

Boundary Layer Control Effects on The Performance of NREL S826 Aerofoil for Wind Turbines.

Aljunayd Aloukili^{*,**}, Mostafa Ali^{*,†}, M. Elfaisal Elrefaie^{**}, Mohamed Elsakka^{***}

^{*} Department of Mechanical Engineering, University of Derna, Derna, Libya

^{**} Department of Mechanical Power Engineering, Al Azhar University, Cairo, Egypt

^{***} Department of Mechanical Power Engineering, Port Said University, Port Said, Egypt

(aljonaid.mohammed@uod.edu.ly, mostafa_azhar2015@yahoo.com, m.abdo202084@gmail.com, elsakka@eng.psu.edu.eg)

[†] Corresponding Author; Aljunayd Aloukili, Tel: +218911697391, aljonaid.mohammed@uod.edu.ly

Received: 07.04.2024 Accepted: 27.05.2024

Abstract- With the increasing demand for sustainable energy sources, wind turbines have emerged as the dominant solution for clean energy production. However, there is a lack of discussion regarding the concept of pattern identification for visualizing the design and evaluation of boundary layer of the blowing and suction technology. In this paper, several simulation models are developed based on previous literature, comparing the results of different cases and attempting to derive a rule for determining the appropriate positions for blowing and suction ports. The impact of the applied jet force on the aerodynamic performance of the NREL S826 aerofoil is also examined. ANSYS CFD program is used to run the simulations with the SST K- ω turbulence model and 5% turbulence intensity. The findings revealed that positioning the blowing port closer to the separation position resulting from large angles of attack significantly improves the lift force generated for each case. Additionally, decreasing the applied momentum coefficient not only improves the aerodynamic performance but also reduces the loads on the turbine blade. For the model S826-0.2B-0.7S, where the blowing port is closest to the position of separation at high angles of attack, there is a substantial increase in the lift coefficient by 49.8% with a relatively small increase in drag value up to 60%. On the other hand, the model S826-0.2B-0.7S, while applying a momentum coefficient of 0.08 provides a lift coefficient increase of 51.1% with a minimal increase in drag coefficient of only 11.3%, making it the most efficient among the other cases.

Keywords- wind turbine, boundary layer control, NREL S826 aerofoil, computational fluid dynamics, lift and drag coefficient.

1. Introduction

Due to the rapid growth of population and urbanization in developing nations, there has been a significant global surge in the demand for electric power. The reliance on diminishing fossil fuel and natural gas reserves, coupled with mounting environmental concerns, necessitates the exploration of renewable and eco-friendly methods for power generation. There is increasing research in the field of aerodynamics of wind turbines. Across the globe, there is a proactive pursuit of renewable energy approaches such as hydroelectric, solar power conversion, and wind power extraction to address the continuously increasing need for electricity. The field of wind turbine aerodynamics has been growing rapidly, driven by the need to improve the efficiency and performance of wind turbines. Researchers are exploring various aspects such as blade design, aerofoil shapes, turbine placement, and wake interactions to optimize energy capture and reduce costs. Advances in numerical methods such as Boundary Element (BE) and Computational Fluid Dynamics

(CFD) have also contributed to advancements in this field [1-3].

The Earth possesses a wide array of renewable energy sources, with solar energy, wind energy, and geothermal energy being prominent examples of diverse and sustainable options for power generation [4-6]. Focusing on wind energy, it has been observed that there are significant variations in wind speeds across the globe, as indicated by the wind atlas [7, 8]. Areas with high wind velocities have already been utilized for energy production, but in the future, there will be a need to explore regions with moderate and low wind speeds [9]. Significant progress has been made in the development of wind turbine technology. Through advancements in materials, manufacturing techniques, intelligent control systems, and rotor aerodynamics, the rotor diameter of a 5MW wind turbine has reached an impressive 126 meters [10, 11]. Extensive research has been conducted to enhance the size, efficiency, and reliability of wind

turbines while simultaneously reducing costs, complexity, and risks associated with their operation. However, despite these advancements, there have been relatively few changes in the aerodynamic profiles of wind turbine blades. Research on the impact of the boundary layer on an aerofoil has been ongoing since the mid-20th century. The existence of the boundary layer has given rise to numerous design challenges in various domains of aerodynamics. Nevertheless, the focus of the investigations has primarily been on studying its impact on the lift and drag of the blade. Currently, the utilization of this boundary layer control is evident in wind turbines. Methods have been devised to modify the boundary layer, with the aim of either augmenting lift or reducing drag. A higher level of influence would result in improved performance of the wind turbine[12].

Blowing and suction techniques are commonly employed for separation control on an aerofoil, both in steady and unsteady conditions. Various methods have been utilized in the past to delay separation by increasing the momentum of the boundary layer [13].

2. Literature Survey

Numerous configurations have been developed to address the issue of separation and improve its mitigation. One approach is the implementation of jet blowing and suction, which involves applying an external moment to the flow stream over the aerofoil. This additional moment helps prolong the adherence of the flow stream to the aerofoil surface, leading to a stall angle delay. Consequently, the aerofoil becomes capable of generating higher lift forces, resulting in higher power coefficients[14].

In their study, Khoshnevis et al. [15] conducted a numerical analysis to evaluate the efficiency of the CFJ aerofoil at five distinct moderate Reynolds numbers. The results indicate that the lift coefficient of the CFJ aerofoil increases as the coefficient of momentum increases. Once the coefficient of jet momentum surpasses the threshold limit, the performance of the CFJ aerofoil decreases. Lei et al. [16] examined the impact of suction control on the laminar separation bubble of the NACA2415 aerofoil at low Reynolds number (Re). They employed a γ - $Re\theta$ transition model and k -turbulence model for their analysis. The simulation outcomes shown that the implementation of suction control significantly mitigated flow separation. In contrast, a higher proximity between the suction position and the separation bubble, along with an increased suction velocity, leads to enhanced control efficacy. Zhang et al. [19] performed an experiment to analyze the impact of DCFJ control on a rotor aerofoil. Regarding computational fluid dynamics (CFD) investigations, the majority of the numerical techniques discussed in previous research papers rely on the utilization of the Reynolds Averaged Navier-Stokes (RANS) equations in conjunction with the Spalart-Allmaras (S-A) turbulence model. Balaji et al. [20] performed an experimental study on a Modified CFJ (MVCJ) aerofoil and compared it to a baseline aerofoil. The revised CFJ incorporated a convergent-divergent (CD) nozzle at the

injection slot to alleviate the workload of the pump. This study determined that the lift coefficient is enhanced by 43% in comparison to the aerofoil without any modifications. Sun et al. [17] conducted an optimization study to investigate the ideal placements of blowing and suction jets as well as the appropriate C_m . Their investigation concluded with three main findings. First, CFJ's flow separation control works best when the blowing and suction jets are positioned 0.03 and 0.8 cm, respectively, from the leading edge. Secondly, the greater C_m of CFJ causes a stall to be delayed and a noticeable rise in C_L . Third, a Darrieus VAWT wind power coefficient is greatly improved by the CFJ control. Wang et al. [18] conducted a numerical study on the impact of Reynolds number on the aerodynamics of a CFJ6421 aerofoil. They discovered that a higher Reynolds number had a positive influence on the aerodynamics of the CFJ aerofoil. In their study, Arunraj et al. [19] conducted experiments to assess the efficacy of aerofoil suction control in a wind tunnel. They demonstrated that suction control exhibits superior performance when operating at low suction pressures and small AOA. Additionally, they found that the highest lift coefficients can be achieved by implementing suction control at 70% of the chord length (c). In their study, Lei et al. [20] utilized the CFJ (Coanda Flow Jet) technique to enhance the low-speed performance of a supersonic narrow aerofoil. This research has potential implications for the design of supersonic vehicles that operate throughout a wide range of speeds. Numerical methods were used to analyze the aerodynamic efficiency of different blade profiles. Due to the low Reynolds number, this study focuses mainly on the performance of vertical-axis wind turbines. The work focused on examining the stall and separation effects in order to develop guidelines for effectively utilizing a particular aerofoil at a certain range of speeds to attain the highest possible power coefficient[21]. Despite many attempts to achieve the best use of CFJ technology by proposing many different designs that are characterized by changing the dimensions and locations of the blowing and suction ports, the optimal position for the location of the suction and blowing port has not been determined. In this study, a number of designs will be presented for NREL S826 aerofoil. Furthermore, this study includes an assessment of how the modified lift and drag coefficients, resulting from suction and blowing, affect the tangential and thrust forces generated by the aerofoil at different conditions.

3. Numerical Modelling

Computational Fluid Dynamics (CFD) stands as a cornerstone in the iterative process of design refinement and validation. It offers an expedited pathway for transitioning designs from concept to real-world applications, often with greater efficiency and cost-effectiveness compared to traditional experimental methods. While experimental approaches remain invaluable for certain scenarios, well-executed CFD simulations yield results that closely approximate experimental findings, thereby providing highly reliable insights.

3.1 Turbulent Model and Governing Equations

In our study, we employ the ANSYS simulation suite to conduct CFD simulations on the NREL S826 aerofoil, utilizing a simultaneous blowing and suction CFJ technique to explore the most suitable parameter set among the tested conditions. Leveraging the RANS-based capabilities of ANSYS FLUENT, we employ a finite element solution technique, incorporating the fundamental governing equations of mass, energy, and momentum. The integration of these governing equations enables us to comprehensively analyze the aerodynamic behavior of the NREL S826

aerofoil under various operating conditions. Additionally, we adopt the SST K- ω turbulence model, which is renowned for its robustness in capturing turbulent flows, characterized by dissipation rate and turbulent kinetic energy, with a 5% turbulence intensity specified for this CFD simulation. This meticulous approach ensures a thorough understanding of the flow dynamics, facilitating informed design decisions and optimization strategies. More details are shown in **Error! Not a valid bookmark self-reference.**

Table 1: ANSYS Simulation details.

	Model specifications	Notes
Solver	Double-Precision	The double-precision uses 64 bits to represent each floating-point number. The extra bits increase not only the precision, but also the range of magnitudes that can be represented.
	Pressure-based solver	The pressure-based solver accounts for the incompressible flow around the aerofoil.
Pressure-Velocity coupling	Coupled	Ensure the pressure field is corrected based on the velocity field to satisfy mass conservation (divergence-free), then the velocity field is corrected based on the new pressure field state to ensure that momentum equations and maintain consistency with the updated pressure distribution.
Mesh topology	Structure mesh	structured mesh topology offers advantages in terms of computational efficiency, numerical accuracy, ease of boundary treatment, and robustness.
Element type	Quad cell	quad cell is a two-dimensional (2D) element consisting of four vertices (nodes) and four edges. It is commonly used to discretize and represent planar or curved surfaces
Computational resources	6 parallel nodes on a PC with i11400h i5 processor with 16GB RAM	Each case is calculated in 15 min.
No. of iterations	1000	This is found to reduce the residuals of the main flow variables below 1E-5.
Solution initialization	Hybrid	It uses the solver algorithms to initialize the flow field iteratively based on the specified boundary conditions and other solver

		settings. Hence, provides more accurate initial conditions, leading to faster convergence and more reliable results.
--	--	--

3.2 Geometry and Computational Domain

The NREL S826 aerofoil is the selected aerofoil for the present case study. However, various changes in its geometry will be considered. The NREL S826 aerofoil is selected as it is characterized by good performance at low Reynolds numbers which are typical for operation conditions, the S826 aerofoil is versatile and can be used in various sections of wind turbine blades and the S826 aerofoil is developed by the National Renewable Energy Laboratory (NREL), which means it has undergone rigorous testing and validation. The Literature reviews highlight some factors to avoid the decreased performance that may accompany the application of the CFJ technique. It's found that for proper blowing effect over the separated flow, the blowing ports should be applied over the range of 0.08 to 0.2 from the cord length measured from the leading edge, applying earlier blowing jet will trigger early turbulence. Meanwhile, the range of suction port should be over the range of 0.7 to 0.8 from the cord length measured from the leading edge, applying earlier suction jet will form a secondary turbulence bubble. Applying 45-degree tilt angle for both blowing and suction ports allows fine air distribution over the aerofoil and avoiding any farther disturbance in the mean stream. The blowing path is convergent path to provide the required high velocity to achieve sufficient momentum with small compressor as possible. The suction path is divergent path is

to increase the low-pressure condition at the suction port, hence, rescuing the compressor required power, crafting the blowing and suction ports with a fixed port depth of 2% from the chord length (C) for the blowing port and 3% from the chord length for the suction port achieves this purpose. Multiple positions for blowing and suction ports and blowing and suction jet momentum coefficients are chosen to monitor their effect on the aerodynamic characteristics of the NREL S826 aerofoil.

The chosen port positions for blowing are 0.08C, 0.1C, and 0.2C and the suction port positions are set to be at 0.7C and 0.8C. The momentum coefficient, C_m , effect is chosen to be monitored at the values of 0.05, 0.08, 0.16, 0.2, 0.4, and 0.5. Figure 1 illustrates the normal configuration for NREL S826. Meanwhile, Figure 2 shows the different sets of geometrical configuration changes performed over the NREL S826 aerofoil. The CFD domain used for the current study is shown in Figure 3, and the mesh generated around the normal NREL S826 aerofoil and the case S826-0.2B-0.7S aerofoil are shown in Figure 4. In the case S826-0.2B-0.7S, "S826" refers to the aerofoil section, "0.2B" indicates the location of the blowing port at 0.2 of the chord length, and "0.7S" indicates the location of the suction port at 0.7 of the chord length, measured from the leading edge

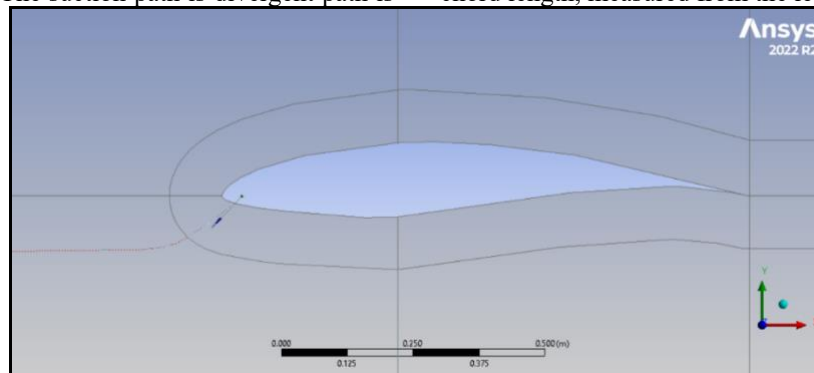
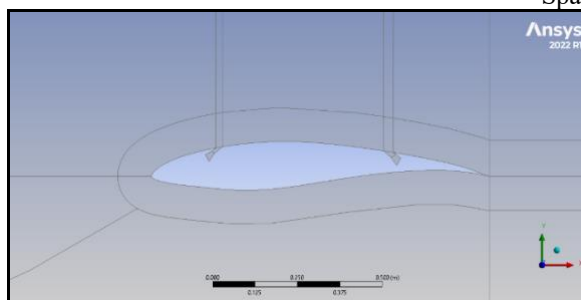
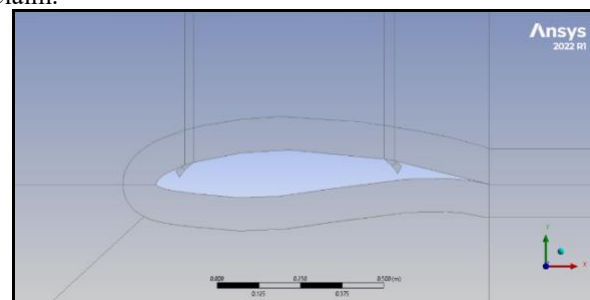


Fig.1. The normal configuration for NREL S826 before applying blowing and suction CFJ configuration on ANSYS SpaceClaim.



(a)



(b)

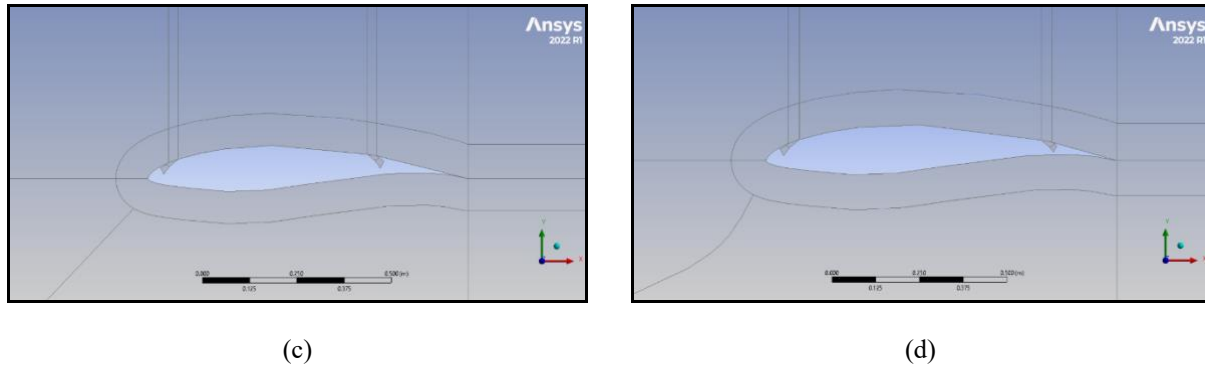


Fig.2. The various augmented CFJ technique configurations over a NREL S826 aerofoil where (a) 0.2B-0.7S (b) 0.1B-0.7S, (c) 0.08B-0.7S and (d) 0.08B-0.8S on ANSYS SpaceClaim.

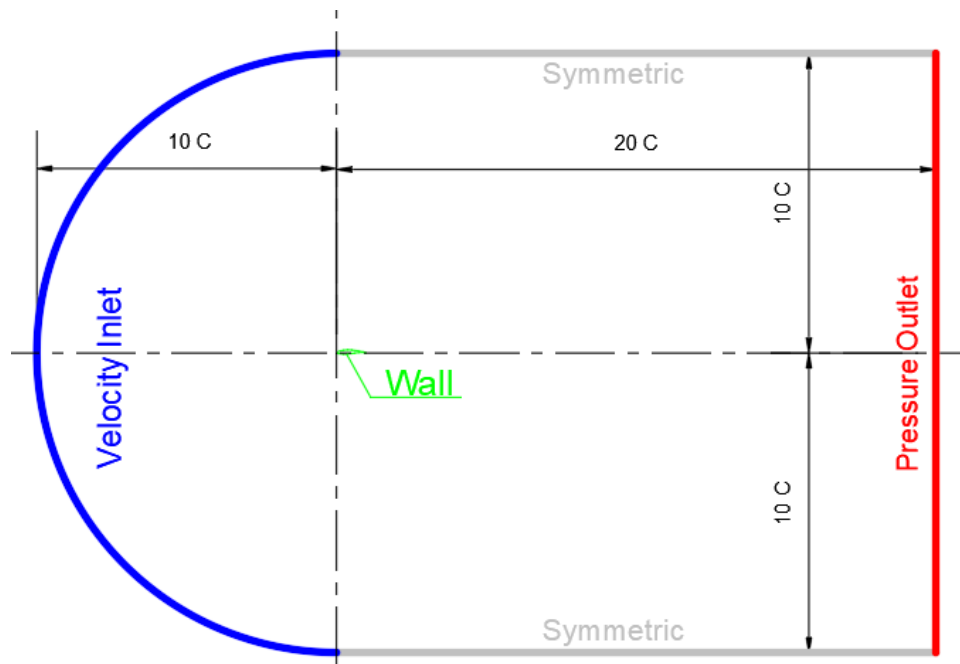


Fig. 3. The CFD simulation domain used through the current case study with 10 times the chord length distance from the leading, pressure side and suction side and 20 times the chord length distance from the trailing edge.

The mesh is generated using the ANSYS Meshing program while taking the common meshing consideration into account to ensure high-accuracy results. The generated mesh as shown in Figure 4 is characterized by a higher number of mesh elements near the aerofoil surface, where rapid and complex changes in fluid flow properties take place within the boundary layer. A mesh with 340700 elements is found to be the most suitable number respecting the simulation results accuracy and the computational cost and time. The simulation is performed at a Reynolds number (Re) of 6×10^5 and applying simultaneous blowing and suction with an equal value of momentum coefficient (C_m) for both the blowing and suction process, Where the momentum coefficient (C_m) can be obtained by **Error!**
Reference source not found.:

$$C_m = \frac{\dot{m}_j V_j}{0.5 \rho U_\infty^2 S}$$

Where:

C_m : momentum coefficient

$\dot{m}_j$: jet mass flow rate

V_j : jet velocity

ρ : air density at normal atmospheric conditions

U_∞ : ambient air velocity

S: the aerofoil span side are

Although the blowing and suction are kept identical in each case, the C_m values vary from 0.05 to 0.5, corresponding to flow rates of 0.24 to 0.76 Kg/s. The specific values for C_m are 0.05, 0.08, 0.16, 0.2, 0.4, and 0.5, while the corresponding flow rates are 0.24, 0.31, 0.43, 0.49, 0.68, and 0.76 Kg/s, respectively. A mass flow inlet boundary

condition is imposed at the entrance of the blowing nozzle, while the mass flow rate is used to measure the mass leaving the suction nozzle exit.

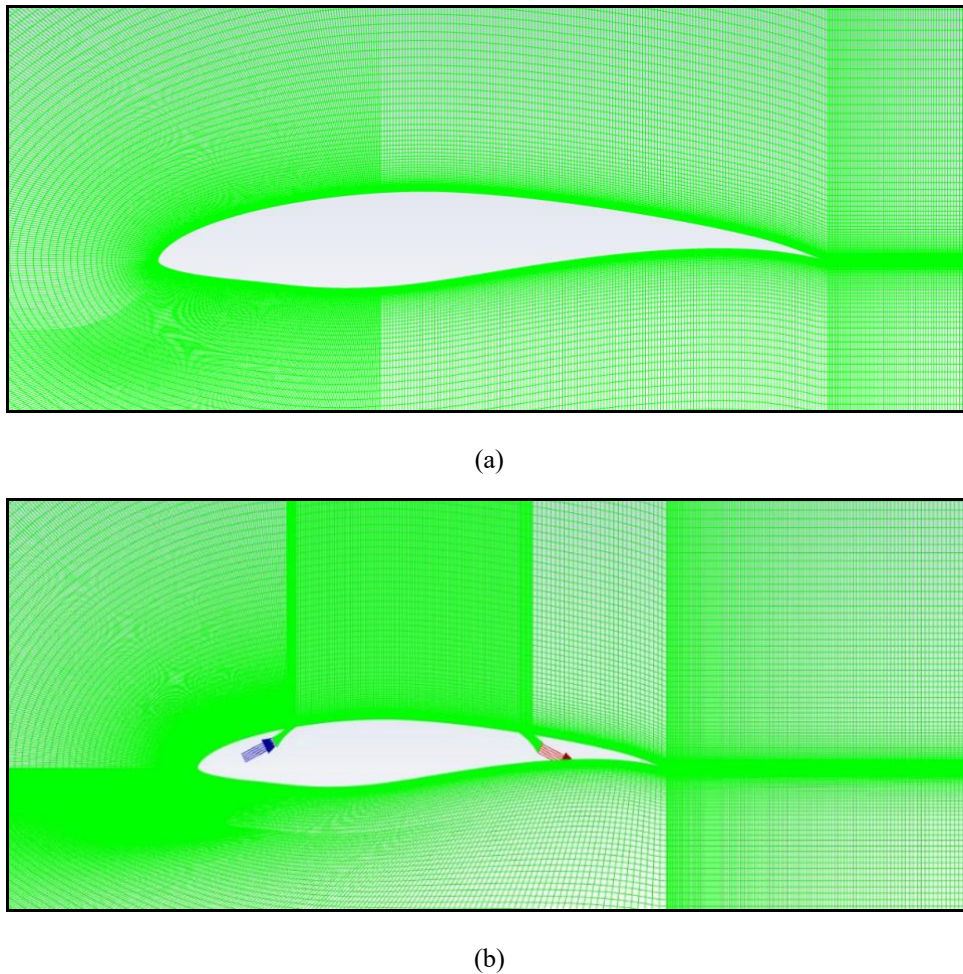


Fig. 4. The mesh configuration generated by ANSYS Meshing program for the CFD simulation over (a) the normal NREL S826 design and (b) the CFJ augmented S826-0.2B-0.7S aerofoil.

4. Results and Discussion

Based on the literature review, the main objective of the blowing and suction co-flow jet (CFJ) technique is to overcome the separation phenomenon which represents a barrier toward a better aerodynamic characteristic over the aerofoil. An aerofoil can be said to have better aerodynamic characteristics when achieving high lift coefficient (C_L) while maintaining a low value of drag coefficient (C_D) or achieving even lower C_D values. This can be obtained by creating conditions where the stream of air flow can attack the aerofoil profile at high values of attack angle (α) without separating from the aerofoil surface, such conditions can be provided using the CFJ technique as observed.

The distance where blowing and a suction port are located with respect to the chord length of the selected aerofoil, $(X/C)_B$ and $(X/C)_S$, the momentum coefficient (C_m), the port height with respect to the aerofoil chord (h/C), and the tilt angle of the CFJ port represent the main and decisive

parameters that affect the conditions provided over the aerofoil surface aiming to prevent the separation phenomenon. Hence, enhancing the aerodynamic properties of an aerofoil. However, the effect of changing the port position and the momentum coefficient are the parameters to be studied in this case.

In order to make sure that the CFD-generated results are reliable and possess a level of realism, the results obtained from the normal configuration of the NREL S826 aerofoil are to be validated by the experimental results obtained by Barti et al. [22].

4.1 Validation

Results shown in Figure 5 represent the validity of the mesh and CFD results generated based on the normal configuration of the NREL S826 aerofoil. Hence, by applying the same process for the geometries designed for

the performed case study would ensure the reliability of the obtained results.

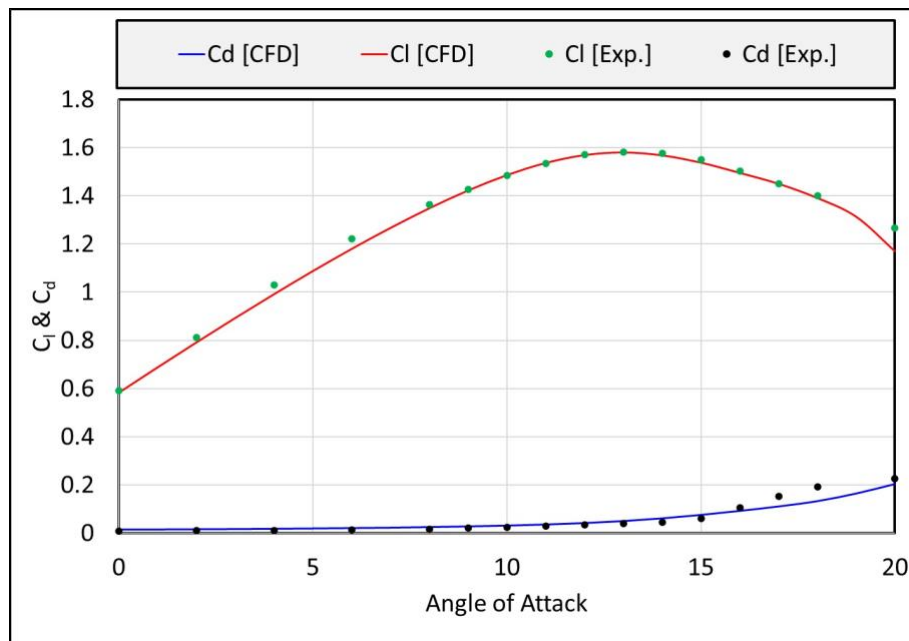


Fig.5. The validation results between the normal configuration NREL S826 CFD simulation results and the experimental results obtained by Barti et al.[22].

4.2 The effect of Momentum Coefficient Variation Over the Aerodynamic Performance of the NREL S826

The simulation is, first, performed at a fixed position for the blowing and suction ports of the model S826-0.2B-0.7S while varying the values of C_m from 0 to 0.5. Based on existing research on NACA profiles the case 0.2B-0.7S at $C_m = 0.2$ is expected to give the best performance among the tested models. However, this isn't the case. As shown in Figure 6, which illustrates the relation between lift and drag coefficient and the angle of attack under the effect of momentum coefficient for the model S826-0.2B-0.7S, at first glance, the prevailing impression is that the higher the value of C_m , the higher the value of C_L , with a slight change in the value of C_D at low angles of attack. Stall delay is also a

common effect that appears over the model S826-0.2B-0.7S for most applied C_m values.

The lift coefficient (C_L) witnesses an enhancement with a minor change in the value of drag coefficient (C_D) and stall delay when activating the blowing and suction CFJ technique. This mainly is due to the intentionally generated vortexes working to exchange their momentum with the main wind streamlines which leads the following results, at low values of attack angle (α) the excess momentum added to the mainstream increases the generated lift coefficient as resulting forces increase. For a bigger attack angle (α), where stall normally occurs at the base condition, the excess applied momentum forces the mainstream to keep attacking the aerofoil surface.

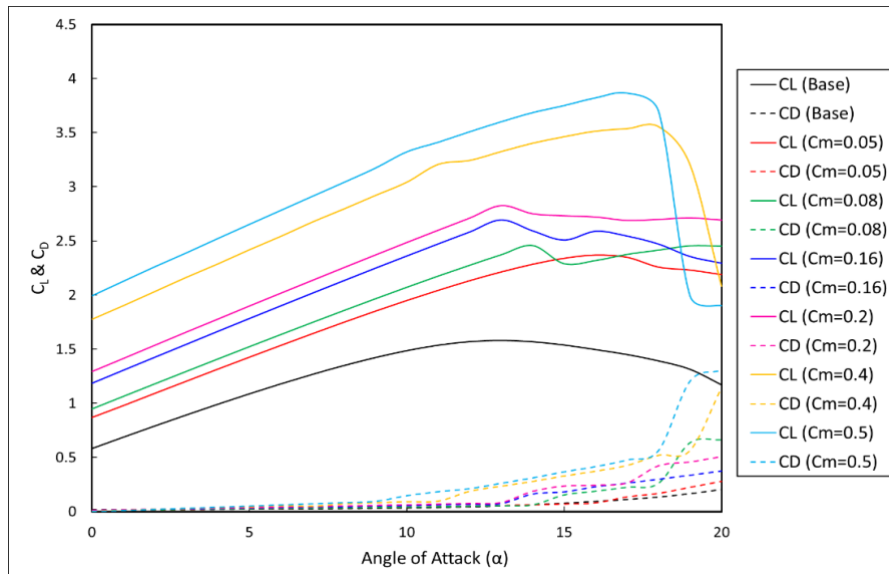
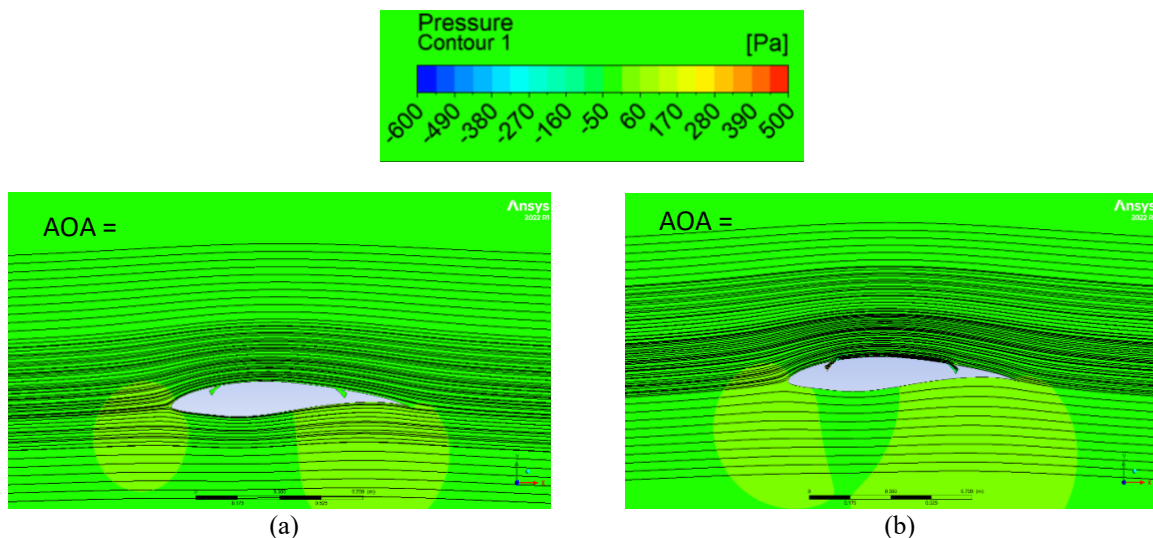


Fig. 6. The relation between lift and drag coefficient and the attack angle (α) under the effect of momentum coefficient (C_m) for the model S826-0.2B-0.7S at $Re = 6 \times 10^5$ simulated using ANSYS Fluent.

The influence of blowing and suction CFJ technique over the model S826-0.2B-0.7S with applying $C_m = 0.05$ can be observed in Figure 7. As can be observed in Figure 7 (a) and (b) the effect of applying blowing and suction CFJ at attack angle $\alpha = 0^\circ$ is reducing the shock of flow at the leading edge and generating high uniformed velocity of wind stream over this aerofoil. hence, more pressure uniformity over both the suction and pressure side of the model leads to a more stable operation and higher lift coefficient value with the negligible change in drag coefficient presented in Figure 6. When increasing the attack angle to $\alpha = 15^\circ$ while keeping the model S826-0.2B-0.7S under observation, the effect of blowing and suction CFJ technique remarkably appears as shown in Figure 7 (c) which illustrates the stall effect over the S826 at base condition, a formation of vortexes can be easily observed at the trailing edge of the aerofoil. These vortexes increase the drag coefficient while causing a

reduction in lift coefficient value which is an unwanted condition for aerofoil aerodynamics. In contrast, in Figure 7 (d), streamlines are fully attached to the model surface with no vortex formation at the leading or trailing edges increasing the lift coefficient and achieving the same effect occurs at the condition S826-0.2B-0.7S at attack angle (α) of 0° from the perspective of pressure distribution.

Figures 7 (e) and (f) represent the worst case at attack angle $\alpha = 20^\circ$. For the base case, the stall effect is remarkably noticed with large vortexes area and big stagnation shock of the flow. In addition to the appearance of small vortexes area, compared to these appears at the base condition, at the trailing edge for the case of model S826-0.2B-0.7S with $C_m = 0.05$ at attack angle (α) of 20° , small vortexes are appeared at the leading edge, shown in Figure 8 increases the drag coefficient and reducing the lift coefficient.



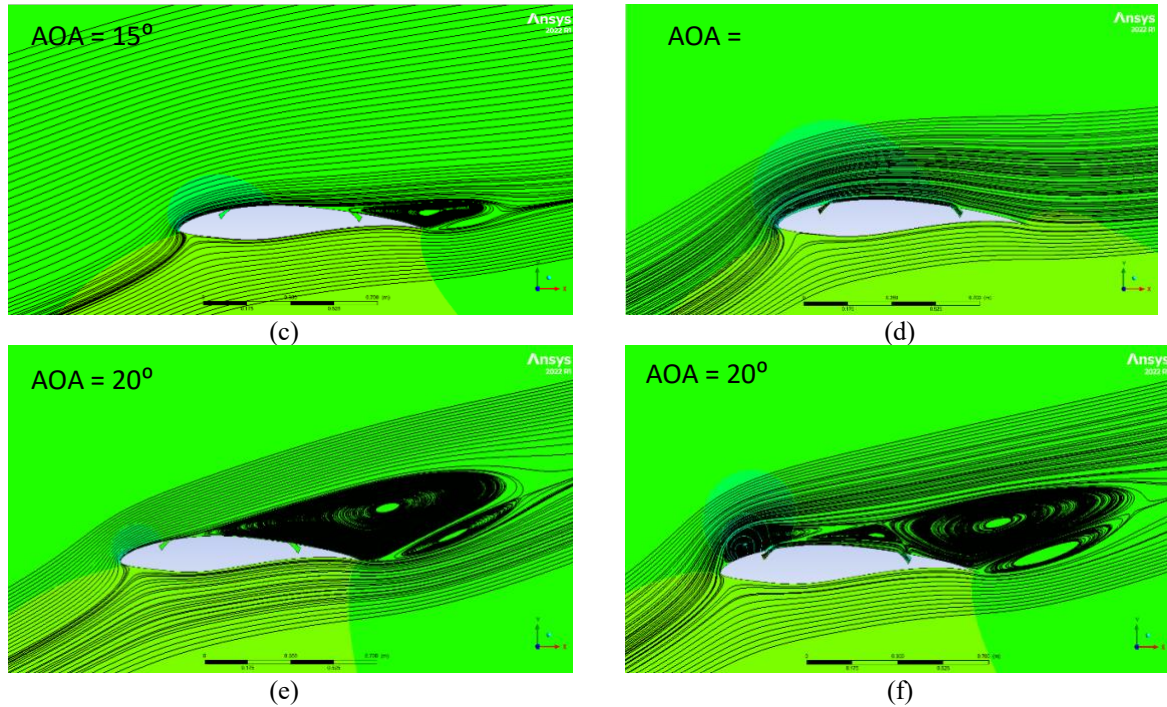


Fig.7. The aerodynamic field without and with blowing and suction CFJ technique over the model S826-0.2B-0.7S applying $C_m = 0.05$ at $Re = 6 \times 10^5$ simulated using ANSYS Fluent post-process.

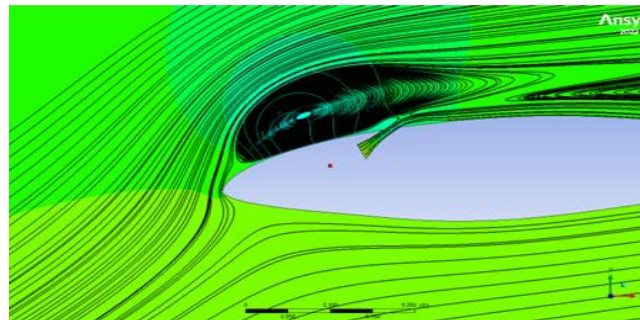
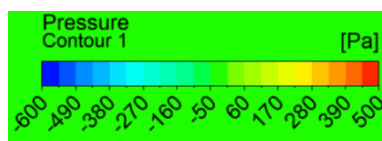


Fig. 6. A closeup view of the formed vortices at the leading edge of model S826-0.2B-0.7S applying $C_m=0.05$ at $\alpha=20^\circ$ at $Re = 6 \times 10^5$ simulated using ANSYS Fluent post-process.

As the minimal considered momentum coefficient managed to enhance the aerodynamic performance of the S826 aerofoil, various momentum coefficients are applied over the model S826-0.2B-0.7S to investigate its effect on the model. Figure 9 shows the effect of applying blowing and suction CFJ at C_m of 0 (base case), 0.2 and 0.5 at attack angles equal to 0° , 15° and 20° , respectively. The model S826-0.2B-0.7S at $C_m = 0.2$ shows an increase in lift coefficient. However, at a high angle of attacks the generation of vortices is demolished at trailing edge and appear at the leading edge instead resulting in an increase in

drag coefficient values, which results in a reduction of the aerodynamic efficiency of the blade compared with the base case. For Model S826-0.2B-0.7S at $C_m = 0.5$, although, the high stall delay is achieved, vortices are generated at the leading edge at early stages of attack angle values with respect to the stall angle delay. Hence, a reduction in the aerodynamic properties is obtained despite the high lift generated.



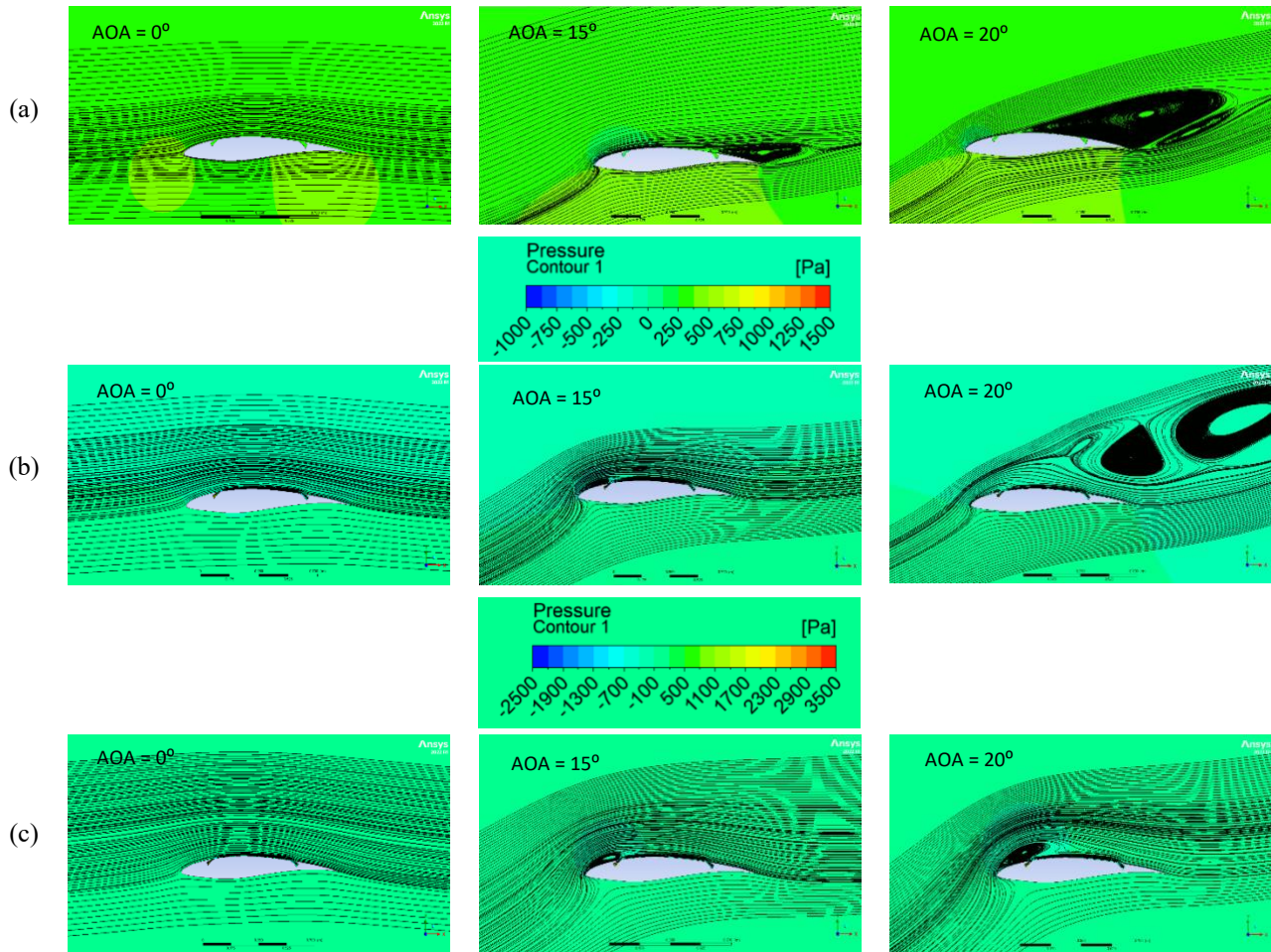


Fig.9. The effect of applying (a) $C_m = 0$ (base case), (b) $C_m = 0.2$, and (c) $C_m = 0.5$ on the aerodynamic field of the model S826-0.2B-0.7S at $Re = 6 \times 10^5$ simulated using ANSYS Fluent post-process.

As observed from Figure 9 and Figure 7 all values of applied momentum coefficient managed to prevent the separation of flow from the model surface at trailing edge. However, high momentum coefficient values result in lower aerodynamic efficiencies compared with the base case. Figure 10 illustrates the relation between the lift to drag ratio and the attack angle with change in momentum coefficient for the model S826-0.2B-0.7S. It can be observed that for the model S826-0.2B-0.7S the lower the C_m value the higher the aerodynamic efficiency for most of the attack angle range compared to the base case. Results of applying blowing and suction CFJ technique on the average lift coefficient, drag coefficient, lift to drag ratio and stall angle delay are

displayed at Table 1 for model S826-0.2B-0.7S at various momentum coefficient C_m values. As can be observed from the data in Table 1, applying blowing and suction CFJ technique is always able to enhance the lift value. However, it must be applied with caution as blowing and suction CFJ technique also increases the value of drag over the blade surface, which is an undesired phenomenon for most applications. Applying high momentum coefficients C_m causes high drag force over the aerofoil surface. For very low and very high momentum coefficient C_m values a delay in stall angle occurs by remarkable values. This phenomenon is suggested to be used for automatic control of the aerofoil or wing characteristic.

Table 2: Results of applying blowing and suction CFJ technique for the model S826-0.2B-0.7S at $Re = 6 \times 10^5$ for the main stream.

C_m	Increase in % C_L average	Increase in % C_D average	Increase in % (C_L / C_D) average	Increase in Stall angle delay
0.05	44.2	21.4	15.4	3°
0.08	52.85	120.7	11.5	1°
0.16	69.4	100.4	11.6	0°
0.20	81.9	142.4	13.9	0°
0.4	124	285.9	57.1	5°

0.5	136.8	390.7	354.3	4°
-----	-------	-------	-------	----

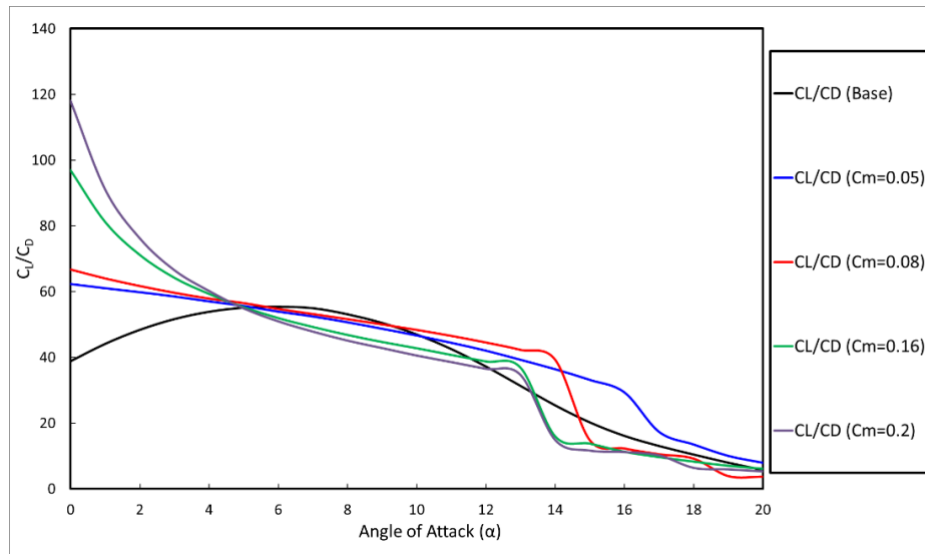


Fig. 10. The effect of increasing C_m value on the C_L / C_D ratio at various attack angle at $Re = 6 \times 10^5$ for the main stream.

When an aerofoil is submerged in an air stream various aerodynamic forces are generated. Figure 11 illustrates these aerodynamic forces, which are lift force F_L , drag force F_D , tangential force F_n , trust force F_T , and resultant force F_R , and the velocity triangle of an aerofoil. The generation of these forces depends primarily on the aerodynamic conditions

surrounding the blade, which are intentionally manipulated by CFJ technology. Based on this principle, it is necessary to study the effect of large values of C_m on the generation of these forces, which may negatively affect the performance of the turbine blade from structural aspects.

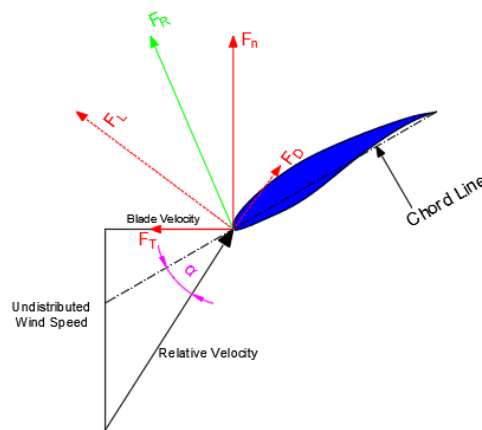


Fig11. Aerodynamic forces analysis applied on the NREL S826 aerofoil.

Considering NTNU (Norwegian university of science and technology) HAWT turbine geometry [23], Figure 12 illustrates both tangential and thrust forces coefficients – C_n and C_T , respectively, and their changes in the spanwise direction on the turbine blade when applying various C_m values. To evaluate the effect of the enhanced lift coefficient for wind turbine applications, the estimated tangential and thrust forces coefficients are examined along the spanwise direction of an NTNU turbine blade. It is observed that

increasing the momentum coefficient increases the tangential and thrust forces. However, there is a significant increase in the thrust force, especially near the blade tip. Therefore, it is advisable to implement suction and blowing in sections away from the blade tip. Stronger and more expensive material will be required for the manufacture of turbines with blowing and suction CFJ technique. For such reasons and for their poor aerodynamic efficiency, high momentum coefficients C_m should be avoided.

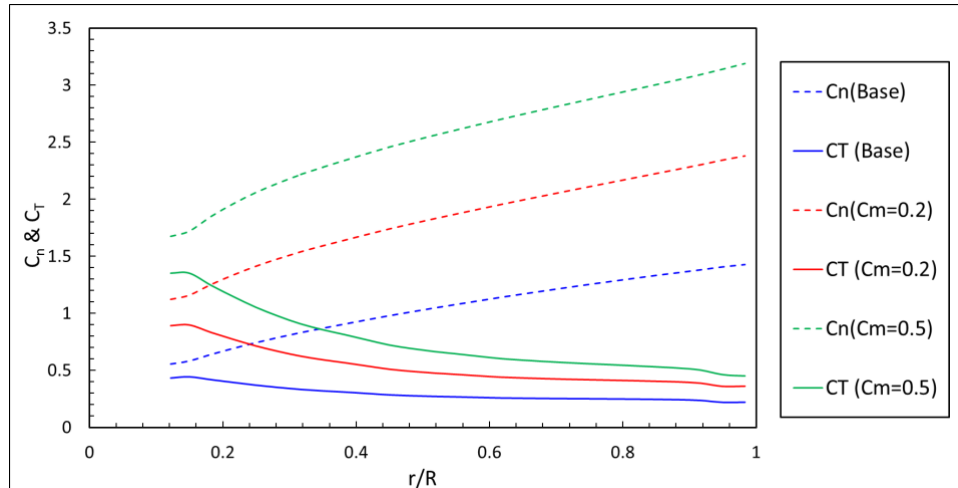


Fig. 12. Tangential and thrust force coefficients – C_n and C_t , respectively – and their relationship to the position at the turbine blade and the momentum coefficient C_m applied at $Re = 6 \times 10^5$ for the main stream, focuses on the potential stresses on the NREL Phase VI blade structure.

4.3 The effect of both blowing and suction ports position and momentum coefficient variation over the aerodynamic performance of the NREL S826

The position of blowing and suction ports can be considered an important design parameter which affect the performance of a blowing and suction CFJ technique. Various positions for blowing and suction plots are simulated to configure the path towards a better performance. To serve that purpose various models for the S826 aerofoil are designed. The designed models are shown in Figure 2. Figure 13 illustrates the effect of ports position on the lift and

drag coefficients when applying a $C_m = 0.2$. As can be observed, the model S826-0.1B-0.7S provides the highest lift enhancement with a value equal to 81.3% and a 3° stall delay. However, the drag produced is also higher than its presumptions leading to low aerodynamic characteristics. It can also be seen that the model S826-0.2B-0.7S provides a high lift with a value equal to 78.6% and no stall delay accompanied with an increase in overall drag values. In general, the presented models in this case all perform low aerodynamic characteristics.

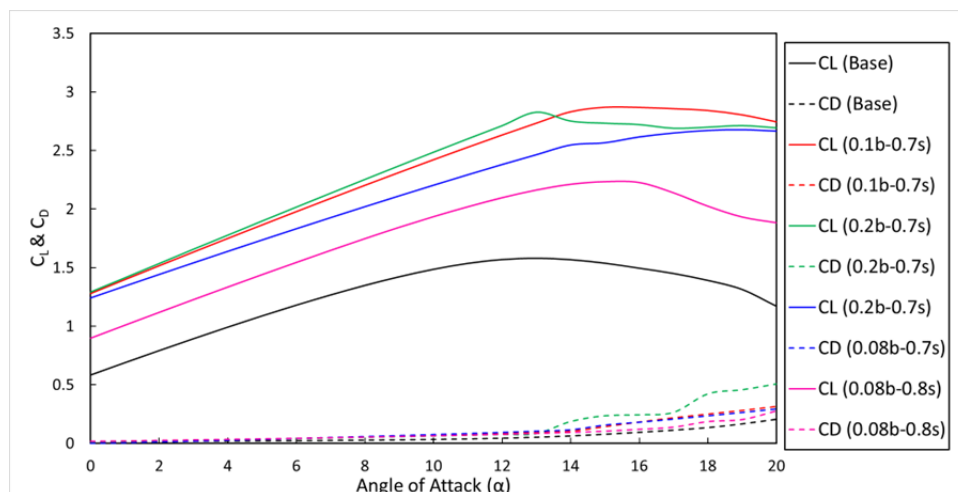


Fig. 7. The effect of ports positions on the lift and drag coefficients when applying a $C_m = 0.2$ at $Re = 6 \times 10^5$ for the main stream.

Reducing the value of momentum coefficient C_m leads to a reduction in the achieved enhancement in lift values for different models and reduces the difference between the maximum lift and drag produced by most models. Figure 14 illustrates the enhancement achieved by various models

specially S826-0.1B-0.7S and S826-0.08B-0.8S in improving the aerodynamic performance of the aerofoil by enhancing the lift value up to 76.8% for S826-0.1B-0.7S and 72.5% for S826-0.08B-0.8S, keeping the drag values low as compared to other designed models – achieving almost the same drag

values – and delay the stall angle by 2° for both.

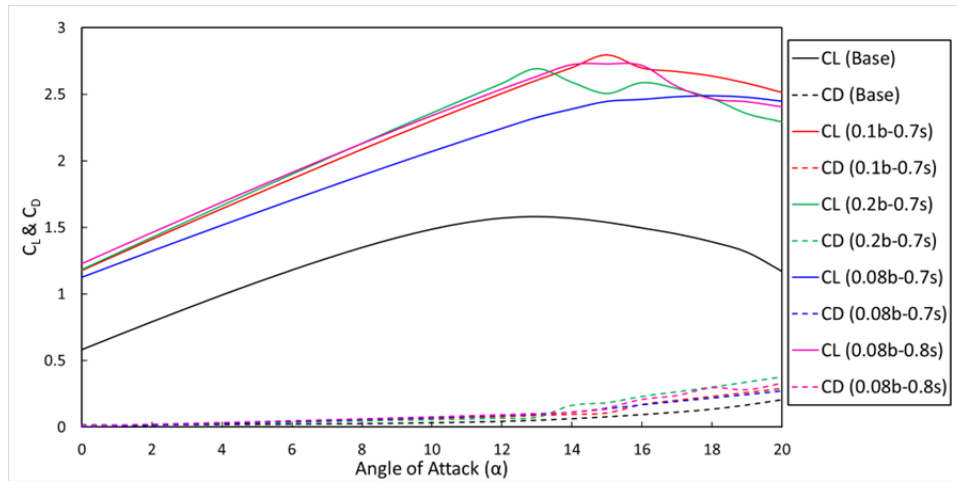
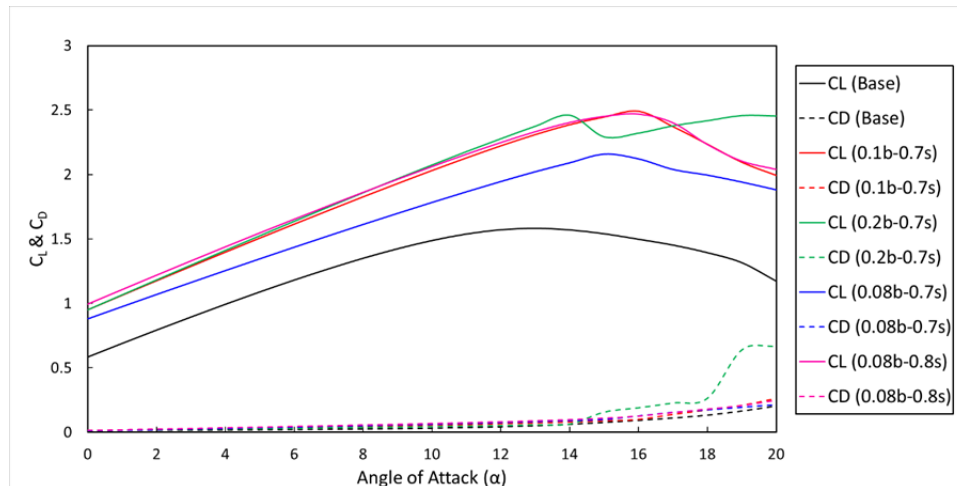


Fig. 8. The effect of ports positions on the lift and drag coefficients when applying a $C_m = 0.16$ at $Re = 6 \times 10^5$ for the main stream.

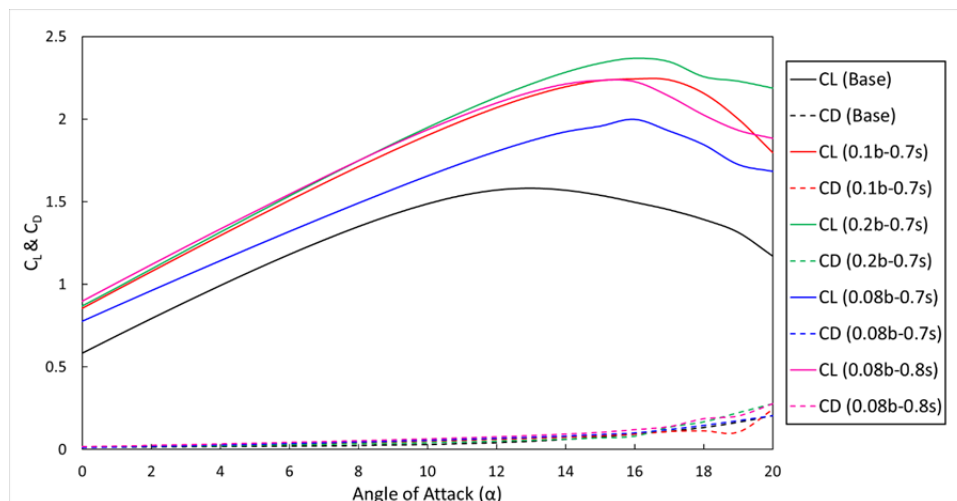
Applying minimal values of momentum coefficient C_m reduces the increase in drag value and allowing to achieve an increase in the lift values close to the achieved values using higher momentum coefficient values. Figure 15 (a) illustrates

of 51.1% and the smallest drag coefficient increase by 11.3%. While Figure 15 (b) shows almost constant drag values for all designed models at all angles of attack compared to the base condition with the ability to increase the value of lift produced by all the designed models when applying a $C_m = 0.05$. If observation is made to Figure 16 (a) and (b) an enhancement in the aerofoil aerodynamic efficiency can be noticed for a wide range of attack angles compared to the other presented models. However, the model S826-0.2B-0.7S shows an irregular response when applying $C_m = 0.05$ as it achieves a delay in stall angle by 3° while achieving a high lift increase with a value of 49.8% and low drag increase by a value of 60% among the considered models at the same operating conditions. The model S826-0.2B-0.7S at $C_m = 0.08$ and $C_m = 0.05$ gives the best aerodynamic enhancement of the presented cases. However, the range of enhancement is longer in the case when applying $C_m = 0.05$, and the enhancement is higher in the case where $C_m = 0.08$ is applied. It is observed that the enhancement in aerofoil performance increases as the C_m value decreases. These findings align with the results of Deng et al [24]. This behavior is attributed to the Kelvin-Helmholtz

the enhancement in lift values and a relatively small increase in drag values for most of the designed models when applying $C_m = 0.08$. Where the model S826-0.2B-0.7S achieves the highest increase in lift coefficient with a value instability phenomenon[25], which occurs when two layers with different velocities or densities interact, forming an interfering layer due to sufficient shear stress between them. This leads to momentum transfer across the interface, resulting in the reattachment of the separated layer on the aerofoil surface. Based on the understanding of the Kelvin-Helmholtz instability phenomenon, when $C_m = 0.5$ and $C_m = 0.4$ are applied, the shear between the interface layers is too strong, forming high instability and complex structures such as vortices. This causes an increase in the pressure difference between the suction and pressure sides of the aerofoil, leading to an increase in lift force accompanied by a high increase in drag force due to the generated vortices. For the cases when $C_m = 0.08$ and $C_m = 0.05$ are applied, the generated shearing at the interface is sufficient to cause the proper amount of momentum transfer between the layers of the mainstream and the CFJ stream, leading to the reattachment of the separated layer without causing high turbulence to create large and strong vortex areas. The application of low C_m values of 0.08 and 0.05 results in a favorable increase in lift force while keeping the increase in drag force minimal.

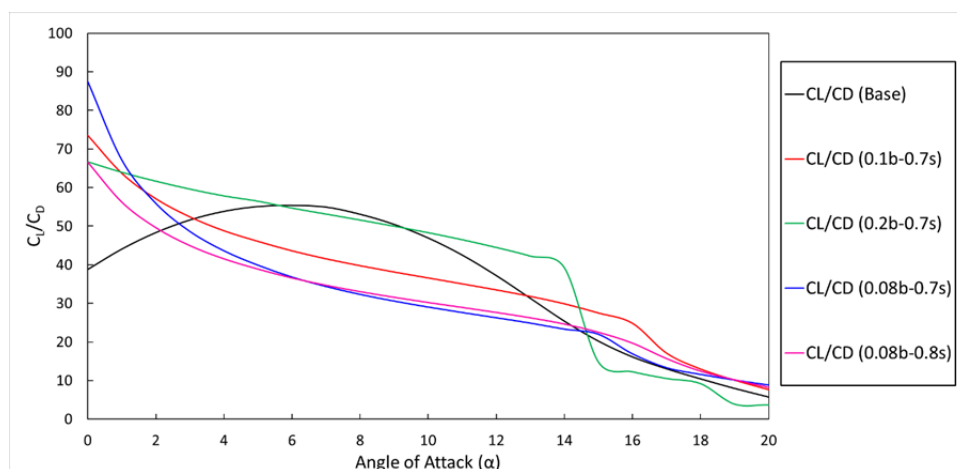


(a)



(b)

Fig. 9. The effect of ports position on the lift and drag coefficients when applying a) $C_m = 0.08$ and b) $C_m = 0.05$ at $Re = 6 \times 10^5$ for the main stream.



(a)

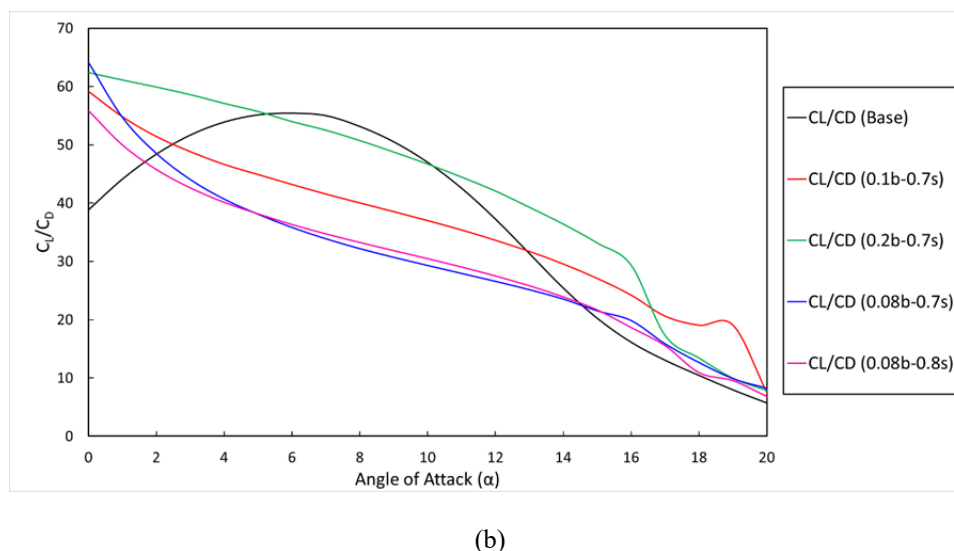


Fig.10. The effect of ports position on the C_L/C_D ratio at a) $C_m=0.08$ and b) $C_m=0.05$ at $Re = 6 \times 10^5$ for the main stream.

5. Conclusion

This study investigates the utilization of suction and blowing techniques as Boundary Layer Control methods and their impact on the performance of the NREL S826 aerofoil used in wind turbine applications. Computational Fluid Dynamics (CFD) is employed to simulate the airflow around the aerofoil at varying suction and blowing flow rates and positions. Six values of the momentum coefficient, C_m (0.05, 0.08, 0.16, 0.2, 0.4, and 0.5), are considered for both suction and blowing. The SST K- ω turbulence model is utilized for turbulence modeling.

One particular model, S826-0.2B-0.7S, emerges as the most efficient among the proposed models. By positioning the blowing port closest to the separation point at high angles of attack, this model achieved a significant increase in the lift coefficient by 49.8%, with a minor increase in drag of up to 60%. Another model, S826-0.2B-0.7S, yielded a lift coefficient increase of 51.1% with a minimal increase in drag coefficient of only 11.3% by applying $C_m = 0.08$.

In order to evaluate the impact of the enhanced lift coefficient on wind turbine applications, the estimated tangential and thrust force coefficients are analyzed along the spanwise direction of an NTNU turbine. The results indicate that increasing the momentum coefficient leads to an increase in both tangential and thrust forces. However, there is a notable increase in the thrust force, particularly near the blade tip. Therefore, it is recommended to implement suction and blowing techniques in sections that are located away from the blade tip.

Acknowledgements

N/A

Author Contributions

All authors were responsible for the conceptualization, validation, resources, data curation, software development, and project administration. They jointly contributed to the methodology, formal analysis, investigation, original draft preparation, review and editing, visualization, supervision, and funding acquisition. All authors have read and agreed to the published version of the manuscript.

Conflict of Interest

The author(s) declared no potential conflicts of interest with respect to the research, authorship, and/or publication of this article.

References

- [1] A. Pourrajabian, S. Rahgozar, M. Dehghan, and D. Wood, "A comprehensive multi-objective optimization study for the aerodynamic noise mitigation of a small wind turbine," *Engineering Analysis with Boundary Elements*, vol. 155, pp. 553-564, 2023. (Article)
- [2] S. K. Aliabadi and S. Rasekh, "Effect of sudden change in free stream velocity on the wind turbine airfoil performance based on boundary element method," *Engineering Analysis with Boundary Elements*, vol. 101, pp. 360-370, 2019. (Article)
- [3] N. R. Hau, L. Ma, D. Ingham, and M. Pourkashanian, "A critical analysis of the stall onset in vertical axis wind turbines," *Journal of Wind Engineering and Industrial Aerodynamics*, vol. 204, p. 104264, 2020. (Article)
- [4] Y. Abdelilah, H. Bahar, T. Criswell, P. Bojek, F. Briens, and P. Feuvre, "Renewables 2020: Analysis and Forecast to 2025," IEA: Paris, France, 2020. (Article)

- [5] S. Sakurai and O. Sakamoto, "Voltage Control of DFIG-based Wind Turbine Generator in Rural Grid," in 2023 12th International Conference on Renewable Energy Research and Applications (ICRERA), 2023, pp. 203-207. (Conference Paper)
- [6] R. Leulmi and L. Salima, "Fuzzy logic controller-based power control of DFIG based on wind energy systems," International Journal of Smart Grid-ijSmartGrid, vol. 8, pp. 74-80, 2024. (Article)
- [7] M. M. Elsakka, D. B. Ingham, L. Ma, and M. Pourkashanian, "Effects of turbulence modelling on the predictions of the pressure distribution around the wing of a small scale vertical axis wind turbine," in Proceedings of the 6th European conference on computational mechanics: solids, structures and coupled problems, ECCM 2018 and 7th European conference on computational fluid dynamics, ECFD, 2018, pp. 3921-3931. (Conference Paper)
- [8] J. A. C. Lipiao, N. M. M. Divina, R. V. M. Santiago, and C. C. Hortinela, "Hybrid J-Type Darrieus and Savonius Vertical Axis Wind Turbine Capable of Harvesting Roadside Wind Energy," in 2022 11th International Conference on Renewable Energy Research and Application (ICRERA), 2022, pp. 1-6. (Conference Paper)
- [9] T. Kara and A. D. Şahin, "Implications of Climate Change on Wind Energy Potential," Sustainability, vol. 15, p. 14822, 2023. (Article)
- [10] M. Bošnjaković, M. Katinić, R. Santa, and D. Marić, "Wind turbine technology trends," Applied Sciences, vol. 12, p. 8653, 2022. (Article)
- [11] M. A. Michon, I. M. Albayati, and A. M. Aliyu, "An Integrated Wind Turbine and Power Grid Mode," in 2023 11th International Conference on Smart Grid (icSmartGrid), 2023, pp. 1-6. (Conference Paper)
- [12] J. S. M. Ali, M. F. Amran, and N. A. Mohamad, "Experimental Study on the Effect of Boundary Layer Control on the Aerodynamics Characteristics of NACA 0021 Aerofoil," Journal of Advanced Research in Fluid Mechanics and Thermal Sciences, vol. 94, pp. 129-137, 2022. (Article)
- [13] V. Kornilov, "Combined blowing/suction flow control on low-speed airfoils," Flow, Turbulence and Combustion, vol. 106, pp. 81-108, 2021. (Article)
- [14] L. Wang, M. M. Alam, S. Rehman, and Y. Zhou, "Effects of blowing and suction jets on the aerodynamic performance of wind turbine airfoil," Renewable Energy, vol. 196, pp. 52-64, 2022. (Article)
- [15] A. Khoshnevis, S. Yazdani, and S. E. Salimipour, "Numerical Investigation of Co-flow Jet Effects on Dynamic Stall of a Thick Airfoil," Fluid Mechanics & Aerodynamics Journal, vol. 9, pp. 167-178, 2020. (Article)
- [16] J. Lei, Q. Liu, and T. Li, "Suction control of laminar separation bubble over an airfoil at low Reynolds number," Proceedings of the Institution of Mechanical Engineers, Part G: Journal of Aerospace Engineering, vol. 233, pp. 81-90, 2019. (Article)
- [17] X. Sun, Y. Xu, and D. Huang, "Numerical simulation and research on improving aerodynamic performance of vertical axis wind turbine by co-flow jet," Journal of Renewable and Sustainable Energy, vol. 11, 2019. (Article)
- [18] Y. Wang and G. Zha, "Study of Reynolds Number Effect for 2D Co-Flow Jet Airfoil at Cruise Conditions," in AIAA AVIATION 2020 FORUM, 2020, p. 2666. (Conference Paper)
- [19] R. Arunraj, K. Logesh, V. Balaji, T. Ravichandran, and G. Yuvashree, "Experimental investigation of lift enhancement by suction-assisted delayed separation of the boundary layer on NACA 0012 airfoil," International Journal of Ambient Energy, vol. 40, pp. 243-247, 2019. (Article)
- [20] Z. Lei and G. Zha, "Lift enhancement of supersonic thin airfoil at low speed by co-flow jet active flow control," in AIAA AVIATION 2021 FORUM, 2021, p. 2591. (Conference Paper)
- [21] S. M. Abul-Ela, Sayed, I. A., Elrefaie, E. M., "Performance Comparative Study of Low Reynolds Number Airfoils Utilized in Vertical-Axis Wind Turbines," presented at the Accepted research paper in Al-Azhar 16th International Conference (AEIC 2024) (Conference Paper)
- [22] J. Bartl, K. F. Sagmo, T. Bracchi, and L. Sætran, "Performance of the NREL S826 airfoil at low to moderate Reynolds numbers—A reference experiment for CFD models," European Journal of Mechanics-B/Fluids, vol. 75, pp. 180-192, 2019. (Article)
- [23] L. Juaristi, "An experimental study of the performance of a wind turbine model with swept blades, for different inflow conditions," NTNU, 2023. (Article)
- [24] Z. Deng, Z. Yang, and W.-L. Chen, "Experimental investigation of the flow control over an airfoil with owl-inspired trailing-edge modification: On the material, length, and spacing sensitivity," Physics of Fluids, vol. 35, 2023. (Article)
- [25] A. Masson and K. Nykyri, "Kelvin–Helmholtz instability: Lessons learned and ways forward," Space Science Reviews, vol. 214, p. 71, 2018. (Article)