 Click here to view linked References

Improving Performance of H-Type NACA 0021 Darrieus Rotor using Leading-Edge Stationary/Rotating Microcylinders: Numerical Studies

W. A. El-Askary[1,†], Massimiliano Burlando[2], Mohamed H. Mohamed[3], Abdelgalil Eltayesh[4]

[1]Department of Mechanical Power Engineering, Faculty of Engineering, Menoufia University, 32511, Egypt

[2]Department of Civil, Chemical, and Environmental Engineering, University of Genoa, Via Montallegro 1, 16145 Genoa, Italy

[3] Mechanical Engineering Dept., College of Engineering and Islamic Architecture, Umm Al-Qura University, PO 5555, Makkah, Saudi Arabia (mhmohamed@uqu.edu.sa)

[4] Mechanical Engineering Department, Benha Faculty of Engineering, Benha University, Benha 13512, Egypt

Abstract

In the current paper, a computational study has been performed to simulate a novel concept in which a fixed/rotating microcylinder is installed upstream of the leading edge of the NACA 0021 airfoil, aiming to enhance the performance of the H-type Darrieus rotor. The idea behind implementing a fixed/rotating microcylinder is to enhance the lift to drag ratio due to increasing near-wall momentum through generated vortices transferring the momentum from the outer flow to the near-wall flow region of the airfoil. Parametric analyses for the microcylinder including its location, size, shape (circular, square and rhombus), and rotation were performed. The commercial Computational Fluid Dynamics (CFD) package ANSYS Fluent was used for solving the Unsteady Reynolds Averaged Navier-Stokes (URANS), and turbulence equations. The code has been based on Finite Volume Method (FVM) with the pressure-based solver developed for low-speed incompressible flows. The semi-implicit method for pressure-linked equation (SIMPLE) was used to solve the discretized equations. The URANS was used with adopting the SST k-ω turbulence model to find out the optimum rotor by utilizing the microcylinder model. An optimization methodology using Response Surface Optimization (RSM) based on Kriging method has been first performed to find the optimum size and location of a circular microcylinder. The optimization study indicated an optimum microcylinder diameter

† Corresponding author (Wageeh El-Askary) Tel.: +201005255817; fax: +20482235695

Present address: Alexandria Higher Institute of Engineering & Technology (AIET), Alexandria, EGYPT

E-mail addresses: wageeh_elaskary@yahoo.com; Wageeh.Elaskary@sh-eng.menofia.edu.eg

(d/C=0.0085313), at an upstream chordwise distance and normal to it 0.070275 and 0.02303 of the chord length (C), respectively. The results showed also that a small static circular microcylinder of d/C=0.009 and installed at 0.075 C from the leading edge (MC5) is an efficient geometry at high regime of tip-speed ratios (TSR ≥ 2.2) rather than lower regime of TSRs. When these superior set of rotating (5 rad/s) microcylinder (MC5) parameters were adopted, significant enhancement of power coefficient (Cp) could be achieved and a considerable improvement of the maximum power coefficient (Cp_{max}) up to 120 % at TSR = 3 appeared. A physical analysis of the flow fields using the contours of vorticity, pressure and turbulence energy has been performed to illustrate the effect of rotating microcylinder for improving rotor performance. The vorticity contours showed the ability of rotating microcylinder for generating a strong vortex structure in its wake for enhancing turbulence production around the blades, which delayed the known strong stall of the blades by diminishing the separation bubble size. This generally improved the blades aerodynamic performance and enhancing the lift-to-drag ratio.

Keywords:

Static/Dynamic microcylinder; Darrieus rotor; CFD; Turbulence models; dynamic stall

Nomenclature			
A	Projected area of the rotor [m]	MC	Microcylinder
AoA	Angle of attack [degree]	Re	Reynolds number
C	Airfoil chord [m]	SST	Shear Stress Transport
Cd	Drag coefficient	TSR	Tip-speed ratio [–]
CFD	Computational Fluid Dynamics	URANS	Unsteady Reynolds Averaged Navier- Stokes
c-GF	Cavity and Gurney Flap	VAWT	Vertical axis wind turbine
Cl	Lift coefficient	V_∞	Wind velocity [m/s]
Cp	Power coefficient	2D	Two dimensional
D	Rotor diameter [m]	3D	Three dimensional
d	Microcylinder diameter [mm]	ρ	Air density [kg/m^3]
GF	Gurney flap	ω	Angular speed [rad/s]

1- Introduction

Renewable energy takes a lot of importance in the last years and the researchers used the solar energy in several interesting applications which can change the lifestyle in world by reducing the usage of the fossil oil [1-5].

Wind energy technologies have been developed rapidly to play a major role in the energy field. However, with the overall energy demand, the use of wind power is still small. Two main problems are facing the utilization of a small- scale wind turbines within the residential areas; the first is the weak, turbulence and unstable air flow within the built environments where they are installed, and the second is small aerodynamic starting torque that results in the difficulty to start the rotor system in the low wind speed sites. Therefore, to harvest a reasonable power output from a small wind turbine located in such an environment, and justify its installation economically, the aerodynamic performance of these turbines must be enhanced. However, the maximum theoretical power coefficient that can be extracted from the wind is 0.593. This limit for the power coefficient is known as the Betz limit [6].

The main reason for the efficiency deterioration of this small-scall wind turbine is the flow separation from blade surface. Separation occurs at suction side of the blade because of changing angle of attack due to the change of wind direction or the turbine rotation. If the air flow past the wind turbine blades could be fully controlled, the efficiency of generated power and thus the energy production would increase by 9 % [7]. For large scale wind turbines, the flow separation is controlled using blade pitch and yaw control while this method is not efficient for small scale wind turbines [7]. Therefore, passive flow control techniques are the most effective ways to control the flow separation on the small-scale wind turbine, thus improving its aerodynamic performance. Some of these passive techniques have been implemented in the last few years such as vortilons, flow vanes, gurney flaps, fixed leading edge slats, aerofoils with protuberances, cavities, passages, slots and microcylinder [8]–[11].

Many studies have been performed in the last years focusing on quantifying the effects of trapped vortex cavity on vertical axis wind turbine (VAWT) performance. Ibrahim et al. [12] performed two-dimensional (2D) numerical calculations to investigate the effect of cavity on the straight bladed Darrieus rotor using URANS simulations. The calculation was conducted for a cavity located on the blade suction side, at 20 % blade chord length from leading edge. The results indicated that the turbine with a cavity blade has higher power coefficient than baseline turbine at low tip speed ratio ($TSR = \omega D/2V_\infty$, in which, D is the rotor diameter, ω is the angular speed of the rotor and V_∞ is the wind velocity). Sobhani et al. [13] studied the effect of dimple location, shape, and size on the VAWT performance using 2D numerical calculations with URANS approach. The results demonstrated that the dimples with a circular shape have superior performance than other shapes of dimple geometries. Also, the performance of the turbine is strongly affected by the dimple locations especially at high TSR. Roshan et al. [14] also investigated the effect of cavity shape, numbers, size and position on the power and torque coefficients of the VAWT using 2D-URANS numerical calculations. It was reported that the turbine with a single cavity near the trailing edge on the upper surface of the blade is more beneficial than blade with double cavity. The maximum improvement in power coefficient was 18% using a single circular cavity with a diameter of 0.08 blade chord length. Eltayesh et al. [15] performed numerical calculations and classical simplified methods (engineering approaches) to improve the performance of Darrieus-Savonius hybrid VAWT with using airfoil dimples.

Later, many studies have focused on others flow control methods, such as slots and jet actuators on VAWTs. The slotted airfoil was considered as a passive flow control, which energizes the boundary layer and postpones flow separation. Belamadi et al. [9] investigated numerically the influence of single slot with different geometrical parameters on aerodynamic performance of a NREL S809 airfoil using 2D calculations. The calculations were performed at an angle of attack (AoA) ranging between 0° to 20°. It was found that superior performance is obtained when the slot outlet is located at 50% of the blade chord, slot inclination angle of 60° and slot inlet to outlet area ratio of 4. Furthermore, the slots efficiency was higher than baseline at AoA between 10° to 20° due to the separation elimination for these locations. While at low AoA, the slot efficiency is less than the baseline airfoil. Mohamed et al. [16] designed and tested numerically the slot location, dimension, and inclination angle on NACA 0018 airfoil at different AoA using 2D static calculations. Subsequently, the optimized slotted airfoil was assessed on a Darrieus VAWT. Their results indicated that the slotted airfoil enhances the turbine performance at low tip speed ratio. This could be attributed to the slotted airfoil assistance of separation delay at high angles of attack. Also, the maximum power coefficient for the slotted airfoil occurs at a tip speed ratio of 2, while the maximum power coefficient for the baseline airfoil is about tip speed ratio of 2.5-3. Ni et al. [17] studied experimentally and numerically the effect of slot width and angle of inclination on the lift and drag coefficients for NACA 634-021 airfoil. The experimental results showed that the slotted airfoil increases lift coefficient and lift to drag ratio by about 58% and 14%, respectively compared to baseline airfoil at AoA greater than 11°. While the baseline airfoil has higher lift to drag ratio than the slotted airfoil at 0° < AoA < 11°. Abdolahifar and Karimian [18] used CFD calculations to investigate the effect of blade height, slot width and outflow locations on the performance of Darrieus VAWT using three dimensional (3D) simulations. The simulation was performed at slot outflow location ranging from 2C/3 to C/6 measured from blade leading edge, for blade height of 0.384, 0.768 and 1.15 m and outflow to inflow width ratio varied from 2/3 to 1/2. Their results demonstrated that the rotor with slotted airfoil located at 2C/3 and slot width ratio of 1/2 has the best performance among all different slot cases. This could be attributed to the elimination of flow separation. Also, the maximum average torque for the slotted blade increases by about 93.4%, 66.1%, and 10.5% at TSRs of 0.5, 1.0, and 1.5, respectively compared to baseline one. Zhang et al. [19] examined numerically the energy capturing for semi-active flapping airfoils power generator at low flow speed to rise the efficiency using leading edge slots considering different inclination angles. The results showed that the energy capture power and efficiency of the semi-active flapping airfoils can be improved by cutting appropriate slots on the leading edge of airfoil. When the inclination angle is from 15° to 17°, better performances are obtained compared with original airfoil without slots.

Many studies on the effect of flow control techniques such as Gurney flap (GF), have received much attention by many authors. The GFs have many advantages such as simple structure, and low production costs. Zhu et al. [20] examined numerically the effect of both dimples and GF on the performance of VAWT with different solidity ratio using CFD calculations. They reported that both GF and GF with dimple located on the suction side of the blade improve the lift coefficient which leads to enhancing the power coefficient of the turbine.

Bianchini et al. [21] also studied the effect of GF on the aerodynamics performance of VAWT using numerical calculations. It was found that the location of GF on the suction side of the blade generates a strong vortex shedding phenomenon. These results are opposite to the results reported in Zhu et al. [20].

Hao et al. [22] investigated numerically the self-adaptive flap on the performance of VAWT. The study was carried out for different flap lengths and locations using CFD calculations combined with fluid–solid interaction. They found that the flow separation generated on the blade leads to self-activated flap and hence blockage of the backflow. Also, the short flap located far from the blade leading edge performed better than the long flap at high TSR. Furthermore, the short flap located near blade leading edge performed better at low TSR.

Liu et al. [23] examined the effect of combined cavity and Gurney Flap (c-GF) on the aerodynamics performance of straight blade VAWT using 2D unsteady numerical calculations. Furthermore, in their study they investigated the movable (c-GF) and compared the results with conventional fixed (c-GF) and original Darrieus rotor. The results indicated that the movable (c-GF) improves the power coefficient (Cp) by about 20% comparing to original Darrieus rotor. Also, tangential force of optimal movable c-GF is higher than fixed (c-GF) in downwind zone at TSRs of 2.33, 2.51, 2.64 and 3.08.

Syawitri et al. [24] performed a numerical study on GF geometry optimization in a full VAWT to consider the effect of rotational turbine blades rather than a single stationary airfoil using CFD Taguchi method. The optimization parameters were GF height, mounting angle, and position from the trailing edge. The results showed that the optimum GF height for the VAWT is 3% C compared to 2% C in case of single stationary airfoil. This could be attributed to the effect of vortices generation due to turbine rotations. Also, they noted that the optimum GF height and mounted angle are 3% C, and 90°, respectively with GF located at the trailing edge. This led to improving Cp to 130.94%, 69.94% and 41.36% at TSRs of 1.44, 2.64 and 3.3, respectively.

Otherwise, extensive investigations on the effect of GF parameters on lift and drag coefficients were carried out using stationary airfoil. For instance, Jain et al. [25] performed computations to study the effect of GF on the aerodynamic performance of NACA 0012 airfoil. The calculations were conducted at six different GF heights ranging from 0.5% to 4% C, seven different GF locations varied from 0% to 20% C measured from the blade trailing edge and seven mounting angles from 30° to 120°. They validated their results against the available experimental data provided by Li et al. [26]. It is reported that the lift coefficient increases with increasing GF height until GF equal to 2% C while above this value, the lift coefficient decreases. Furthermore, it was recommended that the GF should be located about 10% C from blade trailing edge to enhance lift coefficient. Also, reasonable agreement was obtained between the calculations and the experimental measurements up to stall angle.

Other promising passive flow control techniques were recently conducted using microcylinder at the blade leading edge. For instant, Luo et al. [27] assessed numerically the effect of microcylinder locations and size on the passive flow control of a stalled airfoil (NACA0012). The numerical calculations were performed at Reynolds number (Re) of 6×10^6 based on inlet velocity and blade chord length, and at angle of attack AoA varied from 16° to 23°. It was demonstrated that the lift and drag coefficients are highly affected by the distance between the blade surface and the microcylinder. In the case of microcylinder diameter ratio of d/C = 0.01, the optimum spacing was found to be 0.015 of blade chord length. Zhong et al. [28] used CFD analysis to investigate the effect of leading-edge rod of NACA 0018 to control the dynamic stall of VAWT. The findings revealed that the tangential force coefficient in case of airfoil with rod increases with increasing AoA which leads to increase the driving torque and hence increase the output power. The results also indicated that the generated vortices from the rod with large distance from the airfoil is more valuable for transmitted kinetic energy into the boundary layer of the airfoil. This prevented the formation of dynamic stall vortices and the eliminates the flow separation of the airfoil at large AoAs. These results agreed well with the experimental results presented by Choudhry et al. [29], who used 0.95 mm diameter rod (0.95% C) at two positions in front of NACA 0021 airfoil to control the dynamic stall. Extensive investigations have also focused on the effect of microcylinder on the performance of HAWT [30]–[32]. Castelli et al. [33] described and validated computational model against experimental data for a classical NACA 0021 three-bladed rotor. Flow field characteristics were investigated for different tip speed ratio, allowing a comparison among rotor operation at optimum and lower Cp values, for better understanding of rotor performance. Rezaeiha et al. [34] performed URANS simulations with the transition SST turbulence model and found that the method can serve as accurate and reliable CFD simulations of VAWTs at different tip speed ratios. This was noticed from the comparisons with the measurements of Castelli et al. [33].

Asadbeigi et al. [35] purposed CFD studies for optimization, and economic analysis of a Darrieus VAWT for illustration of the aerodynamic design and optimization stages, to suggest the most efficient and economically feasible turbine configuration for the application. They concluded that the three-blade turbine achieves a maximum Cp at the optimal TSR, which registered 14% and 23% power increments compared to the four- and five-blade turbines, respectively. Ansaf et al. [36] performed CFD simulation and optimization using ANSYS Fluent to conduct the Efficiency-Based Design Optimization of an H-Darrieus VAWT coupled with fixed guiding walls surrounding its rotor at different tip speed ratios. Ansaf et al. [36] noticed that among the parameters to control the configuration of fixed guiding walls, wind turbine performance is greatly affected by its design parameters.

Bakhumbsh and Mohamed [37] carried out aerodynamic analyses using ANSYS Fluent software based on finite volumes method and the URANS equations with Shear Stress Transport (k-ω SST) turbulence model for Darrieus turbine (three blades) based on the NACA 0018 airfoil with installed a microcylinder in various locations. Three different microcylinder diameters are examined with a ratio of the microcylinder diameter to the chord (d/C = 0.029, 0.05, 0.065). They concluded that the larger diameter microcylinder located on the suction side or down towards the pressure side of the blade leading edge, causes reduction of the turbine performance.

Syawitri et al. [38] introduced a comprehensive review on using passive flow control devices as performance enhancement of lift-type vertical axis wind turbines and suggested that for evaluation of passive flow control devices, all ranges of tip speed ratios (TSRs) operation conditions must be considered.

The ultimate main goal of the boundary layer control over the wind-turbine airfoil is to enhance the airfoil performance by increasing the lift-to-drag ratio along with stall-delay. The microcylinder is a flow control device that can be implemented in H-type Darrieous rotor to improve its performance. Therefore, the main aim of the present study is to use the leading-edge microcylinders as a passive control technique for boundary layer flow separation, to ensure stall delay and to enhance the lift-to-drag ratio and hence improving the turbine performance. For evaluating turbulence models implemented with URANS equations, validation was first performed against the measurements of Castelli et al. [33] and the computational results from Rezaeiha et al. [34] and Mohamed [39] of the standalone Darrieus rotor, see Table 1.

However, using stationary/rotating microcylinder technique is rarely investigated. Implementing a novel concept of rotating microcylinder instead of a fixed microcylinder as a passive flow control method for improving the performance of a small-scale Darrieous rotor with NACA 0021 airfoil is the main objective of the current study.

2- Numerical procedure

In this study, the Computational Fluid Dynamic (CFD) was used to investigate the effect of microcylinder (MC) on the performance of Darrieus VAWT using two dimensional (2D) unsteady calculations. Different cylinder diameters, locations as well as cylinder shapes are proposed as a passive flow control separation to improve the aerodynamics performance of straight blade NACA 0021 Darrieus rotor.

The turbine blade under investigation was proposed by Castelli et al. [33]. The specification of the Darrieus rotor is listed in Table 2. The microcylinder is located close to the blade leading edge as shown in Figure 1. The corresponding coordinates of the center of microcylinder and its dimensions are listed in Table 3.

The numerical calculations were carried out at inlet wind velocity of 9 m/s corresponding to Reynolds number (Re) of 6.35×10^5 and at inlet turbulence intensity of 5 %. The Re is based on rotor diameter (D), inlet wind velocity (V_∞) and the air properties at freestream temperature 25°C. The performance of turbine is examined at different microcylinder diameter ratio, d/C of 0.009, 0.0113, 0.0167 and 0.025, where d is the diameter of microcylinder, and C is the blade chord length.

The commercial CFD package ANSYS Fluent was used for the simulations to solve the conservation of mass and momentum, and turbulence. The ANSYS Fluent code was based on Finite Volume Method (FVM), in which the fluid domain is divided into discrete control volumes. These individual control volumes allow an iterative solution of the set of partial differential equations for each volume. Finally, linearization of the discretized equations and an approximation solution of the variable is obtained point by point over the entire flow field domain.

The pressure-based solver developed for low-speed incompressible flows was used in the present study. In this solver, equations of continuity, and momentum were solved sequentially (segregated implicit solution algorithm). If the residuals of all variables do not reach the required level, the properties of the flow are updated, and the solution of the equations is repeated. In the segregated solution method non-linear governing equations were linearized to produce a matrix of equations for the dependent variables in every computational cell. Implicit means, for a given variable, the unknown value in each cell was computed using a relation that includes both existing and unknown values from neighboring cells. Therefore, each unknown will appear in more than one equation in the system, and these equations must be solved simultaneously to give the unknown quantities.

The air flow around the rotor is unsteady, and hence the unsteady incompressible Reynolds-averaged Navier–Stokes (URANS) were used in the present computations. In the computational method, the governing equations need to be translated into algebraic equations that can be solved numerically which have been done by means of discretization. The semi-implicit method for pressure-linked equation (SIMPLE) was used to solve the discretized equations. It was chosen to use first order upwind discretization for the initial part of the solution to help the convergence then it was set to second order upwind to obtain higher-order accuracy.

Furthermore, the sliding-mesh technique was employed to simulate the rotational motion of the wind turbine rotor, see El-Askary et al. [8].

The general conservation formulation for sliding meshes can be expressed as:

$$\frac{d}{dt}\int \rho\phi dV + \int \rho\phi(\vec{u} - \vec{u}_{mesh}).d\vec{A} = \int \Gamma\nabla\phi.\overrightarrow{dA} + \int S_{\phi}\, dV \quad (1)$$

ϕ represents the components of relative velocity in all directions, $\vec{u}$ is the velocity vector, $\vec{u}_{mesh}$ represents the velocity of moving mesh, Γ is the diffusion coefficient, and S_{ϕ} is added to represent any source term of ϕ.

The conservation of mass equation is:

$$\nabla.(\vec{u} - \vec{u}_{mesh}) = 0 \quad (2)$$

For sliding mesh all computations were performed without mesh deformation, hence the cell volume remains unchanged, $(\Delta V^{n+1} = \Delta V^{n})$. The unsteady term in the general conservation equation (1) becomes $\frac{d}{dt}\int \rho\varphi dV = \frac{\{(\rho\varphi)^{n+1} - (\rho\varphi)^{n}\}\Delta V}{\Delta t}$; this leads to $\int \vec{u}_{mesh}.d\vec{A} = 0$. The solution of Eq. (1) for the sliding mesh motion was inherently unsteady with moving mesh.

The entire computational domain was divided into two different zones, the first is the stationary zone and the second is the rotating cylindrical zone containing the rotor blades including the microcylinders (rotating diameter is 1.5D) as shown in Fig. 2. Interface boundary condition was used to merge the separated zones, for more details see El-Askary et al. [8]. The inlet plane of the computational domain was placed at 10 rotor diameters, while the outlet plane was placed at

25 rotor diameters downstream, and the side boundaries were set at 10 rotor diameters. All dimensions were measured from the center of the turbine and were previously verified by Rezaeiha et al. [34], [40]. The final computational domain of the current calculations reads 35D × 20D. The boundary conditions at the inlet, outlet and side planes were set as uniform velocity, zero static pressure and symmetry respectively. The interface between the rotating and fixed zone was defined using sliding mesh method. All surfaces of the blade and microcylinder walls were considered in the 'no-slip' state. The convergence threshold for all the parameters were set at a value of 1×10^{-5}. The air density and viscosity are set as a default. The final computational domain dimensions as well as boundary conditions are depicted in Fig. 2.

The 2D-computational domain and mesh generation were created using ANSYS meshing. Hybrid mesh topology was generated for the domain. An unstructured mesh scheme with triangular elements was applied to the entire. The interfaces between rotating and fixed zones were meshed using the same mesh sizes. The final 2D computational mesh is shown in Figure 3. A structure O- type mech was used in the boundary layer near the solid walls of the turbine blades and cylinder to allow fine grid resolution. The boundary layer had 12 rows with the first layer from the wall is 0.001 mm and the layer growth factor is 1.2. The maximum and average dimensionless wall distance y+ values are about 1.8 and 0.47, respectively.

Different turbulence models were implemented with solving RANS equations to calculate the turbine performance and validating the experimental results of Castelli et al. [33]. Three turbulence models were considered namely, RNG k- ε with standard wall function [41], Realizable k- ε with standard wall function [42] and SST k-ω of Menter [43]. The calculations were performed on the standalone Darrieus rotor. The calculated power coefficient (Cp) obtained with the different turbulence models and from Rezaeiha et al. [34] and Mohamed [39] are given in Figure 4. The coefficient of power has been estimated based on the relation $C_p = T\omega/(1/2\rho A V_\infty^3)$ in which, T is the mechanical torque of the rotor, A is the projected area of the rotor (facing the wind), and ρ is the air density. The turbulence models give different results in view of comparisons with the measurements of [33]. The calculation with the RNG and Realizable k- ε turbulence models give good predications at low tip-speed ratio (TSR) until 2.2. At high TSR greater than 2.2 the RNG and Realizable k- ε show underprediction with the experimental measurements.

The discrepancies between the numerical and experimental results are caused by the absence of shafts, and arms, as well as the experimental uncertainties, the numerical study hypothesis, and the modeling assumptions which include the absence of friction or mechanical loss. However, the calculated power coefficient (Cp) using the SST k-ω turbulence models shows a reasonable agreement with the measurement over most turbine TSR. The SST k-ω turbulence model is then used to close the RANS equations in all coming computations, as recommended by El-Askary et al. [8] and Rezaeiha et al. [34], who studied the same rotor profile. Therefore, all the coming calculations were performed using the SST k-ω model.

The grid dependency was examined for the Darrieus rotor without/with different microcylinder shapes. In the case of Darrieus turbine with different microcylinders the grid study was performed for microcylinder installed at $\frac{H}{C} = 0.1$ from the blade leading edge and $\frac{V}{C} = 0.0$ (MC1) with microcylinder diameter ratio of $\frac{d}{C} = 0.025$. The effect of total grid size on the turbine Cp was performed at TSR of 2.5. Four different grids were used to check the grid dependency with total number of cells ranging from about 190×10^3 to 313×10^3. Figure 5 demonstrates that there is no significant change in the turbine Cp when the total mesh increases above 260 - 272×10^3.
The settings made for the inlet velocity and turbulence intensity are the same to those of Castelli et al. [33]. The time step is set corresponding to 1^o of the turbine rotation, which is consonant with the acceptable results from several researchers in the literature, see Rezaeiha et al. [34] and Wong et al. [44]. The number of iterations per time step is 20. The calculations were performed till the change in the torque coefficient for two successive turbine rotations was about 0.0015. The obtained residuals of all variables are below 10^{-5} except for the continuity equation which was about 10^{-4}. All simulations were performed on an HP Z800 (Intel Xeon CPUX5650, 2.66 GHz, 2 processors) workstation with 24 cores available for parallel calculations and 32 GB RAM.

3- Computational Results

3.1 Effect of microcylinder locations

CFD-based optimization is a common tool to determine the best configurations or flow conditions in renewable energy applications [36]. It is widely adopted with several techniques aimed to vertical axis wind turbine optimization [45]. In the present study, the searching for optimum size and location of the microcylinder was first performed based on the optimization study of Ranjbar et al. [45]. Optimization methodology is the discipline of fine-tuning an objective function which is dependent in nature on a particular set of parameters while remaining within a pre-stated set of constraints [36]. Different efforts using optimization methodology have been implemented to enhance the generated power of wind rotor including surrounding geometries [45]. The optimization of Ranjbar et al. [45] was aimed to obtain a maximum approaching velocity to an H-Darrieus VAWT installed inside the optimized duct. Ranjbar et al. [45] developed a code to allow numerical steps including generating the meshes, changing the geometries, and setting up the solver simultaneously. Using that method, deep dynamic stall phenomena have been captured perfectly with leading-edge vortex generation and delaying the vortex separation.

In the present study, an optimization methodology has been performed to find the appropriate size and location of a circular microcylinder for enhancing the performance of H-Type NACA 0021 Darrieus Rotor. The optimization was performed using parameters and design points tools which are available in ANSYS workbench.

In order to specify the optimum microcylinder diameter ratio (d/C) and its location (H/C and V/C), the optimization technique was used using Response Surface Optimization (RSM) which is available in Ansys package. The first step of the optimization is to set the Design of Experiment (DOE). The Latin Hypercube Sampling Design (LHSD) method is used to set the samples (refinement points) in the DOE [46]. In the current study, the microcylinder diameter ratio d/C was varied from 0.008 to 0.025, the microcylinder location upstream the leading edge of the rotor blade along chord line was ranged from H/C of 0.015 to 0.1 and normal to chord line V/C from -0.05 to 0.05, see Fig. 6. However, the refinement was only performed at TSR of 3 to compute the optimum Cp. The stochastic interpolation Kriging method was used to interpolate the data of the DOE during the RSM. This method was used in a previous study which related to vertical axis wind turbine [45, 46]. However, to improve the accuracy of the Kriging method the number of refinement points must be increased. In the current study the number of refinement points used is 16 as illustrated in Table 4, and the results from Kriging method are shown in Fig. 7. Figure 7 illustrates the surface responses of the model formed by altering the independent parameters (H/C and V/C) and the microcylinder size (d/C), while keeping a constant tip-speed ratio (TSR=3.0). Based on this optimization, the optimum microcylinder size and location were that have the highest rotor performance.

Finally, the optimization technique was performed using Nonlinear Programming by Quadratic Lagrangian (NLPQL) which supports a single output parameter objective. The final optimum microcylinder size (d/C) at optimum location (H/C and V/C) using NLPQL is listed in table 5. Table 5 illustrates the data extracted from the program for three chosen optimized locations for the optimum microcylinder size (d/C=0.0085313), in which point 1 is considered the optimum location for the optimum microcylinder size.

For satisfying the optimization strategy, six different locations namely, MC1, MC2, MC3, MC4, MC5, and MC6 are chosen corresponding to $\frac{H}{C} = 0.01, 0.02, 0.025, 0.05, 0.075$ and 0.1, respectively for studying the rotor performance at different tip speed ratios (1.69≤TSR≤3.5).

The effect of circular microcylinder implementation in front of the blade leading edge at various locations on the performance of Darrieus rotor is first investigated to find the appropriate location. Six different locations are designed namely, MC1, MC2, MC3, MC4, MC5, and MC6, corresponding to $\frac{H}{C} = 0.01, 0.02, 0.025, 0.05, 0.075$ and 0.1, respectively. The circular microcylinder of diameter ratio $\frac{d}{C} = 0.009$ was first installed along the chord line without variation in the normal to it ($\frac{V}{C} = 0$). To also check the ability of the optimized microcylinder at all TSR, the rotor with the optimized microcylinder (d/C=0.0085313) installed at H/C=0.070275 and V/C=-0.02302 (candidate point 1) has been simulated against the standalone rotor and a rotor with the other mentioned MC1, MC2, MC3, MC4, MC5, and MC6. The difference between the results of optimized microcylinder and MC5 is less than 0.2% at TSR=3, see Fig. 8. However, the small negative impact of MC5 on the turbine Cp at low TSR range ($TSR \leq 2.25$) disappears with using the optimum microcylinder. Figure 8 illustrates also that the presence of microcylinder (MC4 and MC5) has a small negative impact on the turbine Cp at low TSR range ($TSR \leq 2.25$), while this range is increased for the others as shown in the figure. The reduction of the rotor performance at low tip-speed ratio with implementing microcylinder is due to the laminar wake behind the microcylinder which extracts turbulence energy from the near wall boundary-layer flow at the blade surface, which causes a reduction of the generated lift force. With increasing the tip-speed ratio, the performance with the presence of microcylinder appears better than that of the stand-alone rotor, depending on the location of microcylinder. The figure indicates that the implementation of very close microcylinder MC1 (the nearest one to the blade leading edge) has the worst rotor performance because of its blockage effect on the blade boundary layer. With further increasing the distance of the microcylinder location, the power coefficient of the rotor increases at all tip-speeds. The microcylinder MC3 appears to the first location causing some improvements at high tip-speed ratio ($TSR > 2.7$). There is a high improvement in the performance a long wide range of tip-speed ratio is caused by MC5 in the range 2.2≤TSR≤3.5. However, the presence of MC5 has certainly a positive impact on airfoil attacking the flow, which is responsible for the reduction of separation and hence increasing the generated lift to drag value. Accordingly, the location of microcylinder at H/C= 0.075 and V/C= 0.0 (MC5) is set as the best location along chordwise direction. Further increasing the microcylinder locations to MC6 still leads to improvement of the rotor performance but lower than that generated with using MC4 and MC5. However, MC2 improves the performance in the range of 2.4≤TSR≤3.3, after that the performance decreases sharply, because of its energy dissipation at the high tip-speed ratio.

After that the location of microcylinder was changed for different diameters along the vertical direction to search the best cylinder location along the vertical to chordwise direction. Two different locations were examined: the first one was located upper the chordwise at $\frac{H}{C} = 0.075$ and $\frac{V}{C} = 0.05$ (MC7) and the second one was placed lower the chordwise at $\frac{H}{C} = 0.075$ and $\frac{V}{C} = -0.05$ (MC8). The performance of Darrieous rotor without/with circular microcylinder ($\frac{d}{C} = 0.009$) located at different locations (MC5, MC7 and MC8) is presented in Fig. 9 (a). The improvement in wind rotor Cp_{max} is 12.3 %, 7.6 % and 4.62 % for the microcylinder locations of MC5, MC7 and MC8, respectively at TSR of 3. These findings also indicate that superior performance of the wind rotor is obtained for the microcylinder location of MC5 in the range 2.2≤TSR≤3.5. However, MC7 and MC8 have the same behavior, in which the performance is only improved in the range $2.4 \leq TSR \leq 3.2$. This may be due to the increased drag force on the blade. The microcylinders MC5, MC7 and MC8 were further examined using circular microcylinder (d/C=0.0113, 0.0167 and 0.025) to evaluate the performance using different sizes, see Fig. 9 (b), (c), and (d), respectively. For the case d/C of 0.0113, Fig. 9 (b) indicates that MC5 improves the performance in the range $2.3 \leq TSR \leq 3.5$, while MC7 and MC8 have the same behavior and delay the period of improvement to be in the range $2.8 \leq TSR \leq 3.5$. The visible reduction of the performance in the lower tip-speed ratio is due the extraction of the wall-boundary layer energy, which is responsible for the reduction of generated lift force. Figure 9 (c) illustrates that installing microcylinder with d/C of 0.0167 (MC5) causes some improvement of the performance in the range $2.65 \leq TSR \leq 3.3$, but MC7 reduces this improvement range to be $2.85 \leq TSR \leq 3.5$. Implementing the microcylinder MC8 cause reduction of the rotor performance except in the range $2.9 \leq TSR \leq 3.1$, in which the performance of modified rotor with microcylinder approaches the performance of the standalone rotor. With changing the diameter ratio to be $\frac{d}{C} = 0.025$, MC5 has the same performance of that without microcylinder in the range $2.45 \leq TSR \leq 3.0$, see Fig. 9 (d). Around that, the performance of MC5 becomes lower that of the standard rotor (without microcylinder).

However, MC7 and MC8 have the worst performance, respectively. Increasing the cylinder diameter leads to significant reduction in turbine power for all TSR.

3.2 Effect of Microcylinder Diameter

Figure 10 illustrates the effect of microcylinder diameter variation (d/C= 0.009, 0.0113, 0.0167 and 0.025) on the performance of rotor without/with the presence of microcylinder MC5, MC7 and MC8, respectively compared with the optimum microcylinder. It can be concluded that, the existence of microcylinder at MC5 with d/C= 0.009 still approaches the performance with presence of the optimum microcylinder, in which it makes a significant contribution to improve the turbine performance in the range $2.18 \leq TSR \leq 3.5$ as illustrated in Fig. 10 (a). The improvement is reduced in the range of $2.3 \leq TSR \leq 3.5$ when using a microcylinder with d/C= 0.0113, while there is more reduction of performance in the range of $2.65 \leq TSR \leq 3.3$ when using a microcylinder with d/C= 0.0167.

Figure 10 (b) shows that using a microcylinder (MC7) with d/C=0.009 is the best choice in the range $2.45 \leq TSR \leq 3.2$, while d/C=0.0113 reduces the range of modification to be in the range $2.8 \leq TSR \leq 3.5$. Further increase of microcylinder (d/C=0.0167) leads to narrow range of modification to be $2.85 \leq TSR \leq 3.5$. When increasing the diameter (d/C=0.025), the performance becomes the worst one because of the blockage effect of the large microcylinder. Using a microcylinder (MC8) with d/C=0.009 is also the best choice in the range $2.45 \leq TSR \leq 3.2$ as seen in Fig. 10 (c). The microcylinder MC8 with d/C=0.0113 reduces the range of modification to be in the range $2.8 \leq TSR \leq 3.5$, while d/C=0.0167 approaches the performance of the standalone rotor in the range $2.9 \leq TSR \leq 3.1$. Increasing the diameter (d/C=0.025) leads to strong reduction of the performance. Compared with MC7 and MC8, the performance of wind rotor with presence of the optimum microcylinder is the best one.

3.3 Effect of microcylinder shape

Further examination was performed to evaluate the effect of installing microcylinder with different shapes on the performance of the Darrieus rotor. The calculations were performed for rotor with installed microcylinder at MC5 location which has significant contribution to the Cp. Two other shapes were investigated namely, square microcylinder and rhombus (with equal diagonals) microcylinder. The d/C of all studied shapes was 0.009. Figure 11 shows that the circular microcylinder improves the performance in the range $2.2 \leq TSR \leq 3.5$, and the square microcylinder improves the performance in the range $2.5 \leq TSR \leq 3.2$, while the rhombus one improves the performance in the range $2.35 \leq TSR \leq 3.3$. The comparison still shows that the circular microcylinder is the best choice in view of the improvement of rotor performance.

3.4 Effect of cylinder rotation

In this section the effect of rotating circular microcylinder (MC5) with 5 rad/s (in the same direction of Darrieus-rotor rotation) on the performance of the Darrieus rotor was examined. Figure 12 shows as discussed previously that the (fixed) circular microcylinder improves the performance in the range $\mathbf{2.2 \leq TSR \leq 3.5}$, while the rotating microcylinder enhances the performance for all ranges of TSR compared with the rotor with/without fixed microcylinder. This improvement is based on the generated turbulence in the wake of rotating microcylinder, which enhances the turbulence energy of the near-wall boundary layer of the rotor blade as well as a reduction of the pressure drag on the leading edge of the rotor blade and hence expected enhancement of the generated lift-to-drag ratio. This is clearly visible and discussed in Figs. 13-15.

The dynamic stall of rotor blade is an intrinsic specification of VAWTs. This phenomenon decreases the power coefficient of the turbine significantly [47-48]. The blade of rotor under dynamic stall phenomena causes deformation of the wake behind the rotor blade, which influences the performance of the downstream blades in the rotor [47]. Dynamic stall strongly affects the generated lift and drag forces. It occurs due to a large angle of attack variation during turbine rotation. Fig. 13 shows the non-dimensional vorticity ($\frac{\omega\, C}{V_\infty}$) contours around the blade for standalone rotor, and rotor with rotating microcylinder at different azimuth positions and its corresponding angle of attack angles. Take a single blade in consideration and start from azimuth angle of 335 degrees (corresponding to attack angle 25 degrees). When the azimuth angle is 360 degrees, the attack angle equals to zero degree and with further rotation of the blade the angle of attack becomes negative as shown in Fig. 13. However, due to rotation of the rotor, the flow fields around the blade and the upper as well as the lower boundary layers are unsymmetrical.

There is relatively strong vortex structure on the blade surface facing outward flow with respect of the turning circle of the rotor at attack angle 25 degrees. This is because the blade has strong unsymmetrical flow fields. For the original blade of standalone rotor without microcylinder, the separation bubble on the trailing-edge suction surface gradually expands at first to form a large recirculation region as AoA decreases from 25 to 20, and then rapidly decreases as AoA further decreases. This rapid change of bubble size causes the sudden drop of lift and it is known as the heavy stall as presented in [27].

For all positive attack angles, the presence of rotating microcylinder causes enhancement of vortex structure near the leading edge of the blade, and this is responsible for generating more turbulence on the upper surface boundary layer and causes reduction of the recirculation eddy size, see Fig. 13 (a)-(d). Arriving azimuth angle of 360 degrees, the flow is seen to be symmetrical and fully attached to the blade walls as previously seen in [12].

When the considered blade moves to azimuth angle 10 degrees, as shown in Fig. 13 (h), the positive vortex at the inside part of blade squeezes to the trailing edge and the trapped vortex reversed direction. The negative vortex near the trailing edge part decreases but the shedding positive vortex near the trailing edge of the blade becomes thicker with large elongation of vortex after the trailing edge with the appearance of separation bubble on the inner side of the

blade. The presence of the microcylinder still strongly generates vortex flow ahead of the leading edge of the blade with feeding turbulence energy to the inner-side boundary layer of the blade.
For the azimuth angles greater than 10 degrees, large vortex covers the whole blade's inner side of the blade with the presence of recirculation zone for all studied cases. The flow is seen to be fully separated from the inner side of the blade and a strong vortex is shedding from the trailing edge for the case of standalone rotor, while small recirculation size appears near the leading edge of the blade for the case of rotor with microcylinder.
At this point, the pressure distribution at inner side and outer side after middle part of the blade is expected to be very close, and only the leading edge of blade has work capability because of the pressure difference near the leading edge. The presence of microcylinder starts to activate the outer wall with turbulence sufficient to prevent any expected separation with the attacked flow with more rotation of the blade. A reduction of the leading-edge pressure of rotor with rotating microcylinder is noticed also from the distributions of the pressure contours, see Fig. 14. This of course reduces the drag force generated on the airfoil and enhances the generated power of the rotor.

Another interesting finding is drawn from Fig. 15, in which the presence of stationary microcylinder produces some increase of the turbulence kinetic energy approaching the blade of the rotor. This increases the turbulence level in the near-wall boundary layer. The figure also shows that the presence of rotational microcylinder ahead of the leading edge of the rotor blade produces more turbulence than that of the blade with stationary microcylinder and hence enhancement of the generated power.

The coefficients of lift and drag were also derived, as presented in the following equations:

$$Cl = \frac{Lift\ Force}{\frac{1}{2}\rho V_\infty^2 H_t D} \tag{3}$$

$$Cd = \frac{Drag\ Force}{\frac{1}{2}\rho V_\infty^2 H_t D} \tag{4}$$

Where $H_t D$ represents the frontal area of the rotor with H_t as the rotor height. The ratio Cl/Cd is known as the lift to drag ratio.

For the fixed/rotating microcylinder MC5 with d/C=0.009, the angular positions at the quasi-steady state revolution of the lift coefficient, drag coefficient and lift to drag ratio were plotted in

Figs. 16, 17, 18 and19 for TSR values of 2, 2.5, 3, and 3.5, respectively. The results indicate a minimal difference in the instantaneous Cl, Cd and Cl/Cd for the two cases (stationary/rotating microcylinder). The fluid flow is becoming highly unstable and turbulent when the microcylinder rotates with constant angular speed (5 rad/s). The cyclical nature of the curves for all cases are kept throughout the simulation.

In addition, rotating microcylinders could increase the lift-to-drag generation compared with the original rotor (without microcylinder) around the best performance of rotor due to the reduction of pressure drag associated with expected dynamic stall vortex caused by the reduction of apparent thickness of the aerofoils in the presence of leading-edge microcylinder. Hence, this characteristic can be beneficial for lift-type vertical axis wind turbine (Darrieus rotor) as it can improve the time-averaged lift-to-drag ratio, see Fig. 20.

4- Conclusions

The present URANS computational study discusses the effect of static/dynamic (fixed/rotating) microcylinder on the performance of an H-type Darrieus rotor with NACA 0021 airfoil. Previous published measurements and computations of a standalone H-Type NACA 0021 Darrieus rotor were first used to verify the computational results using three different turbulence models namely, RNG k-ε, Realizable k-ε and SST k-ω with clear superiority of SST k-ω turbulence model over the others. The computational study was then extended based on SST k-ω turbulence model for static/dynamic microcylinder. The diameter and the location of the microcylinder installed upstream the leading-edge of the rotor blade, were the main parameters to be considered in this investigation. So, an optimization methodology using Response Surface Optimization (RSM) which is based on Kriging method has been first introduced to find the optimum size and location of a circular microcylinder required to produce the best rotor performance.

Different parameters including the locations, the size, the shape (circular, square and rhombus), and the rotation of microcylinder (5 rad/s) were also implemented to improve performance of rotor. The dynamic stall as an intrinsic phenomenon, which decreases the power coefficient of the turbine significantly, has been also discussed through simulating non-dimensional vorticity contours at different angle of attack of the blade during the rotor rotation with/without the presence of the circular optimized microcylinder.

The main conclusions that can be extracted from the present work are:

1- The SST k-ω turbulence model can be considered as a suitable turbulence model for computing the performance H-type Darrieus rotor.
2- The optimum dimensionless microcylinder diameter of 0.0085313 installed at an upstream dimensionless chordwise distance and normal to it 0.070275 and 0.02303, respectively can produce the best rotor power.
3- The microcylinder of diameter 0.009 of chord length located along the chord line upstream of the leading edge at a distance 0.075 chord length (named as MC5) approaches the performance of the optimum microcylinderand is found to be the best one compared with all studied cases from the point of view of the rotor performance.
4- The dynamic circular microcylinder (MC5) is only effective at high regime of tip-speed ratios (TSR $\geq$ 2.2) rather than lower regime of TSRs.
5- The rotating microcylinder MC5 installed at distance 0.075 chord length upstream of the leading edge of the rotor blades produces significant enhancement of H-type Darrieus rotor performance due to the increase of lift-to-drag ratio around the optimal operation of the rotor. A considerable improvement of the maximum power coefficient (Cp_{max}) up to 120 % at TSR = 3 is achieved.
6- The rotating microcylinder is able to generate a strong vortex structure in its wake for enhancing turbulence production around the walls of the rotor blades.
7- Using such a rotating microcylinder, the strong stall of the blades can be delayed with diminishing the size of separation bubble on the blade suction surface. This is responsible for improving aerodynamic performance of the rotor blades and enhancing the generated lift-to-drag ratio.

Nevertheless, further studies of a real Darrieus-rotor configuration are still needed to confirm the ability of leading-edge (stationary/rotating) microcylinder on the improvement of Darrieus rotor and its capability at all regimes of TSRs also needs to be considered.

Moreover, how the rotating microcylinders are integrated to the blade of Darrieus rotor in real configuration must be further studied so the structural addition to accommodate rotating microcylinders on the blade of Darrieus rotor does not add significant mass and design complexity.

Acknowledgment

The third author would like to thank the Deanship of Scientific Research at Umm Al-Qura University for supporting his work by Grant Code #23UQU4361231DSR019.

References

[1] Ssennoga Twaha, Mohd HafiziIdris, Makbul Anwari, Azhar Khairuddin, "Applying grid-connected photovoltaic system as alternative source of electricity to supplement hydro power instead of using diesel in Uganda," Energy, Volume 37, Issue 1, January 2012, pp. 185-194. https://doi.org/10.1016/j.energy.2011.11.051

[2] M.M Salah El-Din, "On the optimization of solar-driven refrigerators," Renewable Energy, Volume 20, Issue 1, May 2000, pp. 87-93. https://doi.org/10.1016/S0960-1481(99)00090-7

[3] A.A. Abdullah, H.M. Fallatah, K.A. Lindsay, M.M. Oreijah, "Measurements of the performance of the experimental salt-gradient solar pond at Makkah one year after commissioning," Solar Energy, Volume 150, 1 July 2017, Pages 212-219. https://doi.org/10.1016/j.solener.2017.04.040

[4] A.M. Abdel Dayem, Abdullah AlZahrani, "Psychometric study and performance investigation of an efficient evaporative solar HDH water desalination system," Sustainable Energy Technologies and Assessments, Volume 52, Part A, August 2022, 102030. https://doi.org/10.1016/j.seta.2022.102030

[5] Mohamed K. Hassan, Ibrahim M. Alqurashi, Ahmed E. Salama, Ahmed F. Mohamed, "Investigation the performance of PV solar cells in extremely hot environments," *Journal of Umm Al-Qura University for Engineering and Architecture* **13**, pages 18–26 (2022). https://doi.org/10.1007/s43995-022-00005-x

[6] J. G. M. and A. L. R. J.F. Manwell, *Wind Energy Explained: Theory, Design and Application*, vol. 111, no. 479, John Wiley & Sons Ltd, 2002. doi: 10.1192/bjp.111.479.1009-a.

[7] G. P. Corten, *Flow Separation on Wind Turbines Blades*. 2001. [Online]. Available: https://ui.adsabs.harvard.edu/abs/2001PhDT.......287C

[8] W. A. El-Askary, I. M. Sakr, M. A. Kotb, and A. M. Abdelsalam, "Enhancing performance of small capacity horizontal axis wind turbine using grooved linearized-chord blades," *J. Energy Resour. Technol. Trans. ASME*, vol. 145, no. 4, pp. 1–13, 2023. doi: 10.1115/1.4055204.

[9] R. Belamadi, A. Djemili, A. Ilinca, and R. Mdouki, "Aerodynamic performance analysis of slotted airfoils for application to wind turbine blades," *J. Wind Eng. Ind. Aerodyn.*, vol. 151, pp. 79–99, 2016. doi: 10.1016/j.jweia.2016.01.011.

[10] M. Moshfeghi, S. Shams, and N. Hur, "Aerodynamic performance enhancement analysis of horizontal axis wind turbines using a passive flow control method via split blade," *J.*

Wind Eng. Ind. Aerodyn., vol. 167, August 2017, pp. 148–159, 2017. doi: 10.1016/j.jweia.2017.04.001.

[11] S. Beyhaghi and R. S. Amano, “A parametric study on leading-edge slots used on wind turbine airfoils at various angles of attack,” *J. Wind Eng. Ind. Aerodyn.*, vol. 175, April 2018, pp. 43–52, 2018. doi: 10.1016/j.jweia.2018.01.007.

[12] A. A. Ibrahim, A. M. R. Elbaz, P. F. Melani, O. S. Mohamed, and A. Bianchini, “Power augmentation of Darrieus wind turbine blades using trapped vortex cavity,” *J. Wind Eng. Ind. Aerodyn.*, vol. 223, April 2022, p. 104949, 2022. doi: 10.1016/j.jweia.2022.104949.

[13] E. Sobhani, M. Ghaffari, and M. J. Maghrebi, “Numerical investigation of dimple effects on darrieus vertical axis wind turbine,” *Energy*, vol. 133, pp. 231–241, August 2017. doi: 10.1016/j.energy.2017.05.105.

[14] M. Y. Roshan, J. Khaleghinia, M. E. Nimvari, H. Salarian, “Performance improvement of Darrieus wind turbine using different cavity layouts,” *Energy Convers. Manag.*, vol. 246, 15 October 2021, p. 114693, 2021. doi: 10.1016/j.enconman.2021.114693.

[15] A. Eltayesh, F. Castellani, F. Natili, M. Burlando, and A. Khedr, “Aerodynamic upgrades of a Darrieus vertical axis small wind turbine,” *Energy Sustain. Dev.*, vol. 73, pp. 126–143, Apr. 2023. doi: 10.1016/j.esd.2023.01.018.

[16] O. S. Mohamed, A. A. Ibrahim, A. K. Etman, A. A. Abdelfatah, and A. M. R. Elbaz, “Numerical investigation of Darrieus wind turbine with slotted airfoil blades,” *Energy Convers. Manag. X*, vol. 5, January 2020, p. 100026, 2020. doi: 10.1016/j.ecmx.2019.100026.

[17] Z. Ni, M. Dhanak, and T. Su, “Improved performance of a slotted blade using a novel slot design,” *J. Wind Eng. Ind. Aerodyn.*, vol. 189, June 2019, pp. 34-44, 2019. doi: 10.1016/j.jweia.2019.03.018.

[18] A. Abdolahifar and S. M. H. Karimian, “A comprehensive three-dimensional study on Darrieus vertical axis wind turbine with slotted blade to reduce flow separation,” *Energy*, vol. 248, 1 June 2022, p. 123632, 2022. doi: 10.1016/j.energy.2022.123632.

[19] J. Zhang, Y. Yan, P. Xie, L. Jiang, and J. Zhu, “Effect of leading-edge slot on energy capture performance of semi-active flapping airfoil,” *Energy Reports*, vol. 8, pp. 5225–5234, 2022. doi: 10.1016/j.egyr.2022.03.193.

[20] H. Zhu, W. Hao, C. Li, and Q. Ding, “Numerical study of effect of solidity on vertical axis wind turbine with Gurney flap,” *J. Wind Eng. Ind. Aerodyn.*, vol. 186, March 2019, pp. 17–31, 2019. doi: 10.1016/j.jweia.2018.12.016.

[21] A. Bianchini, F. Balduzzi, D. Di Rosa, and G. Ferrara, “On the use of Gurney flaps for the aerodynamic performance augmentation of Darrieus wind turbines,” *Energy Convers. Manag.*, vol. 184, 15 March 2019, pp. 402–415, 2019. doi: 10.1016/j.enconman.2019.01.068.

[22] W. Hao, M. Bashir, C. Li, and C. Sun, “Flow control for high-solidity vertical axis wind turbine based on adaptive flap,” *Energy Convers. Manag.*, vol. 249, 1 December 2021, p.

114845, 2021. doi: 10.1016/j.enconman.2021.114845.

[23] Q. Liu, W. Miao, Q. Ye, and C. Li, "Performance assessment of an innovative Gurney flap for straight-bladed vertical axis wind turbine," *Renew. Energy*, vol. 185, February 2022, pp. 1124–1138, 2022. doi: 10.1016/j.renene.2021.12.098.

[24] T. P. Syawitri, Y. Yao, J. Yao, and B. Chandra, "Geometry optimisation of vertical axis wind turbine with Gurney flap for performance enhancement at low, medium and high ranges of tip speed ratios," *Sustain. Energy Technol. Assessments*, vol. 49, no. February 2022, p. 101779, 2022. doi: 10.1016/j.seta.2021.101779.

[25] S. Jain, N. Sitaram, and S. Krishnaswamy, "Computational investigations on the effects of Gurney flap on airfoil aerodynamics," *International Scholarly Research Notices,* vol. 2015, Article ID 402358, 11 pages, 2015. https://doi.org/10.1155/2015/402358.

[26] Y. Li, J. Wang, and P. Zhang, "Influences of mounting angles and locations on the effects of Gurney flaps," Journal of Aircraft, vol 40(3), pp. 494-498, 2003. https://doi.org/10.2514/2.3144.

[27] D. Luo, D. Huang, and X. Sun, "Passive flow control of a stalled airfoil using a microcylinder," *J. Wind Eng. Ind. Aerodyn.*, vol. 170, November 2017, pp. 256-273, 2017. https://doi.org/10.1016/j.jweia.2017.08.020.

[28] J. Zhong, J. Li, P. Guo, and Y. Wang, "Dynamic stall control on a vertical axis wind turbine aerofoil using leading-edge rod," *Energy*, vol. 174, 1 May 2019, pp. 246-260, 2019. doi: 10.1016/j.energy.2019.02.176.

[29] A. Choudhry, M. Arjomandi, and R. Kelso, "Methods to control dynamic stall for wind turbine applications," *Renew. Energy*, vol. 86, February 2016, pp. 26-37, 2016. doi: 10.1016/j.renene.2015.07.097.

[30] 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, 15 January 2018, pp. 1107-1124, 2018. doi: 10.1016/j.energy.2017.10.094.

[31] W. Mostafa, A. Abdelsamie, M. Sedrak, D. Thévenin, and M. H. Mohamed, "Quantitative impact of a micro-cylinder as a passive flow control on a horizontal axis wind turbine performance," *Energy*, vol. 244, 1 April 2022, p. 122654, 2022. doi: 10.1016/j.energy.2021.122654.

[32] Y. Qi, S. Xu, and D. Huang, "Investigation on aerodynamic performance of horizontal axis wind turbine by setting micro-plate in front of the blade leading edge," *Renew. Energy*, vol. 179, December 2021, pp. 2309–2321, 2021. doi: 10.1016/j.renene.2021.08.035.

[33] M. R. Castelli, A. Englaro, and E. Benini, "The Darrieus wind turbine: Proposal for a new performance prediction model based on CFD," *Energy*, vol. 36, no. 8, August 2011, pp. 4919–4934, 2011. doi: 10.1016/j.energy.2011.05.036.

[34] A. Rezaeiha, H. Montazeri, and B. Blocken, "Towards accurate CFD simulations of

vertical axis wind turbines at different tip speed ratios and solidities: Guidelines for azimuthal increment, domain size and convergence," *Energy Convers. Manag.*, vol. 156, 15 January 2018, pp. 301–316, 2018. doi: 10.1016/j.enconman.2017.11.026.

[35] M. Asadbeigi, F. Ghafoorian, M. Mehrpooy, S. Chegini, and A. Jarrahian, "A 3D study of the Darrieus wind turbine with auxiliary blades and economic analysis based on an optimal design from a parametric investigation," Sustainability 2023, 15, 4684. 2023. https://doi.org/10.3390/su15054684.

[36] R. Ansaf, H.S. Abdelhameed, I. Hashem, and Z. Harun, "Efficiency-based design optimization of the H-type Darrieus wind turbine with fixed guiding-walls," Energy Reports, 9 (2023) 3576–3592, 2023. https://doi.org/10.1016/j.egyr.2023.01.117.

[37] S. A. Bakhumbsh, and M. H. Mohamed, "Effect of micro-cylinder as a passive controller on the Darrieus wind turbine performance," Ocean Engineering, Volume 266, Part 5, 15 December 2022, 113118, 2022. https://doi.org/10.1016/j.oceaneng.2022.113118.

[38] T. P. Syawitri, Y. Yao, J. Yao, and B. Chandra, "A review on the use of passive flow control devices as performance enhancement of lift-type vertical axis wind turbines," WIREs Energy Environ. 2022;11:e435. https://doi.org/10.1002/wene.435.

[39] M. H. Mohamed, "Performance investigation of H-rotor Darrieus turbine with new airfoil shapes," *Energy*, vol. 47, no. 1, November 2012, pp. 522–530, 2012, doi: 10.1016/j.energy.2012.08.044.

[40] A. Rezaeiha, I. Kalkman, and B. Blocken, "CFD simulation of a vertical axis wind turbine operating at a moderate tip speed ratio: Guidelines for minimum domain size and azimuthal increment," *Renew. Energy*, vol. 107, July 2017, pp. 373–385, 2017. doi: 10.1016/j.renene.2017.02.006.

[41] V. Yakhot, and S. A. Orszag, "Renormalization group analysis of turbulence. I. Basic theory," *Sci. Comput.*, vol. 1(1), pp. 3-51, 1986. https://doi.org/10.1007/BF01061452.

[42] T-H. Shih, W. W. Liou, A. Shabbir, Z. Yang, and J. Zhu, "A new k-ϵ eddy viscosity model for high Reynolds number turbulent flows," *Comput. Fluids*, vol. 24, no. 3, March 1995, pp. 227–238, 1995. https://doi.org/10.1016/0045-7930(94)00032-T.

[43] F. R. Menter, "Two-equation eddy-viscosity turbulence models for engineering applications," *AIAA J.*, vol. 32 (8), pp. 1598–1605, 1994. https://doi.org/10.2514/3.12149.

[44] K. H. Wong, W. T. Chong, S. C. Poh, Y. Shiah, and N. Liana, "3D CFD simulation and parametric study of a flat plate deflector for vertical axis wind turbine," *Renew. Energy*, vol. 129, Part A, December 2018, pp. 32-55, 2018. doi: 10.1016/j.renene.2018.05.085.

[45] M.H. Ranjbar, B. Rafiei, S.A. Nasrazadani, K. Gharali, M. Soltani, A. Al-Haq, and J. Nathwani, "Power enhancement of a vertical axis wind turbine equipped with an improved duct," Energies 14 (18), 5780, 2022. https://doi.org/10.3390/en14185780.

[46] W. Mostafa, A. Abdelsamie, M. Mohamed, D. Thévenin and M. Sedrak "Aerodynamic performance improvement using a micro-cylinder as a passive flow control around the S809 airfoil," IOP Conference Series: Materials Science and Engineering. Vol. 973. No. 1. IOP Publishing, 2020. DOI 10.1088/1757-899X/973/1/012040

[47] S. M. Dastjerdi, K. Gharali, A. Al-Haq, and Jatin Nathwani, “Application of simultaneous symmetric and cambered airfoils in novel vertical axis wind turbines,” Applied Sciences 11 (17), 8011, 2021. https://doi.org/10.3390/app11178011

[48] E. Bakhtiari, K. Gharali, and S. F. Chini “Corrigendum to super-hydrophobicity effects on performance of a dynamic wind turbine blade element under yaw loads” Renewable Energy, Volume 147, Part 1, Page 2528, 2020. https://doi.org/10.1016/j.renene.2019.08.076