and the airfoil should be increased to maximize the aerodynamic benefits.

Shi et al. [10] presented a computational investigation of inserting a vibrating micro-cylinder adjacent to the stalled airfoil (**S809**) at angles of attack ranging from 10^o to 24^o at $\mathbf{Re = 10^6}$. When compared to static micro-cylinders and original airfoils, vibrating micro-cylinders had a greater influence on total aerodynamic forces. When compared to the original airfoil, an optimal oscillation approach raised the **lift-to-drag ratio** by **88.21** percent.

Xie et al. [11] proposed creating a slot in the S809 airfoil. This slot was separated through the airfoil's body and spanned the length of the airfoil with a width-to-chord ratio of 0.01. The pressure side of the airfoil had one end of the slot's cross-section located at 10% of the chord, and the suction side had the other end situated at 60% of the chord. The pressure differential between the slot's two ends creates a jet that energizes the boundary layer flow. The investigation was conducted at angles of attack of 0^o, 10^o, 15^o, and 20^o. Due to the slot's jet, the detachment region split into tiny vortices at 20^o. At angles of 15^o and 20^o, where drag was reduced and lift increased, the slot's effect was visible. This slot has not much effect at angles of attack less than 10^o.

Vortex generators (VG) and vortilons are stationary aerodynamic devices that serve as passive flow control elements. These devices were conceived and brought to fruition by [12], [13]. Vortex generators and vortilons are both technologies that function on the same fundamental idea. These attachments are intended to create vortices that improve the downstream free flow's mixing. The flow stays bonded to the airfoil's surface for an extended amount of time due to the re-energization of the boundary layer. These objects come in a variety of designs and are made up of thin vertical plates that are normally mounted on the wing in their most basic form.

Outlined a CFD approach for dealing with a geometrically resolute VG on an airfoil segment The **FFA-W3-301** and **FFA-W3-360** airfoils were used as study items [14]. The calculated findings were compared to the results of wind tunnel studies. Thirty percent thick FFA-W3-301 airfoil was examined through the VG element positioned with its leading edge at x/c = 0.15, and x/c = 0.20, respectively. For all simulations, at $Re = 3 *10^6$. A one-degree delay in the stalling angle can be seen when the VG is moved closer to the leading edge when comparing cases with the VG positioned at 15% chord with those with the VG positioned at 20% chordwise. Both calculated configurations provide better lift performance than the clean design, but at lower angles of attack, they result in somewhat greater drag.

Wang et al. [15] studied the impact of vortex generators on the S809 airfoil's aerodynamic performance when used in wind turbines. The vortex generators created isolated vortices. The boundary layer involves the high momentum fluid of the main area, while the detached vortices involve the low momentum fluid of the boundary layer in the main region. As a result, the vortex generators efficiently regulate flow separation. The stall angle of attack was raised from 14^o to 18^o by installing vortex generators at 10% of the chord. It was discovered that adding vortex generators decreases the boundary layer's thickness, lowering the drag coefficient. Higher angles of attack improve the lift coefficient while adding double vortex generator configurations decreased the boundary layer thickness. The two vortex generators boost wind turbine output power by 96.48%.

The NACA 23024 airfoil was subjected to a CFD analysis in which the coefficients C_L/C_D ratio were computed and compared at 14.25 m/s intake airflow velocity with and without the Vortex Generator (airfoil) [16]. The investigation was conducted at angles of attack of 0^o, 0^o, 6^o, and 12^o and Reynolds number 10^6. According to the research, the Vortex Generator airfoil had better drag (C_D) and lift coefficient (C_L), as well as delayed flow separation over the airfoil.

Madani et al [17] described a laboratory optimization of a NACA 4415 airfoil fitted with vortex-generating devices (VGs) to control separation of flow. The following geometrical characteristics were suggested: vortex generator thickness and height, position, orientation angle concerning the mean flow direction, and span wise spacing. A unique arrangement with micro generators hidden below the regular units was studied. Wind tunnel studies were performed in comparison in all of these examples at various angles of attack and Reynolds numbers up to 2×10^5. The findings revealed that triangular vortex generators are the most beneficial for managing separation of boundary layers. An ideal VGs angle of 12° was achieved with a 3 mm gap between vortex generators situated at 50% of the chord. Micro vortex generators were extremely successful at regulating flow with minimum parasitic drag. A coupled vortex generator airfoil's C_{Lmax} is raised by 21%, while the separation of flows is 17° pushed back. Fortunately, the presence of parasitic drag offsets this good performance.

A rod controlling the flow of an SD7062 airfoil was presented in the work of Durhasan [18], utilizing the particle image velocimetry (PIV) technique at pre-stall angles of attack at Reynolds number Re = 30000. The rod was inserted at several chord wise points on the suction surface of the airfoil. The rod diameter '*d*' was standardized to the airfoil chord length *'c'*, and three diameter ratios (d /c = 0.017, 0.033, and 0.044) were investigated at angles of attack of = 6°, 8°, and 10°. The creation of a laminar separation bubble for the baseline airfoil, as well as the influence of the rod, was thoroughly explored. It was discovered that optimal rod position and diameter ratio lowered the height of the boundary layer by 22%. Furthermore, the rod considerably reduced the unstable vortices over the suction surface of the airfoil. As a result, the rod reduced turbulent statistics' maximum values by as much as 30%. The improvement in aerodynamic characteristics is shown in table 1

Table 1: Improvement comparison between different 2D studies

Passive flow control type	**Researchers**	**Improvement %**
Micro-cylinders	Luo et al. [9]	Heavy stall is significantly delayed by 2^o by placing MC on suction side (NACA 0012, Re= $6x10^6$).
Micro-cylinders	Shi et al. [10]	Lift-to-drag ratio by 88.21% (S809, Re = 10^6).
Vortex generators	Wang et al. [15]	The stall angle of attack was raised from 14^o to 18^o by installing vortex generators (S809).
Vortex generators	Madani et al [17]	Maximum lift coefficient is raised by 21% (NACA 4415).

4. **The effects of passive flow regulation on the aerodynamic efficiency of a 3D blade**

The effect of various passive techniques on wind turbine blade aerodynamic performance has been studied in several publications, for example installing micro cylinder, micro tab, vortex generators, gurney flaps, tubercle leading edge, split blade, slot, and rod vortex generators.
The result of inserting micro-cylinders with varying sizes and placements in the vicinity of the wind turbine blades of the NREL Phase VI was examined in detail by Wang et al. [19]. It was discovered that (1) it is possible to reducing separation of flow on the blades of wind turbines, (2) since the micro-cylinder is situated close to the blade's leading edge point, with a small vertical distance = 2.3×10^{-2} and diameter = 6.78×10^{-3}, (In this work, dimensions have been normalized by chord), the blade torque is improved by 27.3 %, and (3) The main improvement in flow separation was obtained using velocity streamlines in the region more than 60% of blade span.

Tahani et al. [20] used a new linearization approach to attain the advantages of linear chords and twist angles distribution to simplify the manufacturing process. This method is applied to 1 MW HAWT designed by blade element momentum theory. To investigate the influence of chord and twist linearization, the airfoil families Du 93-W-210, Risø-A1-21, S814, S809, and Risø-A1- 15 18 were chosen. When adopting the airfoils DU 93-W-210 and S809, the power coefficients change vary less. Employing Risø A1-18 in the blade design also results in a maximum power coefficient when compared to other airfoils.
Winglets help in the divergence of blade tip vortices and the reduction of produced drag. Kaya et al. [21] evaluate the aerodynamic characteristics of HAWT with forward and backward blades. Wind turbines with four sweep start positions and four offset values are subjected to CFD simulations. Forward swept blades with smaller sweep start up sections and larger tip offset values produce more power overall. The wind turbine with the forward swept blade showed the greatest power performance gain, with a 2.9 percent rise in the power coefficient. The backward swept blade produces the greatest reduction in the thrust coefficient to summarize, forward swept blades can boost performance whereas backward swept blades can reduce the thrust coefficient.
Another proposed improvement is the use of winglets (WLs) to increase the efficiency of HAWT. The WLs have recently gained popularity (HAWT). Diverging vortices at the tips of turbine blades and decreasing generated drag, the WL geometry is a critical parameter. The WL height and Toe angle are the two key geometrical characteristics which were investigated by Mourad et al. [22], while the Cant angle is kept constant in all numerical situations at 90 degrees. In their work, the performance of a three-bladed rotor with a 1m diameter and SD8000 airfoil was numerically evaluated on a polyhedral mesh using ANSYS 17.2 CFD. The upwind WL (positive toe angle) drove the tip vortex center away from the blade tip, increasing the power coefficient, whilst the down-wind WL (negative toe angle) retained the vortex center near the blade tip, lowering it.
Also, Elfarra et al. [23] studied different winglet configurations, and the power generation results were contrasted to those of the original blade NREL VI. When comparing the various configurations, Wind turbine blades with winglets added and oriented toward the suction side of the blade generate more power than those without them. Even with this suction side configuration, varied power outputs are produced by varying the cant and twist angles of the winglet. According

to the findings, a wind turbine blade's tip vortex properties and downwash distribution at the tip region are altered by the addition of a winglet, hence enhancing power generation. A winglet boosts not just power but also axial thrust. However, the gain in thrust is minor in comparison to the rise in power, which is particularly noticeable at high wind speeds.

Experimental analysis and numerical simulations by Rawad [24] were performed to emphasize the significance of the VG shape on the aerodynamic characteristics of the wing. This research begins by confirming the numerical technique using experimental data. Following that, a trapezoidal VG is proposed and tried in this work with different parameters, and the influence on overall performance is demonstrated. The outcomes might be summed up by emphasizing the significance of well-designed vortex generators in improving aerodynamic performance. At the commencement of, the stall, the suggested non-conventional trapezoidal VG exhibits a 21% increase in lift over drag on the airfoil. During stall, the trapezoidal VG enhancement is also observed, with the lift over drag ratio increasing by 120 % for the airfoil and by 10 % for the triangle winglets recorded in the literature. It was discovered that the height of the VG should not surpass the thickness of the boundary layer.

The vortex properties VGs and the airflow properties of a plate and an airfoil were explored by Xinkai et al. [25], using both wind tunnel tests and computational methods to investigate the influence of vortex generators (VGs) height on boundary-layer flow's controlling impact. On a flat plate boundary layer, the ratio of VG height (H) to boundary layer thickness (δ) was investigated first; H values range from 0.1δ to 2.0δ. The findings show a logarithmic connection between concentrated vortex strength and VG height, with vortex intensity corresponding to the fluid's average kinetic energy in the VG height range. Second, using three VGs with H = 0.66δ, 1.0δ, and 1.33δ, in a wind tunnel, the impact of height on the aerodynamic performance of airfoils was examined. The airfoil's stall angle with and without VGs is 18^o and 8^o, respectively, meaning the VGs raise the stall angle by 10 degrees. The VGs have no effect on the airfoil's maximum lift-drag ratio since the airfoil with VGs has a lower maximum lift-drag ratio than an airfoil without VGs. Conversely, a VG increases the ideal lift-drag ratio's angle of attack.

Significant literature exists on the subject in terms of both experimental and computational modeling simulation research. The goal of these investigations was to measure performance improvement and identify The method of flow regulation in tubercles at the forefront [26]–[28]. According to research, the unusual location of leading-edge tubercles on humpback whale flippers serves as an increased lift mechanism, controlling flow across the flipper and maintaining lift at a high AoA. Similarly, leading-edge tubercles used in lifting body design have been found to provide aerodynamic benefits by improving stall characteristics. For the last decade, cutting-edge alterations featuring tubercles have been the focus of study due to their flow-control capability. The pectoral flippers of humpback whales have a rough, bumpy leading edge made up of tubercles. The edge of pectoral flippers resembles a sinusoidal curve, with a peak-to-valley amplitude ranging from 5% to 24% of the flipper's chord length. When seeking to grab prey, the whale's flipper tubercles allow it to make abrupt bends without losing lift [27].

Miklosovic et al. [29] found that leading-edge tubercles on a model humpback whale flipper increased the stall angle by 40% and improved lift and drag performance. These results suggest that tubercles may help humpback whales maneuver more efficiently. The optimum model scale (NACA 0020) of the artificial humpback whale flipper with and without tubercles were used to produce experimental results at Reynolds numbers that varied from 505,000 to 520,000.

At Reynolds number 183,000, Johari et al. [30] examined the influence of tubercle wavelength and amplitude on separation of flow for an improved NACA 634-021 airfoil. They demonstrated that, when tubercles were present, they typically reduce lift and raise drag for small angles of attack, it resulted in a 50% increase in lift over the standard airfoil, with no additional drag cost, at angles of attack greater than the standard stall angle. Test to see if two wings (NACA 0021 and NACA 65-021) with tiny bumps on the front would affect their performance. They believed that these bumps would function similarly to vortex generators, which are apparatuses described by Hansen et al. [31] that are used to enhance the performance of wings and other aerodynamic surfaces.

An experimental study by Zhang et al. [32] examined the impact of leading-edge protrusions on the aerodynamic properties of a NACA 634-021 airfoil at Reynolds number 200,000 for an extensive variety of angles of attack. The results showed that the airfoil with leading-edge tubercles avoided the abrupt stall that the baseline airfoil displayed by reducing performance in the pre-stall region and gradually decreasing lift coefficient for the post-stall behavior.

Leading flipper protuberances of aquatic creatures have been linked to improved hydrodynamic performance, as has been observed in nature. Huang et al. [33] are motivated to examine how leading-edge protuberances impact the aerodynamic performance of the blades in a small-scale HAWT system experimentally. The 3D static model and the rotor-blade model were both tested. The usage of leading-edge protuberances may be beneficial if the small-scale HAWT system is meant to function at low wind speeds, but not so much if the system is specifically built for high wind speeds.

Asli et al. [34] evaluated the influence of a bumpy leading edge on the S809 airfoil numerically. When compared to the baseline airfoil, the vortices were enlarged, resulting in a slight drop in lift force but an increase in flow momentum, which led to an increase in the attached flow area at high angles of attack. Because the separation zone on the suction side of the airfoil in the bumpy leading edge is larger than the baseline airfoil, the drag force is greater in all operating ranges studied.

The effect of applying specific surface modifications in the form of dimples on the aerodynamic properties of an airfoil is explored by Mashud et al. [35]. It is found when an airfoil is at a given angle of attack, dimples assist reduce pressure drag in the same way that a vortex generator does. Dimples cause turbulence in the flow, which slows the separation of the boundary layer and minimizes the wake, lowering the pressure drag. The flow separation delaying by using dimples on the suction side which leads to increasing lift by 13% and decreasing drag by 21.6%.

Computational fluid dynamics (CFD) analysis was done on the frontal section of a turbine blade with a NACA S814 profile by Aziz et al. [36]. By putting an array of dimpled patterns on the blade surface, the drag force has been lowered. Because the dimpled structures have more momentum than the laminar flow, they create a turbulent boundary layer flow on their surface, which decreases drag and modifies lift. The simulation findings are validated by experimental data obtained in a wind tunnel, which agree closely with the simulated results.

Ebrahimi and Movahhedi [37] investigated the micro tab effect on an NREL Phase VI horizontal axis wind turbine. The micro tab is located on the lower surface of the blade at 95% chord length. By examining four cases, the dimensions and locations along the span are altered. In every scenario, power is enhanced. It is more effective to locate a micro tab outboard than an inboard. For sub-rated wind speeds, the optimistic result conserves 17% of the lost wind energy of the baseline blade.

Mostafa et al. [38] studied the optimization of micro-cylinder diameter and location around the S809 airfoil using the Response Surface Method (RSM). Maximum values of lift-to-drag ratio are around an angle of attack of 6.65^o. Lift-to-drag ratio was increased at Reynolds number above 5×10^6 in the range of angle of attack from 0^o to 12^o. It is discovered that placing the micro-cylinder in front of the airfoil leading edge has a greater impact.

Mostafa et al. look at the addition of micro-cylinder as a passive flow control surrounding the typical straight-bladed wind turbine from the National Renewable Energy Laboratory (NREL Phase II) [39]. The current computational study uses the Reynolds Average Navier-Stokes (RANS) equations for steady-state incompressible flow along with the Shear Stress Transport (k- SST) turbulence model. In this work, a parametric investigation is carried out for various micro-cylinder placements and sizes. Seven examples are looked at in all. By altering the cylinder diameter (three different sizes, diameter/chord = 0.0131, 0.0175, and 0.022), the impact of micro-cylinder size is investigated. It has been discovered that when the micro-cylinder diameter is reduced, the output power increases. Four distinct instances to evaluate the impact of micro-cylinder positions are introduced. In every situation, the power output rises. Furthermore, it has been discovered that placing the micro-cylinder in front of the blade leading edge has a greater impact on power output than placing it on the pressure side.

Table 2 illustrates the enhancement in aerodynamic properties for 3D blades.

Table 2: Improvement comparison between different 3D studies

Blade modification	Researchers	Improvement %
Micro-cylinders	Wang et al. [19]	NREL Phase VI, the blade torque is improved by 27.3 %.
Winglets	Kaya et al. [21]	The wind turbine with the forward swept blade 2.9 % increase in the power coefficient.
Leading-edge tubercles	Miklosovic et al. [29]	NACA 0020, Humpback whale flipper delays the stall angle by 40%.
	Johari et al. [30]	NACA 634-021, it resulted in a 50% gain in lift over the baseline airfoil.
Dimples	Mashud et al.[35]	lift is increased by 13% and drag is decreased by 21.6%.

Micro tab	Ebrahimi et al [37]	NREL Phase VI, 17% improvement in power output.
Micro-cylinders	Mostafa et al. [39]	NREL Phase II, 22% improvement in power output.

5. Conclusion

The HAWT blades' overall aerodynamic effectiveness has been successfully increased using the aforementioned passive flow regulation methods. Techniques for passive flow control are easier to use than active flow control systems because they don't need any extra energy sources in order to operate. It has been demonstrated that these tactics are incredibly economical, easy to implement, and low maintenance. Since they don't need any additional sources of energy to operate, passive flow control methods are simpler than active flow control systems. These tactics can be used with ease and require little upkeep, and they have been proved to be quite cost-effective. These strategies have improved C_L and C_D, increased AoA, enhanced the features of the stall, and postponed the flow's separation, according to a review analysis of numerous of these techniques.

6. References

[1] IRENA, IREA. "Future of wind: deployment, investment, technology, grid integration and socio-economic aspects." Abu Dhabi, 2019.
[2] W. Tong, Wind power generation and wind turbine design, WIT press, vol. 48, no. 07. 2011.
[3] R. E. N. Members, "Renewables 2022-global status report." ,2022.
[4] R. E. N. Members, "Renewables 2020-global status report." ,2020.
[5] J. Lee and F. Zhao, “GWEC global wind report 2021,” Global wind energy council: Bonn, Germany, p. 75, 2021
[6] European Wind Energy Association, “Pure power: wind energy targets for 2020 and 2030,” 2011.
[7] E. Hau, Wind turbines: fundamentals, technologies, application, economics, Springer Science & Business Media, 2nd edition, 2013.
[8] B. Navin Kumar, K. M. Paramasivam, M. Prasanna, and A. Z. G. Mohamet Karis, “Computational fluid dynamics dnalysis of aerodynamic characteristics of Naca 4412 vs S809 airfoil for wind turbine applications,” Int. J. Adv. Eng. Technol., vol. 7, no. 3, pp. 168–173, 2016.
[9] D. Luo, D. Huang, and X. Sun, “Passive flow control of a stalled airfoil using a microcylinder,” J. Wind Eng. Ind. Aerodyn., vol. 170, no. August, pp. 256–273, 2017.
[10] X. Shi, S. Xu, L. Ding, and D. Huang, “Passive flow control of a stalled airfoil using an oscillating micro-cylinder” Computers & Fluids, 2018.
[11] Y. Xie, J. Chen, H. Qu, G. Xie, D. Zhang, and M. Moshfeghi, “Numerical and experimental investigation on the flow separation control of S809 airfoil with slot,” Math. Probl. Eng., vol. 2013, 2013.
[12] L. J. Lina and W. H. Reed, “A preliminary flight investigation of the effects of vortex generators on separation due to shock,” NACA, 1950.
[13] R. S. Shevell, R. D. Schaufele, and R. L. Roensch. "Stall control device for swept wings."

United States, 1968.
[14] N. N. Sørensen, F. Zahle, C. Bak, and T. Vronsky, "Prediction of the effect of vortex generators on airfoil performance," J. Phys. Conf. Ser., vol. 524, no. 1, 2014.
[15] H. Wang, B. Zhang, Q. Qiu, and X. Xu, "Flow control on the NREL S809 wind turbine airfoil using vortex generators," Energy, vol. 118, pp. 1210–1221, 2017.
[16] N. Janjanam, K. R. Babu, S. S. Chowhan, A. Khan, and M. Teja, "Comparison of NACA 23024 aerofoil with and without vortex generators using CFD," Int. J. Mech. Eng. Technol., vol. 8, no. 5, pp. 556–566, 2017.
[17] O. Madani, M. Medale, O. Imine, and B. Imine, "Design optimization of the aerodynamic passive flow control on NACA 4415 airfoil using vortex generators," Eur. J. Mech. B/Fluids, vol. 56, pp. 82–96, 2016
[18] T. Durhasan, "Passive flow control over an airfoil by control rod at low Reynolds Number," Journal of applied fluid mechanics, vol. 13, no. 6, pp. 1819–1833, 2020.
[19] Y. Wang, G. Li, S. Shen, D. Huang, and Z. Zheng, "Investigation on aerodynamic performance of horizontal axis wind turbine by setting micro-cylinder in front of the blade leading edge," Energy, vol. 143, pp. 1107–1124, 2018.
[20] M. Tahani, G. Kavari, M. Masdari, and M. Mirhosseini, "Aerodynamic design of horizontal axis wind turbine with innovative local linearization of chord and twist distributions," Energy, vol. 131, pp. 78–91, 2017.
[21] M. N. Kaya, F. Kose, D. Ingham, L. Ma, and M. Pourkashanian, "Aerodynamic performance of a horizontal axis wind turbine with forward and backward swept blades," J. Wind Eng. Ind. Aerodyn., vol. 176, no. December 2017, pp. 166–173, 2018.
[22] M. G. Mourad, I. Shahin, S. S. Ayad, O. E. Abdellatif, and T. A. Mekhail, "Effect of winglet geometry on horizontal axis wind turbine performance," Eng. Reports, vol. 2, no. 1, pp. 1–19, 2020.
[23] M. A. Elfarra, N. S. Uzol, and I. S. Akmandor, "Investigations on blade tip tilting for hawt rotor blades using CFD," Int. J. Green Energy, vol. 12, no. 2, pp. 125–138, 2015.
[24] R. Himo, "Aerodynamic performance enhancement of an Airfoil using trapezoidal vortex generators," vol. 1, no. June 2020, pp. 76–84, 2021.
[25] X. Li, K. Yang, and X. Wang, "Experimental and numerical analysis of the effect of vortex generator height on vortex characteristics and airfoil aerodynamic performance," Energies, vol. 12, no. 5, 2019.
[26] W. Chester, "Hydrodynamic design of the humpback whale flipper," Journal of morphology, vol. 60, pp. 51–60, 1995.
[27] F. E. Fish, Ã. L. E. Howle, and M. M. Murray, "Hydrodynamic flow control in marine mammals," vol. 48, pp. 788–800, 2008.
[28] F. E. Fish, P. W. Weber, M. M. Murray, and L. E. Howle, "The tubercles on humpback whales ' flippers : application of bio-inspired technology," vol. 51, no. 1, pp. 203–213, 2011.
[29] D. S. Miklosovic, M. M. Murray, L. E. Howle, and F. E. Fish, "flippers Leading-edge tubercles delay stall on humpback whale flippers," AIP publishing, vol. 39, no. 2004, 2012.
[30] H. Johari, "Effects of leading-edge protuberances on airfoil performance," vol. 45, no. 11, 2007.
[31] K. L. Hansen, R. M. Kelso, and B. B. Dally, "Performance variations of leading-edge tubercles for distinct airfoil profiles," AIAA Journal, vol. 49, no. 1, 2011.
[32] M. M. Z. G. F. W. J. Z. Xu, "Experimental study of flow separation control on a low- Re airfoil using leading-edge protuberance method," no. 11, 2014.

[33] G. Huang, Y. C. Shiah, C. Bai, and W. T. Chong, "Experimental study of the protuberance effect on the blade performance of a small horizontal axis wind turbine," Jnl. Wind Eng. Ind. Aerodyn., vol. 147, pp. 202–211, 2015.

[34] M. Asli, B. M. Gholamali, and A. M. Tousi, "Numerical analysis of wind turbine airfoil aerodynamic performance with leading edge bump," Math. Probl. Eng., vol. 2015, 2015.

[35] M. Mashud, D. Mondal, and M. E. Haque, "Experimental investigation of an airfoil with multiple dimples on the upper surface," ICMERE 2017-PI-237, no. December 2017, 2018.

[36] S. Aziz, A. Khan, I. Shah, T. A. Khan, Y. Ali, and M. U. Sohail, "Computational fluid dynamics and experimental analysis of a wind turbine blade ' s frontal section with and without Arrays of dimpled structures," energies, no. September, 2022.

[37] A. Ebrahimi and M. Movahhedi, "Wind turbine power improvement utilizing passive flow control with microtab," Energy, vol. 150, pp. 575–582, 2018.

[38] Mostafa, W., Abdelsamie, A., Mohamed, M., Thévenin, D. and Sedrak, M., 2020, November. Aerodynamic performance improvement using a micro-cylinder as a passive flow control around the S809 airfoil. In IOP Conference Series: Materials Science and Engineering (Vol. 973, No. 1, p. 012040). IOP Publishing.

[39] Mostafa, W., Abdelsamie, A., Sedrak, M., Thévenin, D. and Mohamed, M.H., 2022. Quantitative impact of a micro-cylinder as a passive flow control on a horizontal axis wind turbine performance. Energy, 244, p.122654.