

CFD and experimental investigations of a novel vertical axis Omni-flow wind turbine shroud system operating at low Reynolds numbers, typical urban flow conditions

Micky Mike Ngouani Siewe^{1*}, Yong Kang Chen^{2*}, Rodney Day³, Opukuro David-West⁴

^{1,2,4}*School of Physics, Engineering and Computer Science, University of Hertfordshire, Hatfield, Herts AL10 9AB, UK*

³*Pro Vice-Chancellor (Research & Innovation), London South Bank University, Borough Rd, London SE1 0AA, UK*

*Corresponding authors: m.ngouani-siewe2@herts.ac.uk; y.k.chen@herts.ac.uk

Abstract

This study examined the aerodynamic performance and flow characteristics of an innovative shrouded omnidirectional wind turbine system at low Reynold numbers to tackle the SDG 7 of the UN. Four turbulence models, notably the Shear Stress Transport (SST Menter) $k-\omega$, the Reynolds Stress Transport (RST), the Improved Delay Detached Eddies Simulation (IDDES) SST $k-\omega$, and the Large Eddy Simulation (LES WALE), were tested to find the best model(s) for this new omni-flow wind turbine. Compared with experimental results, the LES WALE and the IDDES SST $k-\omega$ models best predicted turbulent and uniform flow regions inside and outside the shroud, including CFD parameters, with one being more accurate on specific parameters such as C_d and another on other parameters. The IDDES SST $k-\omega$ excelled in predicting velocity ratios, lift coefficient (C_l) and provided the best balance of accuracy and computational efficiency of this innovative omni-direction wind turbine shroud.

The novel shroud design, with its unique ability to self-start and harness omnidirectional wind at any free stream velocity for electricity generation, was a key focus of this study. The airflow inside and around the innovative shroud system was meticulously characterised for potential optimisation. Findings demonstrated that the shroud effectively redirected airflow as low as 1.5 m/s into the turbine housing and with 1.5 maximum speed ratio the airflow flew towards the outlet and generated power, leading to superior self-starting capabilities. The power output increased non-linearly with the rotational speed, achieving optimal efficiency beyond 1000 rpm. It is evident that this new system has the potential for enhanced energy capture under varying wind conditions in urban areas.

Keywords

Omni-Flow shrouded vertical axis wind turbine, Low noise self-starting system, CFD turbulence models, Energy harvesting in urban areas, UN sustainable development goals.

Nomenclature			
D	Drag [N]	λ	tip speed ratio [–]
L	Lift [N]	ν	kinematic viscosity [m ² /s]
ω	Rotational speed [RPM]	ρ	air density [kg/m ³]
e°	Flow Entrance angle [°]	τ_{ij}	tensor of stress [Pa]
N	Blades Number [–]	Ω_{ij}	Vorticity [1/s]
P	Power [W]	θ	Wind direction angle [°]
C_{power}	Coefficient of power [–]	Abbreviations	
R	Rotor Blade radius [m]	3D	Three-dimensional
c	chord length [m]	CFD	Computational Fluid Dynamics
C_p	Pressure coefficient [–]	DDES	Delayed Detached Eddy Simulation
D_r	Rotor diameter [m]	DES	Detached Eddy Simulation
H	Shroud case height [m]	HAWT	Horizontal Axis Wind Turbine
k	Turbulent kinetic energy [m ² /s ²]	VAWT	Vertical Axis Wind Turbine
L_{IDDES}	IDDES length scale [m]	IDDES	Improved Delayed Detached Eddy Simulation
L_{LES}	LES length scale [m]	LES	Large Eddy Simulation
L_{RANS}	RANS length scale [m]	N-S	Navier-Stokes
S_{ij}	mean strain rate [1/s]	RANS	Reynolds-averaged Navier-Stokes
t	Time [s]	SST	Shear Stress Transport
u_j	Velocity components [m/s]	TSR	Tip Speed Ratio
U_∞	Free stream wind speed [m/s]	TKE	Turbulent Kinetic Energy
x_j	Coordinate variables [m]	WALE	wall-adapting local eddy-viscosity
y^+	Dimensionless wall distance [–]	Cyl	Cylinder
Greek symbols		UN SDG	United Nations Sustainable Development Goals
α	Wing base angle of attack [°]	avg	Average
β	Rotor blade pitch [°]	max	Maximum

1 Introduction

Global warming and the concern for sustainable energy resources for a better world have led many countries and organisations to launch ambitious wind power development programs to exploit this renewable energy [1]. The Sustainable Development Goal 7 (SDG 7) of the United Nations (UN) is to provide affordable and clean energy for all by 2030, which highlights the slow pace of renewable energy development and the low access to clean energy technology in least-developed countries [2]. Most of the programs focus on onshore and offshore horizontal-axis wind power projects [3]. However, large-scale horizontal-axis wind turbine farms with rotor diameters greater than 5 m [4], produce great noise and have been shown to contribute to the rise of global temperature; thus, global warming [5, 6]. Small-scale vertical-axis wind turbines have been the focus since they can be employed precisely in urban areas and on buildings without the issues of large-scale ones [7, 8]. Similarly, the implementation of wind turbines in built environments presents specific challenges mainly due to the highly non-uniform, fluctuating, and turbulent wind [9, 10].

Small vertical-axis wind turbines have become increasingly popular for their excellent adaptability [11] with their low manufacturing/installation costs, low noise pollution, no yaw mechanism due to their omnidirectional nature and performance in turbulent conditions [12]. However, small wind turbines currently in use may exhibit reduced efficiency under complex environmental conditions, with transient loads and vibrations potentially compromising their durability [9, 13]. Very little research has involved the unpredictable wind conditions in urban areas, making it challenging to generate a steady and reliable energy source [14]. In particular, flow over shrouded wind turbines at low Reynolds number or low wind speed [15] is characterised by complex phenomena such as flow separation, transition to turbulence and small and large turbulent eddies, thus making it more difficult or less viable to harness energy [16, 17]. An accurate prediction of these phenomena is crucial for this novel shroud wind turbine to harness wind energy efficiently in such an environment. Similarly, very little research has been reported on the characterisation of the wind flow behaviour at such speed and in very turbulent environments in and around precisely vertical wind turbine shrouds [18]. Most available studies have not implemented the aesthetic and practicality of their vertical axis wind turbine designs in built environments, which requires good starting capabilities. [19] to help to tackle the vast consumption of grid energy by buildings [20].

The SAVONIUS and DARRIEUS wind turbines represent two primary concepts in vertical axis wind turbine (VAWT) design, each addressing different operational challenges. The SAVONIUS rotor, functioning on differential drag between its buckets, is a promising solution for low wind speed conditions but suffers from inherently low efficiency, limiting its broader applicability. In

contrast, DARRIEUS turbines, which operate on lift-based principles, offer improved power performance over SAVONIUS rotors but have poor self-starting capability and continue to exhibit suboptimal overall efficiencies due to their complex and often unpredictable aerodynamic behaviour [21, 22, 23]. Shrouded turbines bypass the theoretical limit for power production by concentrating air onto the turbines, thus artificially raising the efficiency [24]. A compact brimmed diffuser (wind-lens structure) was developed, resulting in a significant increase in output power by approximately 2 to 5 times compared to conventional bare wind turbines due to the concentration of wind energy. This enhancement is attributed to a low-pressure region formed by a strong vortex behind the broad brim, which draws increased mass flow to the wind turbine inside the diffuser [25]. Shrouded turbines were reported to have substantial potential for retrofitting existing buildings and clear benefits in situations where visual impact and safety were critical concerns [26]. A Rooftop flanged diffuser shroud design was presented. The mechanism effectively redirected and accelerated rooftop airflow toward the wind turbine within the diffuser casing, enhancing wind power generation over buildings in complex urban settings. However, it just captured the wind in one direction [18].

A new concept power augmentation wind shrouded system such as the INVELOX was investigated due to their excellent performance [27]. Many optimised concepts of this system have been investigated and showed a considerable power increase: the net mass flow rate of air entering the INVELOX increased by 64.7%, and wind power at the Venturi's throat rose by 279.9%. The wind speed ratio improved from 1.6 to 2.4 when comparing the primary and optimal INVELOX designs [28]. A similar concept with a morphing omnidirectional panel mechanism was looked at, and it improved the flow speed at the Venturi section by at least 6% when the outside wind speed was low [29]. However, these concepts are for large-scale applications and require many complex parts and materials, which are not ideal for urban environments where size and implementation are crucial. Furthermore, a sole vertical shroud at a low Reynolds number had been investigated, and its effective omnidirectional property and the potential to drive a turbine have been reported [30].

Most shrouded wind turbines are horizontal axis turbines (HAWTs) [31] and most of the current designs face some more limitations. Typically, they have all struggled to deal with the versatile nature of wind in urban areas which is constantly changing speeds and directions. Although current shrouded systems increase the performance of the turbine, they are usually unidirectional. Horizontal shrouded design still produces high amount of disturbed air in their wakes, thus limiting their ability to be placed in a closer cluster arrangement which is an aspect of urban areas where buildings are closely arranged, hence reducing their scalability in this environment. HAWTs demand constant maintenance due to the amount of moving parts, thus

reducing their life span. Regarding noise pollution and aesthetic of HAWTs, they are a huge concern in cities. Shrouded HAWTs still project most of their wakes in the horizontal direction and their noise is still relatively high. Finally, they are also source of potential harm to the wildlife such as birds who can be caught in the super high-speed tip of a blade.

Thus, to tackle SDG 7 of the UN and addresses the limitations of existing shrouded configurations, this study presents a groundbreaking vertical omni-flow shrouded wind turbine system which uses a circular wing base structure to capture the wind in low-speed conditions, such as in urban areas and the characterisation of the flow inside the shroud to enhance the wind turbine efficiency. The performance of the shroud has been analysed using both CFD and experimental approaches at low Reynolds number flows ($Re \approx 10^5$), and the accuracy of various turbulence models is evaluated. Four turbulence models were employed to identify the best model for this novel system, which were the Shear Stress Transport (SST Menter) $k-\omega$, the Reynolds Stress Transport model (RST), the Improved Delay Detached Eddies Simulation model (IDDES) SST $k-\omega$, and Large Eddy Simulation (LES). A thorough comparison and discussion of the CFD turbulent models and experimental results, including power performance, were carried out, and the best suitable model for this vertical omni-flow shrouded wind turbine system has been identified.

2 Methodology and numerical analysis

2.1 Omni-flow wind turbine shroud systems

The urban wind nature of low wind speed, changeable speed directions and turbulence was considered in designing the Omni-flow wind turbine. As shown in Figure 1 (a - b), the Omni-flow wind turbine consists of a fixed shroud and a rotor. The shroud was 3-D printed using ABS plastic, the surface of which was polished, as the near-wall phenomenon was significant in the shroud structure. It has a circular inverted wing base ring and a cylindrical shroud case. The dimensions of this system are given in Table 1 and Figure 1 (c). One of the application fields of this turbine is in urban environments as shown in Figure 1 (e). It could be used to provide some of the clean energy to industrial buildings as well as residential houses in a cluster arrangement. Because of its portable features, it could be employed as streetlights (road, highways and park pathway) where internet and phone signal booster could be implemented. It could also power a smart IT pole with integrated systems such as security camera, digital screen for advertisement, city information and map, as well as point of charging city EV bike and smart phone or tablet.

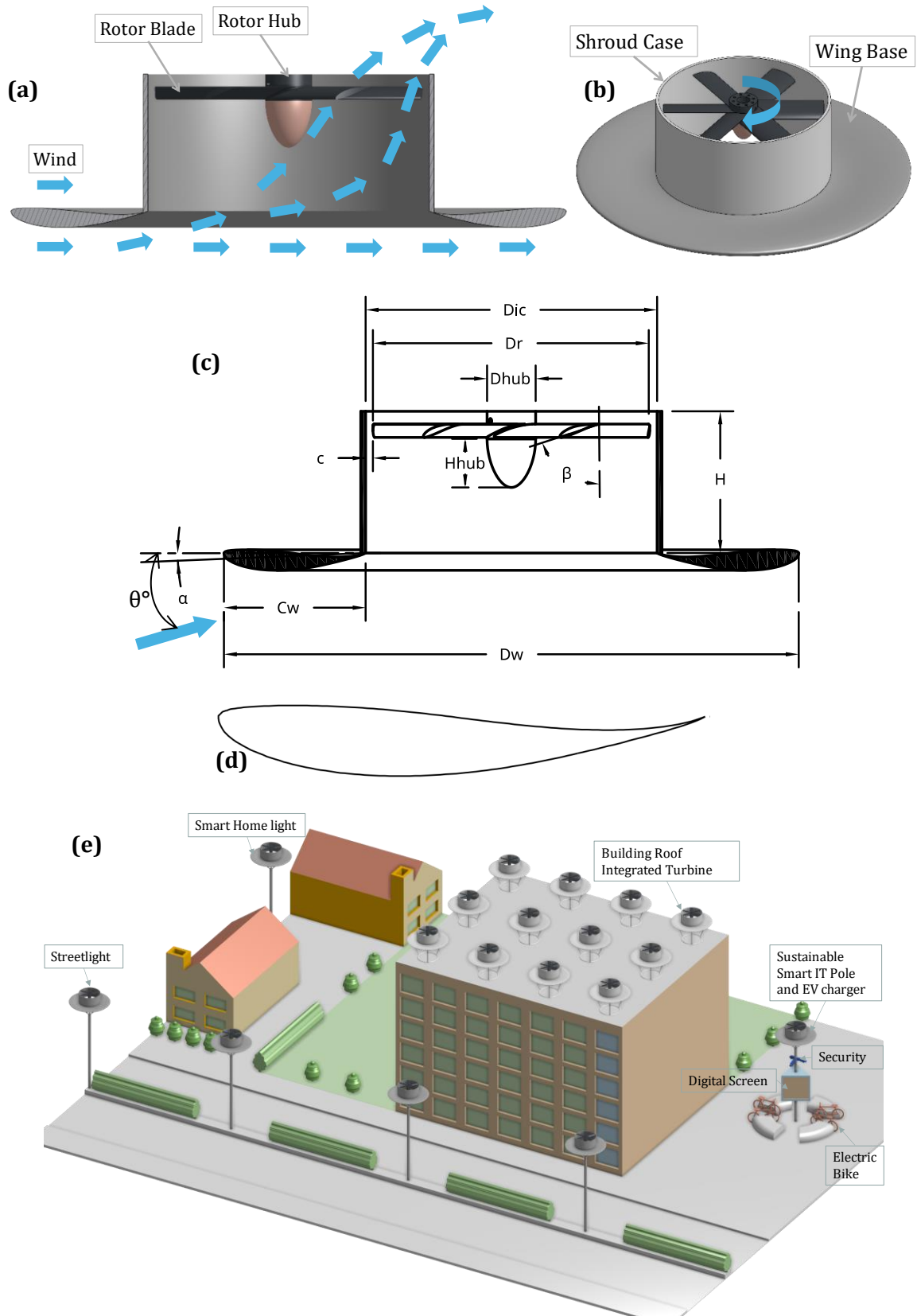


Figure 1: (a) Schematic of the wind turbine shroud system; (b) Isometric view of the shrouded wind turbine system; (c) Overall dimensions of a scaled-down wind turbine shroud system; (d) Profile of the inverted Aerofoil FX 63-137 [32]; (e) Applications of the turbine system in urban areas.

This Omni-flow shroud system is made of a wing base, shroud case and rotor blade. The wing base used the low Reynold number Wortmann FX 63-137 aerofoil in the inverted configuration and this aerofoil was also employed for the rotor blades with a Maximum thickness of 13.7% at 30.9% chord, a Maximum camber of 5.8% at 56.5% chord as shown in Figure 1(d). This aerofoil was selected due to its ability to keep the flow well attached. Its smooth pressure coefficient distribution leading to low stall characteristics and excellent constant lift coefficient, reaching about 1.6, which is not found in many aerofoils, has low drag, a high lift-to-drag ratio (L/D), and superior aerodynamics at low Reynolds number and low speeds [32] [33] . All these attributes have made this aerofoil suitable for this study. The solidity of the rotor blade was calculated using the formula $\sigma = NC_r/D_r$ where N is the number of blades, C_r the rotor blade chord length and D_r the rotor blade diameter [34].

Table 1: Dimensions of a wind turbine shroud system.

Turbine shroud		Turbine Rotor blade	
Parameter	Value	Parameter	Value
Wing base Aerofoil	FX 63-137	Rotor blade aerofoil	FX 63-137
Wing base outer diameter D_w [m]	0.35 ($2D_r$)	Rotor diameter D_r [m]	0.175
Wing base chord C_w [m]	0.0875 ($0.5D_r$)	Number of blades N	6
Shroud case inner diameter D_{ic} [m]	0.18 ($1.03D_r$)	Rotor blade pitch β [°]	75
Shroud case height H [m]	0.0875 ($0.5D_r$)	hub diameter D_{hub} [m]	0.03($0.17D_r$)
Shroud case wall thickness [m]	0.003	hub Heigh H_{hub} [m]	0.03($0.17D_r$)
Shroud case capacity C [m ³]	0.0023	Rotor blade chord C_r [m]	0.03 ($0.17D_r$)
Wing base angle of attack α [°]	0	Solidity ratio	1.03
Clearance c [m]	0.0025 ($0.014D_r$)		

Table 1 presents key information about the turbine shroud and the turbine rotor blade design specification. The turbine system design was chosen to represent a scaled down turbine of rotor diameter 1.05 m due to the size restriction of the wind tunnel. Furthermore, small scale vertical axis wind turbines [35], such as this system, which are designed for residential or urban environment applications, have typically a rotor diameter of about 2 m. Hence the scale factor was chosen to be 1/6 of the benchmark typical turbine. A high rotor blade solidity was chosen due to the low-speed condition of this study and this blade property was found to have better starting performance. The clearance between the rotor blade and the shroud case inner diameter was chosen to be 1.4% D_r as the smaller the clearance number, the less effect it has on the efficiency and performance of the turbine. The wing base dimensions were also chosen after extensive initial tests and iterations in the wind tunnel and in the CFD simulations. Furthermore,

the attention was also brought on the aesthetics, scalability and practicality of the turbine, specially that it would be used in urban settings where the public appeal is of significance.

2.2 Numerical simulation: turbulence models

Numerical approaches and experimental tests are commonly used to determine the performance of innovative VAWTs. The following four turbulence models were employed and evaluated:

2.2.1 SST $k-\omega$ model turbulent model

Menter has proposed the SST $k-\omega$ turbulence model as a hybrid approach integrating the $k-\omega$ and $k-\varepsilon$ models. Specifically, the $k-\omega$ model is applied in the near-wall region, while the $k-\varepsilon$ model is utilised in areas further from the wall. Although the $k-\omega$ model requires fine meshing for accurate near-wall flow analysis, making it more computationally expensive than the $k-\varepsilon$ model, it yields more reliable results. [36]. In addition, this turbulent shear stress transportation is essential to predict severe adverse pressure gradient flows. Consequently, researchers widely employ the SST $k-\omega$ turbulence model in numerous CFD simulations to investigate flow characteristics around VAWTs such as the Darrieus wind turbine and its variants [21]. Though, the model may overpredict turbulence levels in separated flows and can struggle with unsteady or highly anisotropic turbulence. Therefore, the SST $k-\omega$ model has been selected for this turbulence modelling investigation. [37]. Its governing equations with low-Re correction can be found in [36].

2.2.2 The Reynolds stress model RST

The Reynolds Stress Transport (RST) model, also known as the Reynolds Stress Model (RSM), provides higher-fidelity turbulence modelling by solving transport equations for Reynolds stresses, enabling better capture of turbulence anisotropy and complex flows compared to simpler models like $k-\varepsilon$ and $k-\omega$. Despite its increased accuracy in predicting secondary motions, separation, and reattachment, the RST model requires higher computational resources, finer grid resolution, and higher quality meshes due to its complexity and additional transport equations and may suffer from stability issues in complex geometries [38].

2.2.3 Detached eddy simulation: DES (IDDES SST $k-\omega$ turbulence model)

The DES-SST method is a unified LES/RANS hybrid that separates the near-wall region, solved with RANS equations, from the rest, solved with LES equations, based on the grid's properties, with DES originally proposed using the Spalart–Allmaras model but extendable to other turbulence models like SST. It has gained popularity in the research environment due to its well

balance accuracy to computational power merits [39]. The IDDES (Improved Delay Detached Eddy Simulation) model combines DES and wall-modelled LES. LES is employed for the external flow and separation regions. This method overcomes the excessive computational demands of LES and has better accuracy than the RANS and providing accurate unsteady flow predictions for separated flows and wakes though dependence on grid resolution in the LES region and challenges in modelling transition zones can affect prediction reliability in complex flows [39]. The hybrid RANS-LES turbulence model was employed in vertical axis wind turbine applications of the Darrieus wind turbines [40]. The DES modification in the SST model is applied to the dissipation term in the turbulent kinetic energy (k) equation. In the SST RANS model, it reads as shown in Equation (1) [41]:

$$\frac{\partial(\rho k)}{\partial t} + \frac{\partial(\rho u_j k)}{\partial x_j} = \frac{\partial}{\partial x_j} \left[\left(\mu + \frac{1}{\sigma_k} \mu_t \right) \frac{\partial k}{\partial x_j} \right] + \tau_{ij} S_{ij} - \frac{\rho k^{1.5}}{L_{IDDES}} \quad (1)$$

where ρ , k , t , u_j , μ , μ_t , τ_{ij} , S_{ij} are the density, turbulent kinetic energy, time, velocity, molecular viscosity, turbulent viscosity, tensor of stress and mean strain rate, respectively.

The IDDES length scale L_{IDDES} is expressed in Equation (2) [42]:

$$L_{IDDES} = \bar{f}_d(1 + f_e)L_{RANS} + (1 - \bar{f}_d)L_{LES} \quad (2)$$

where the length scales of RANS and LES are defined in Equation (3) [43]:

$$L_{RANS} = \frac{k^{0.5}}{\beta^* \omega} L_{LES} = C_{DES} \Delta \quad (3)$$

2.2.4 Large eddy simulation (LES WALE)

The study utilises the Large-Eddy Simulation (LES) with the Wall-Adapting Local Eddy-viscosity (WALE) model, which excels in unsteady problems and performs better near walls than other sub-grid scale (SGS) models like the Smagorinsky model, without needing additional damping functions [44]. The LES method resolves large turbulent scales while filtering smaller scales based on mesh resolution, requiring significant computational resources, and the WALE model is particularly effective for simulating transitional flow by providing zero eddy viscosity in laminar flow conditions. It is well-suited for capturing transient and high-fidelity flow structures. However, extremely high computational cost and sensitivity to grid quality make LES with WALE impractical for large-scale industrial flows. The filtered continuity and momentum equations are given in in Equations (4-6) [45].

$$\frac{\partial \bar{u}_i}{\partial x_i} = 0 \quad (4)$$

$$\frac{\partial \rho \bar{u}_i}{\partial t} + \frac{\partial \rho (\bar{u}_i \bar{u}_j)}{\partial x_j} = -\frac{1}{\rho} \frac{\partial \bar{p}}{\partial x_i} + \frac{\partial}{\partial x_j} \left[\nu \left(\frac{\partial \bar{u}_i}{\partial x_j} + \frac{\partial \bar{u}_j}{\partial x_i} \right) - \tau_{ij} \right] + f_i \quad (5)$$

where $\bar{u}_i$ is the filtered velocity, $\bar{p}$ is the filtered pressure, f_i (N/m³) represents the Eulerian dynamic force vector, ν (m²/s) is the kinematic viscosity and τ_{ij} is the sub-grid scale (SGS) stress tensor:

$$\tau_{ij} = \nu_t \left(\frac{\partial \bar{u}_i}{\partial x_j} + \frac{\partial \bar{u}_j}{\partial x_i} \right) \quad (6)$$

The WALE model provides an expression for the sub-grid scale eddy viscosity ν_t which is used to model the SGS stresses is given in equation (7), τ_{ij} is the sub-grid scale (SGS) stress tensor [46]:

$$\nu_t = (C_\omega \Delta)^2 \frac{(S_{ij}^d S_{ij}^d)^{1.5}}{\left(\overline{S_{ij}^d S_{ij}^d} \right)^{2.5} + (S_{ij}^d S_{ij}^d)^{2.5}} \quad (7)$$

The WALE constant C_ω is approximately 0.325. The detailed information can be found in [46].

2.3 Experimental setup

A multi-tube manometer connected to pressure taps on the shroud was used to measure local pressures, as shown in Figure 2 (a). The experiment was conducted in a closed return wind tunnel with adjustable wind speeds up to 25 m/s at the University of Hertfordshire, as shown in Figure 2 (b). The tunnel has a test section of 1145 mm x 840 mm (width, height) with a flow uniformity of 98% and turbulence intensity of less than 1%. The model blockage was 2.08%. As the turbine was tested in a closed return wind tunnel, the blockage effect, as suggested by Barlow et al [47], was considered and applied on U , C_d and C_l . A pitot tube and a high-speed camera were employed to measure the actual free stream speed and the rotor's RPM after calibration. Finally, an undersized AC motor with a resistance box and an oscilloscope were also used to measure the power produced by the turbine, as shown in Figure 2 (c, d, e, f, g, h).

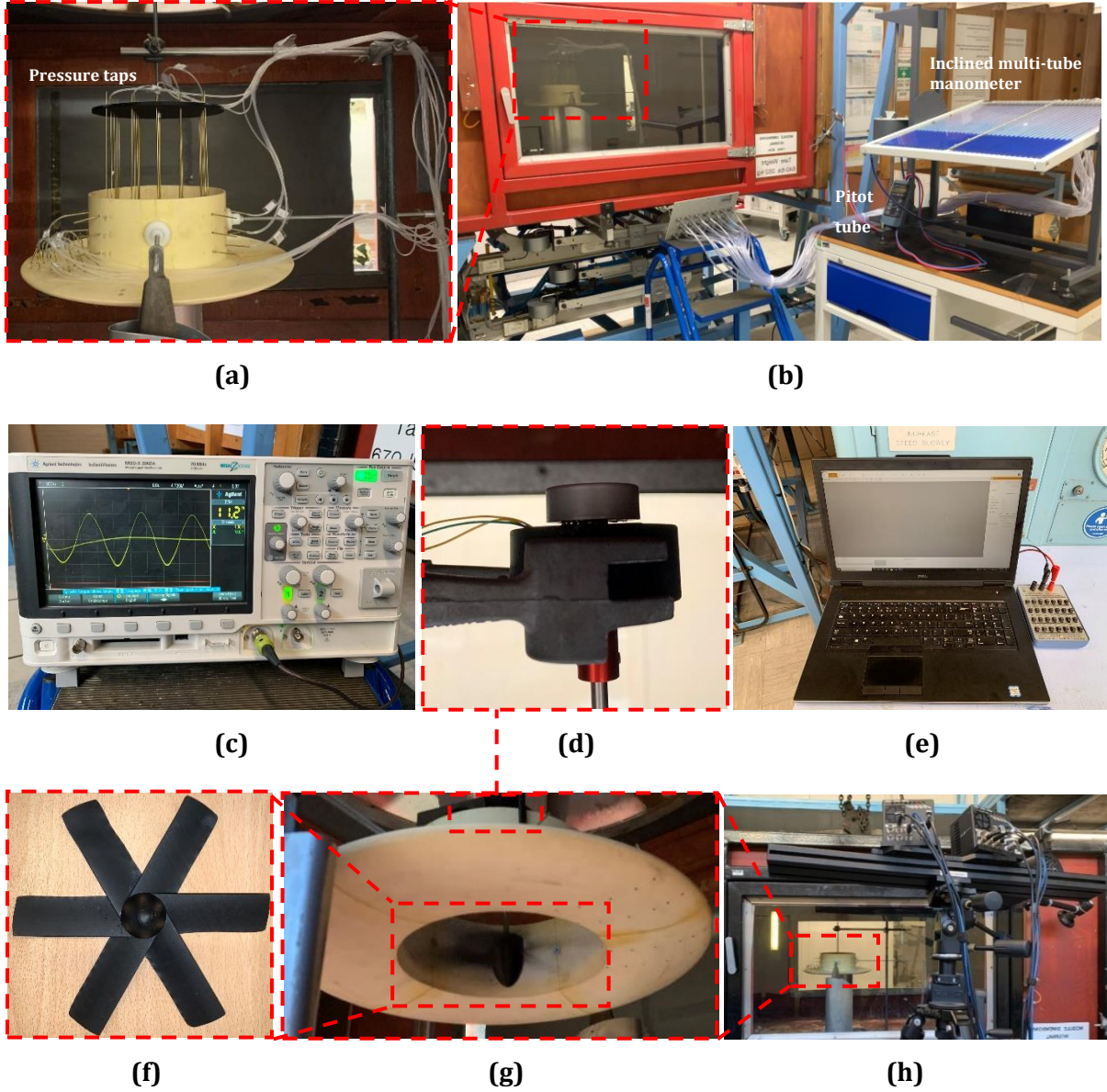


Figure 2. Wind tunnel testing apparatus: (a) fitted pressure taps; (b) wind tunnel test section; (c) Oscilloscope; (d) Small AC generation; (e) Data acquisition and resistance box; (f) Rotor turbine blade; (g) Rotating turbine in the Shroud; (h) High-Speed Camera and light projector.

2.4 Computational domain and mesh setting

The aerodynamic characteristics of this innovative wind turbine were investigated with the CFD package of Star-CCM+, which is a finite volume method (FVM) CFD solver based on Navier Stokes equations which solves the integral form of conservation equations over control volumes thus, conserves mass, momentum, and energy and works well on unstructured grids. The computational domain had dimensions $6D_w \times 20H \times 16D_w$, as shown in Figure 3 (a) and the blockage ratio achieved was 1% [36]. The shroud case internal region was divided into five planes from the inlet plane (Inlet Cyl) to the outlet plane (Outlet Cyl) through $\frac{1}{4}$ Cyl, $\frac{1}{2}$ Cyl and $\frac{3}{4}$ Cyl planes, as shown in Figure 3 (b) & (c).

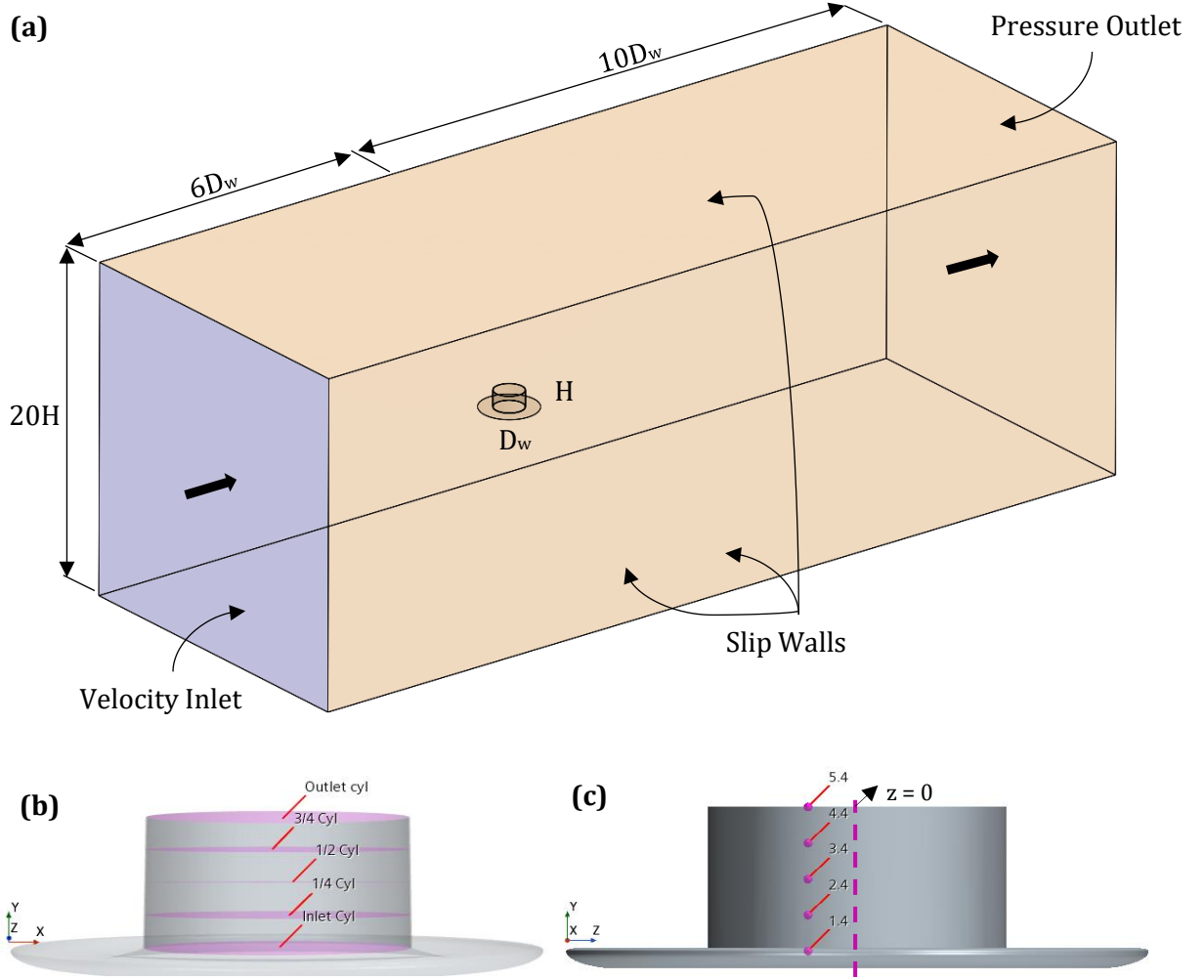


Figure 3: (a) Computational domain and boundary conditions; (b) Shroud case planes, Inlet Cyl ($y=0$), $\frac{1}{4}$ Cyl H , $\frac{1}{2}$ Cyl H , $\frac{3}{4}$ Cyl H , Outlet Cyl; (c) radial lines probe from which data was collected 1.4 ($y=0$), 2.4 ($y=\frac{1}{4}H$), 3.4 ($y=\frac{1}{2}H$), 4.4($y=\frac{3}{4}H$), 5.4 ($y=H$).

The mesh approach was a region-based meshing with the polyhedral and prism layer meshes. Thus, a 3-dimensional unstructured polyhedral cell mesh was generated, with refined regions around the shroud to capture boundary layers and flow separation accurately, as shown in Figure 4 (a). A structured polyhedral cell refinement cylindrical volume was set in the shroud case, as shown in Figure 4 (b) better accuracy. IDDES and LES models require a high-quality mesh, and careful consideration was made when choosing low wall treatment to achieve a y^+ ($=\rho u_\tau y / \mu$) < 1 as near wall phenomenon was crucial to these simulations [48] [49]. This would guarantee that the boundary layer could be fully solved. Where, u_τ is the friction velocity and y the distance from the first cell centre to the nearest wall [50]. To achieve this y^+ value, the number of prism layers was set to 20 inflation layers, the prism layer stretching was set to 1.3, its thickness was set to $2.2\%D_r$, and the minimum mesh element size was $0.07\%D_r$, as shown in Figure 4 (c, d, e, f, g, h). This created a mesh of ~ 13.4 million cells.

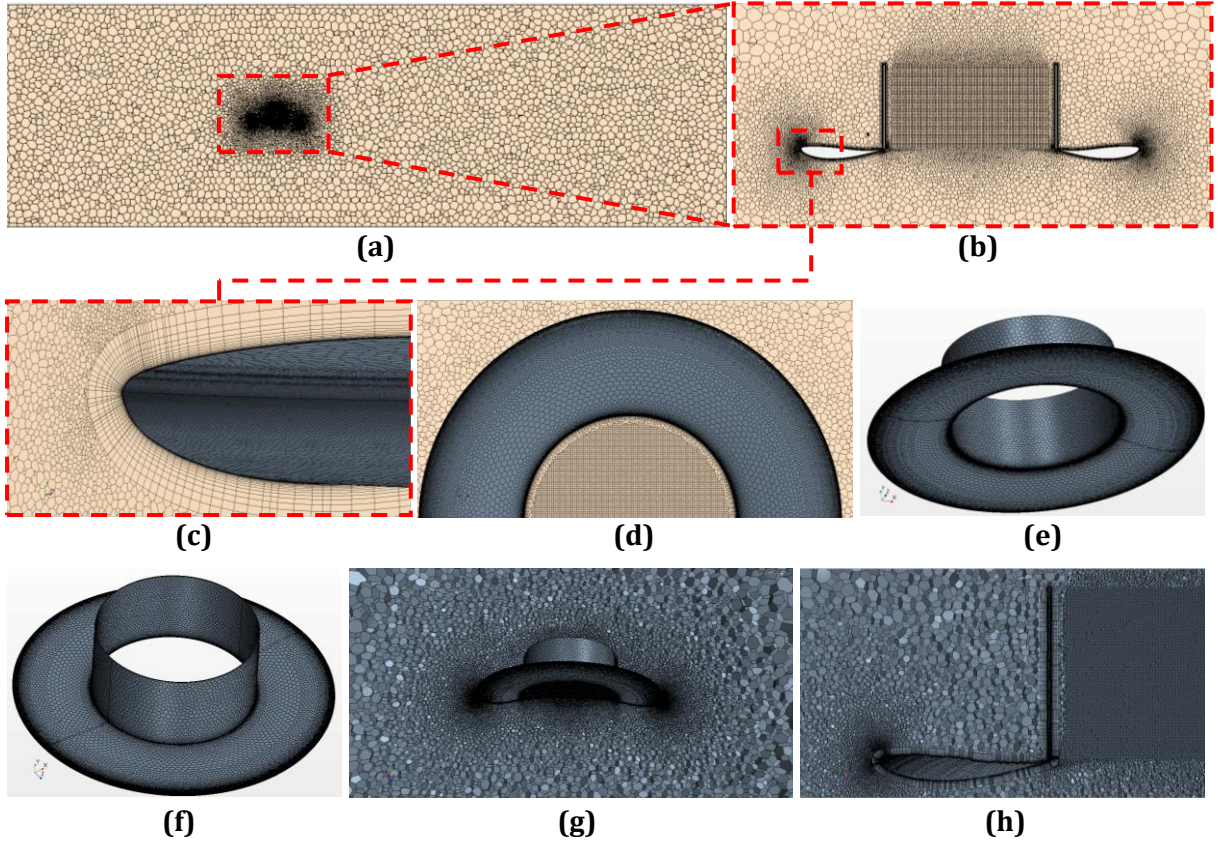


Figure 4. Mesh generation around the shroud: (a) mesh structure within the domain; (b) structured mesh within the shroud; (c) prism layers distribution near the shroud; (d) mesh in the bottom view of the shroud; (e) surface mesh on the shroud bottom view; (f) surface mesh on the shroud isometric view; (g) 3D polyhedral mesh elements around shroud; (h) 3D polyhedral mesh elements inside the shroud.

2.4.1 Solver setting

The inlet was set as a uniform velocity inlet profile of 6 m/s, the turbulent intensity was set to 1% to match the wind tunnel. The outlet was set as a pressure outlet with zero-gauge pressure with the reference operation pressure of 101325 Pa. The Reynolds number Re ($Re = 1.45 \times 10^5$) based on the wing base diameter is defined as $Re_{D_w} = U_\infty D_w / \nu$ where, U_∞ is the velocity of approaching flow and ν is the fluid's kinematic viscosity [51]. Walls were set in No-slip condition as symmetry planes. For RANS, the implicit unsteady segregated flow method using the SIMPLE algorithm for the pressure-velocity coupling was employed to solve the discretised continuity and momentum equations second-order discretization scheme for time. The PISO Unsteady utilising the PISO algorithm for the pressure-velocity coupling with also the second-order discretization scheme for time; for the IDDES AND LES. The convergence of the solutions was established when the residuals reached a threshold of 10^{-4} or lower. The simulation was conducted using nine CPU cores operating in parallel.

2.4.2 Grid and time step independency study

An optimal mesh size balances accuracy and computational efficiency [52]. Thus, as done by Rezaeiha et al, [53] the grid convergence analysis was performed to confirm the independence of the results on the grid resolution and several grid sizes were tested. Figures 5 (a) and (b) show the drag, lift and inlet mass flow rate from five different numbers of elements at an air-free stream velocity of 6m/s ($Re = 1.45 \times 10^5$). It could be observed that the lift and drag, including the mass flow rate, reached the lowest deviation when the cell number was 10634271. However, by increasing the cell number to 13392030, the predicted lift and mass flow rate changed by less than 0.1%. Thus, to ensure that the adequate maximum y^+ of 0.9 was reached, the grid mesh index of 13392030 elements was adopted as the standard for the subsequent simulations.

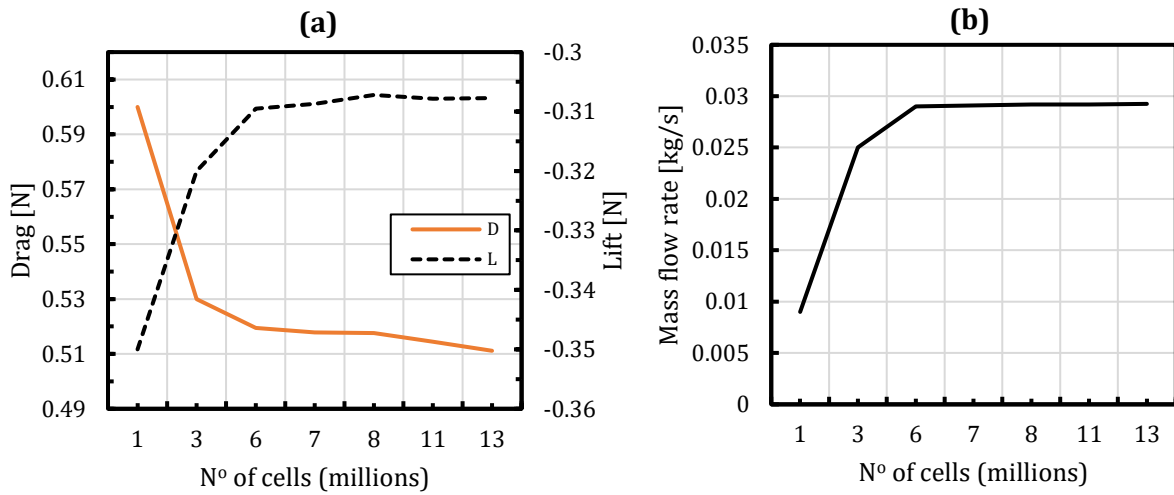


Figure 5. (a) Convergence of drag and lift with the mesh size, (b) Convergence of shroud internal mass flow rate.

In this simulation, the time step was set to 0.01 seconds [48]. The time step size was evaluated by adopting four time step sizes which were 0.015 s, 0.0125 s, 0.01 s, 0.0075 s. From the analysis, it was found that 0.01 s provided a time-independent solution. There was no significant difference between 0.01 and 0.0075 with discretization error of 0.12 %. Subsequently, 0.01 s was used in the presented numerical simulations.

The following aerodynamic parameters were investigated in this study due to their high importance in understanding the behaviour of the fluid around the shroud and the performance of the turbine to highlight future areas of optimisation. The first was the pressures coefficient which can help to understand the pressure distribution and its impact on flow characteristics and performance and is defined in Equation (8):

$$C_p = \frac{P_i - P_\infty}{\frac{1}{2} \rho U_\infty^2} \quad (8)$$

where $P_i - P_\infty$ is the gauge pressure, P_i is the measured surface pressure at the i -th pressure tap, P_∞ is the reference static pressure [54].

The second parameter was the skin friction coefficient which can be expressed in Equation (9)

$$C_f = \frac{\tau_w}{\frac{1}{2}\rho U_\infty^2} \quad (9)$$

where τ_w is the wall shear stress at the wing, ρ is the density and U_∞ is the free stream speed [55].

The third one was the derived ratio of wind turbine power to shroud case diameter $\frac{P_w}{D_{ic}^2}$ (the power ratio output) which is expressed by Equation (10):

$$\frac{P_w}{D_{ic}^2} = \frac{1}{8}\rho\pi C_{power} U_\infty^3 \quad (10)$$

where U_∞ is the air velocity; ρ is the air density, D_{ic} is the shroud case internal diameter and C_{power} is the coefficient of power [56].

The tip speed ratio (λ) was the fourth one which was a critical parameter in wind turbine design and performance analysis. Understanding and optimising λ is essential for maximising a wind turbine's efficiency and power output. λ can be expressed in Equation (11):

$$\lambda = \frac{\omega R}{U_\infty} \quad (11)$$

where ω is the angular velocity of the rotor, R is the rotor radius, and U_∞ is the wind-free stream speed [57]. A higher λ indicates that the blades are moving faster than the wind speed.

3 Results and discussion

3.1 Flow field analysis

3.1.1 Velocity contours and distribution

As shown in Figure 6, The SST $k-\omega$ model predicted a 16% flow acceleration from inlet to outlet, while the RST model showed a 13% acceleration with more fluctuating velocities, maintaining uniform flow over only 15% of the plane area after the inlet. Figure 6 (i) shows the SST $k-\omega$ model has predicted a maximum speed of 7 m/s within the shroud, with two distinct flow regions evolving towards the outlet plane (e). Initially, 70% of the inlet surface at Figure 6 (i)(a) experienced uniform acceleration, which dropped to 20% uniform flow on plane (b) $\frac{1}{4}H$. By plane (d), 15% of the area exhibited uniform accelerated flow. In Figure 6 (ii), the RST model predicted a maximum speed of 6.8 m/s, with 65% uniform acceleration at the inlet, reducing to 15% on the

plane (b), and reaching a maximum speed over 12% of the outlet plane's surface area, with a fully developed recirculation flow. The IDDES SST $k-\omega$ model shown in Figure 6 (iii) predicted a maximum speed of 7.2 m/s, with 75% of the inlet area experiencing uniform acceleration and has indicated a 20% acceleration and partial turbulence. Figure 6 (iv) shows that the LES WALE model predicted a 25% speed increase with a 12% uniform area, indicating dominance by velocity fluctuations within the shroud.

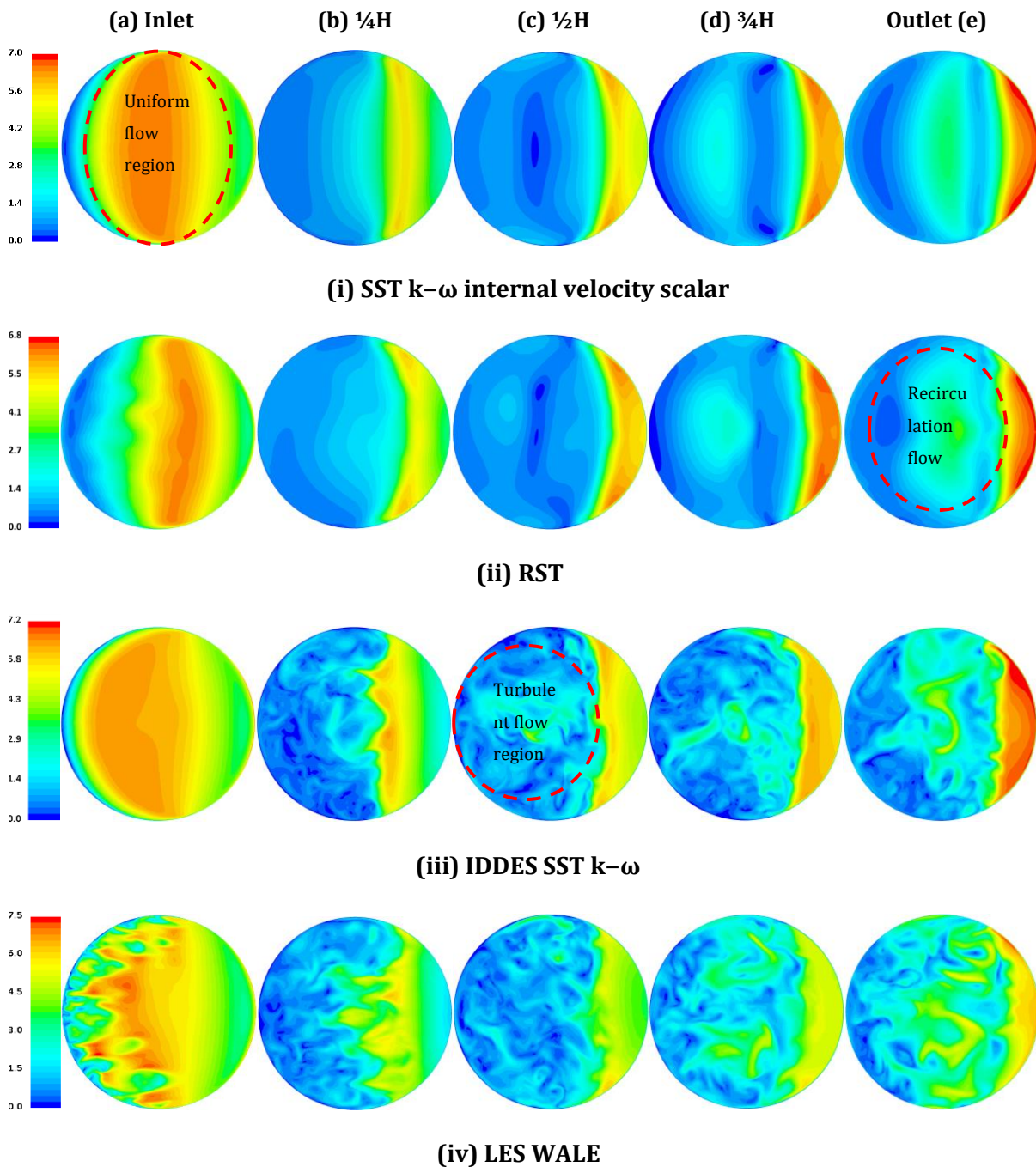


Figure 6: Internal velocity contour for different turbulent models (i), (ii), (iii), (iv) on planes: (a) inlet ; (b) $\frac{1}{4}H$; (c) $\frac{1}{2}H$; (d) $\frac{3}{4}H$; (e) outlet.

As shown in Figure 7, the shroud aims to divert external flow into the case and accelerate it towards the outlet, with maximum speed reached at the outlet. Most models show two distinct areas of turbulent small-scale eddies and uniform accelerated flow from the plane (c) to the outlet plane (e). The IDDES SST $k-\omega$ and LES WALE models accurately capture velocity fluctuations and recirculation. The RST model predicts more significant turbulence and low velocity at the inlet with sharper changes.

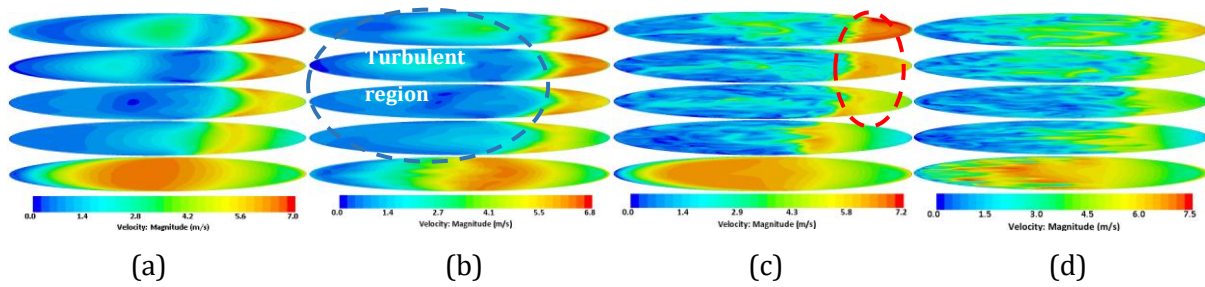


Figure 7. Velocity scalar contour across different planes within the shroud for all turbulent models (a) SST $k-\omega$; (b) RST model; (c) IDDES SST $k-\omega$; (d) LES WALE.

3.2 Validation of the modelling results

C_d represents the amount of drag force on an object relative to its cross-sectional area and the density of the fluid. C_l represents the lift force an aerodynamic object generates based on the fluid flow's dynamic pressure and the surface's reference area. The detailed information can be found in [56].

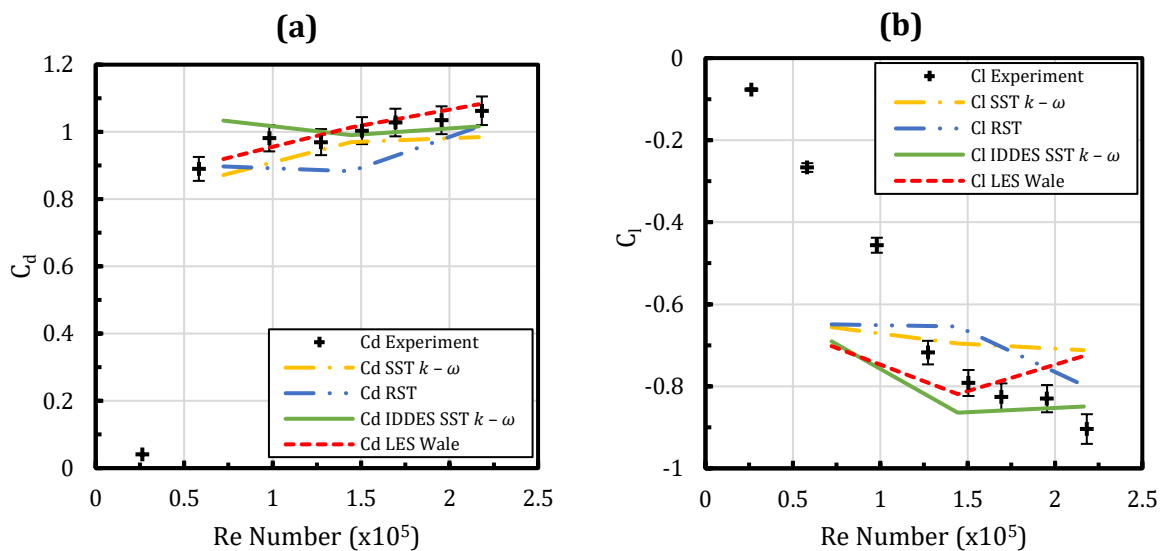


Figure 8. Experimental and numerical distribution of C_d (a) and C_l (b) concerning the wing base Re number ($\times 10^5$).

Figure 8 (a) the LES WALE model accurately predicts the drag coefficient (C_d) for Reynolds numbers over 1.5×10^5 , with a relative error of 0.932% at $Re = 1.45 \times 10^5$ (6 m/s). The IDDES SST $k-\omega$ model shows a similar accuracy with a 1.34% error. In contrast, the SST $k-\omega$ and RST models are less accurate, with 3.4% and 12.03% errors, respectively. Figure 8 (b) shows the coefficient of lift (C_l); all models predict a similar negative trend, with experimental data showing a sharp decrease in C_l up to $Re = 0.98 \times 10^5$, then a slower decline. The IDDES SST $k-\omega$ and LES WALE models closely match experimental results, with relative errors of 9.14% and 3.402%, respectively. Meanwhile, the SST $k-\omega$ and RST models overpredicted C_l , with errors of 12.1% and 17.4%. The drag and lift coefficients were found as 1.004 and -0.792 respectively, with the drag coefficient relatively consistent with the literature [58, 59]. Both LES WALE and IDDES SST $k-\omega$ models provided the most accurate predictions for C_d and C_l and were used in further analysis.

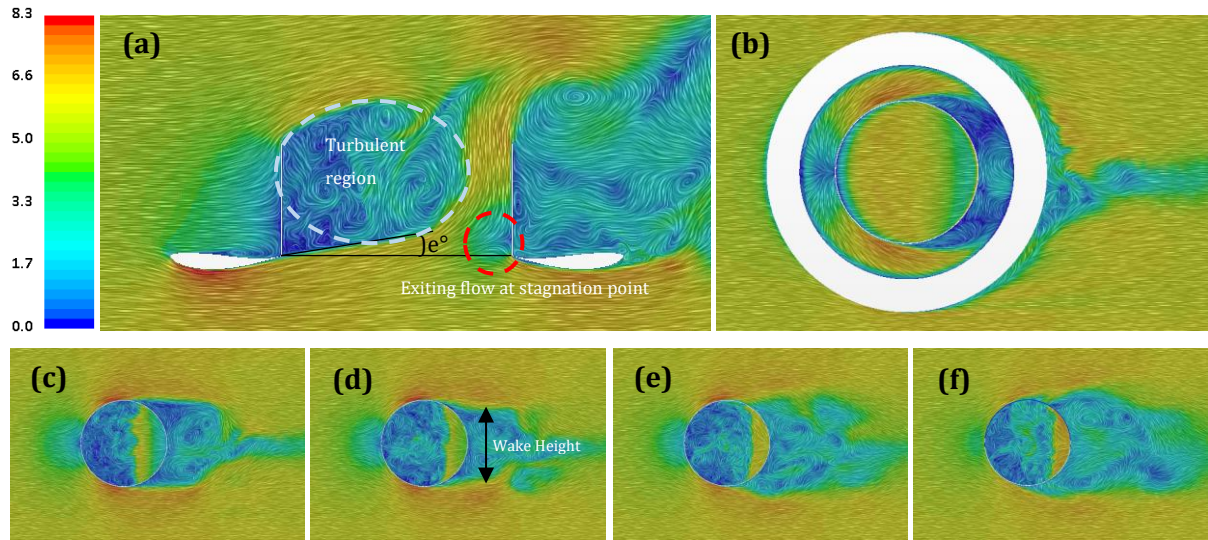
3.3 Aerodynamics characteristics and performance of the new shrouded system

3.3.1 Velocity contours and distribution

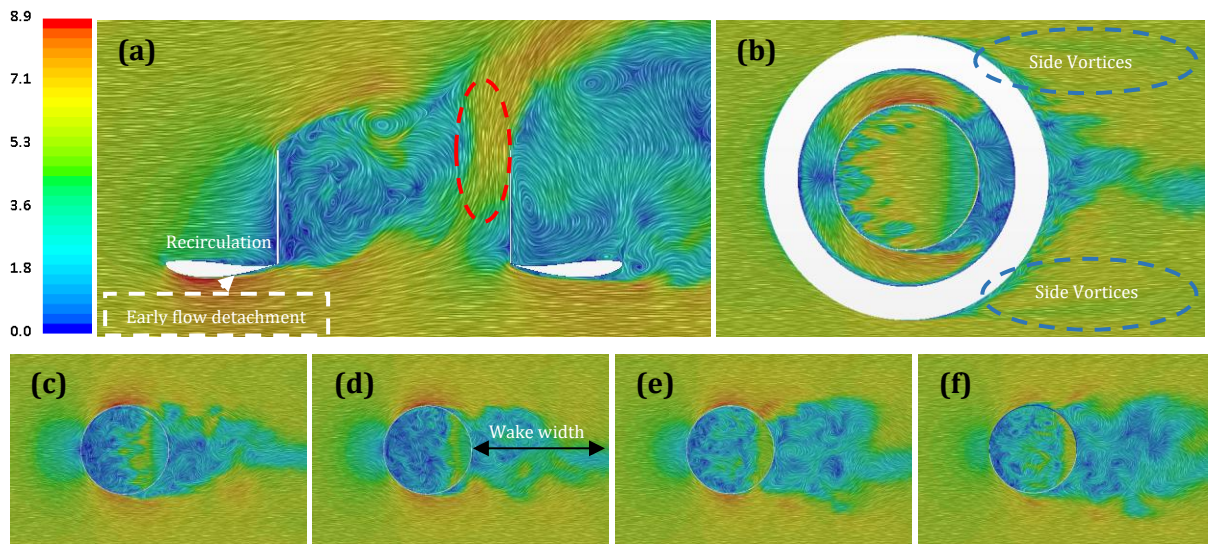
Figure 9 (i) shows in the DDES SST $k-\omega$ model that the flow accelerates at a higher entrance angle (e°), forming two distinct regions. On the left side, the flow is chaotic with recirculating bubbles, while the right side exhibits a more uniform flow with 112% free stream velocity over 75% of the case height. The wake is highly non-uniform, with separation occurring at the diameter and larger wake dimensions observed at higher planes, accompanied by more pronounced wing base vorticities.

Similarly, the LES WALE model (ii) depicts accelerated but less uniform flow. It shows two distinct regions: a turbulent left side and a more uniform right side over 50% of the case height. Detachment and reattachment occur at 75% of the wing base chord length. The LES WALE model better captures wing base vorticities, although both models exhibit complex turbulence and large wake formations, with flow exiting at 15% of the case height on the bottom right side.

It appears that only the LES WALE model clearly showed vorticities on the wing base side, more chaotic turbulence, and an early flow detachment on the suction side of the wing base, showing the relevance of using this model to characterise the flow.



(i) IDDES SST $k-\omega$



(ii) LES WALE

Figure 9. Vector convolution inside and around the shroud on section planes (a) $z=0$, (b) inlet, (c) $\frac{1}{4}H$, (d) $\frac{1}{2}H$, (e) $\frac{3}{4}H$, (f) outlet, for turbulent models: (i) IDDES SST $k-\omega$; (ii) LES WALE.

Figure 10 (a) shows the velocity profile from the IDDES SST $k-\omega$ model, where the flow enters the shroud at slightly above free stream velocity, decreasing at 90% of the shroud diameter. Velocity fluctuates between 10% and 20% in the first half, rising to 100% free stream speed before dropping on the right side. Middle planes reach 110% free stream velocity, stabilising the outlet at 120%. The LES WALE model as shown in Figure 10 (b) predicted an initial acceleration of 115%, followed by fluctuations between 10% and 50% within 50% of the case diameter, with higher inlet than outlet speeds. Figure 10 (c) visualises velocity ratios, showing flow evolution at various heights within the shroud. The difference between both models indicated that each model

had its own limits in modelling this system and both models are needed to capture the true characteristic of this system.

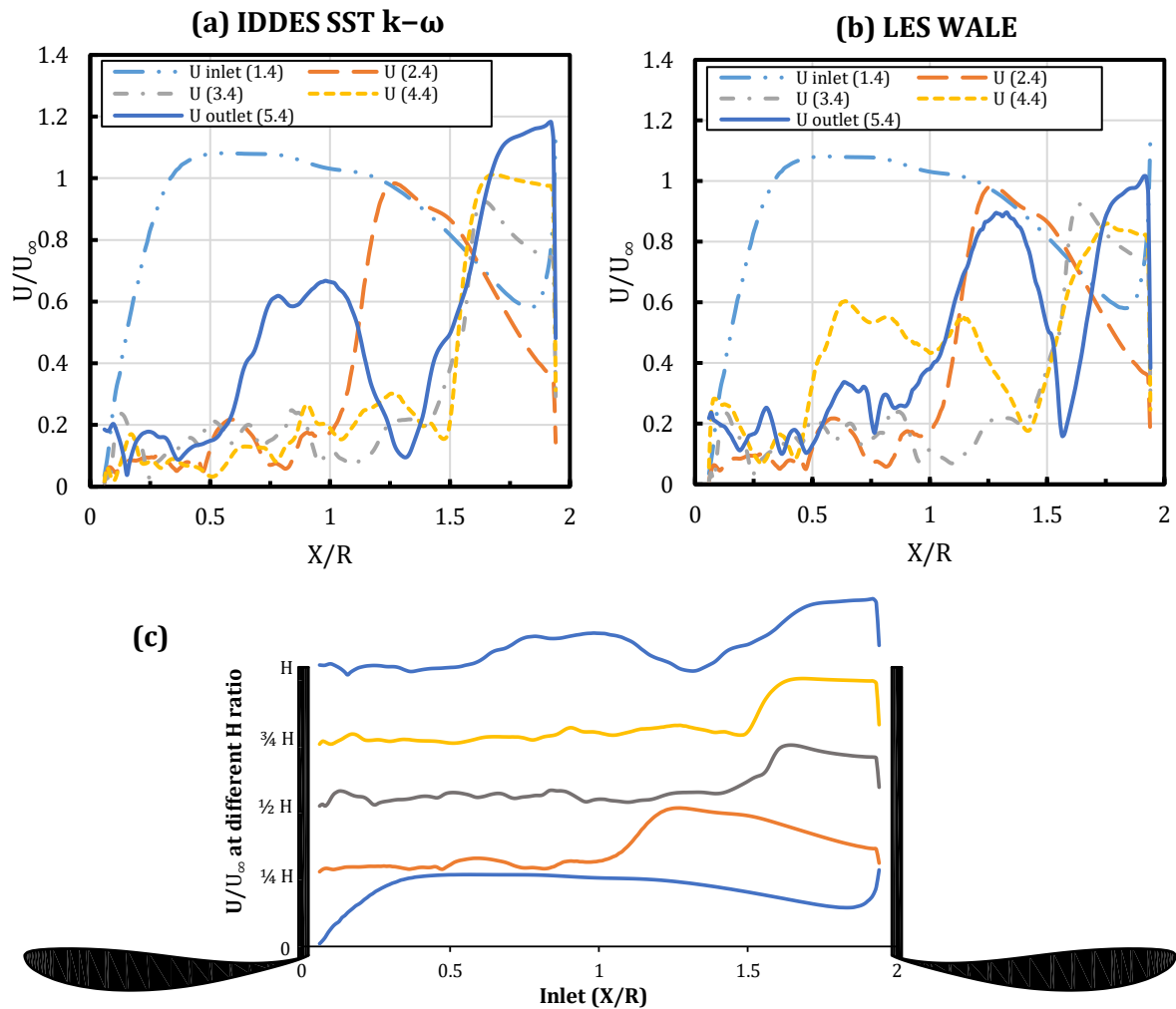


Figure 10. Normalised Internal velocity distribution diagram on radial lines probe 1.4, 2.4, 3.4, 4.4, 5.5: (a) IDDES SST k- ω ; (b) LES WALE; (c) visualisation of velocity ratio in the shroud.

Figure 11 (a) This shows that the maximum velocity ratio is its highest at the outlet, at 1.22, whereas the average velocity ratio is half at the outlet. The former decreases gradually to $1/4 H$, then slightly to $1/2 H$ and slightly rise to $3/4 H$ before increasing sharply to the outlet. On the other hand, the average velocity ratio decreases sharply from the inlet to $1/4 H$ before increasing progressively from $1/2 H$ to the outlet. The IDDES SST k- ω model had a better prediction with an average error of 0.5%, whereas the LES WALE had 1.3%. This then led to the evaluation of the mass flow rate, considering the geometric characteristics of the model and the non-uniform flow. [60]. The mass flow rate was calculated computationally to be 0.0298 kg/s.

Figure 11 (b) shows a sensitivity analysis of the variation of the speed ratio and mass flow rate under a changing wind direction angle relative to the horizontal flow direction, in a typical urban

area. It can be observed that the maximum speed ratio about 1.5 was achieved when a wind flow arrived at the turbine system at an angle of 30 degrees. This ratio increased and decreased rapidly between the angles of 20 and 40 degrees although it increased steadily with the wind direction angle. In addition, the average speed ratio rose strongly across the range of angles investigated. It also can be seen that the mass flow rate rises sharply with the direction of the wind.

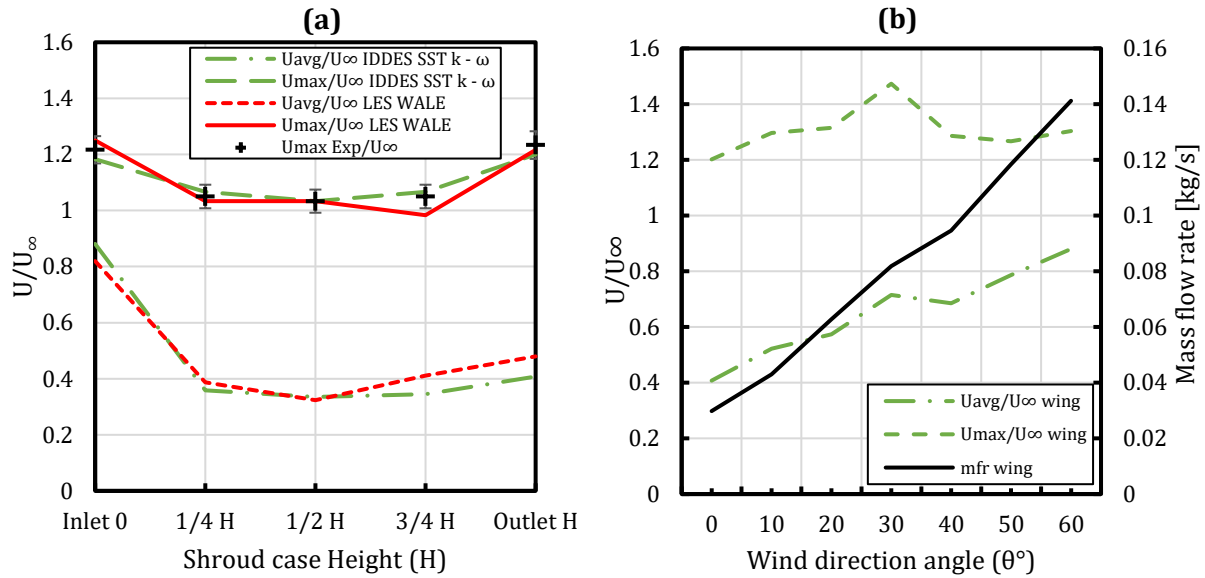


Figure 11. (a) The average and maximum velocity ratio along the shroud case height for IDDES SST $k-\omega$ and LES WALE models compared with the experiment; (b) The average, maximum velocity ratio and mass flow rate (mfr) at the outlet plane with changing wind direction.

Both IDDES SST $k-\omega$ and LES WALE have shown the turbulent region velocity scale at the planes' entrance, where airspeed fluctuates wildly up to $\frac{3}{4}$ of the case diameter. The IDDES SST $k-\omega$ showed that flow reached its maximum velocity at the casing outlet. The literature also supports this flow profile at a bend, where the turbulent flow in a 90-degree water pipe elbow at $Re = 10^4$ was investigated using LES and PIV (Particle imagery velocimetry) [61, 62]. In addition, the velocity ratios are at their highest at the inlet and the outlet with the maximum velocity at the outlet. Hence, the inlet and outlet would be significant positions for the turbine rotor blade.

The observed superior performance of the IDDES SST $k-\omega$ model over LES WALE in this scenario where uniform and non-uniform flows interact very closely and gives complex flow transition areas of high pressure gradient may be due to their different approaches. By associating RANS and LES methodologies, IDDES SST $k-\omega$ can achieve both computational efficiency and high accuracy in transitional flows. This is because the IDDES dynamically switches between RANS near walls, ensuring stable and efficient predictions of boundary layer behavior, and LES in regions of separation or unsteadiness, capturing large-scale turbulent structures in handling flow transitions [63]. This adaptive mechanism of combining RANS for uniform flow regions and LES

for non-uniform turbulent regions allows accurate modeling of complex transitions in velocity and flow type, such as the flow inside the omni-low shroud. Whereas, LES WALE relies entirely on resolving turbulence, requiring fine grids everywhere [64]. An insufficient resolution can lead to errors in capturing flow physics in transitional regions, particularly at low Reynolds numbers. Similarly, the $k-\omega$ SST component in IDDES models near-wall turbulence without needing to resolve all turbulent scales, crucial in low Reynolds number flows where boundary layers frequently separate and reattach [65]. LES WALE, however, demands very high resolution near walls. Insufficient grid refinement can degrade its accuracy, leading to errors in predicting separation and reattachment [66].

Figure 12 shows the velocity contours at the outlet under different wind direction angles. It can be observed that the higher angle of the wind direction is, the more uniform the flow is inside of the shroud case. The maximum speed was found to be 8.84 m/s at 30 degrees. The area covered by the uniform flow at 60 degrees is about 70% of the shroud cross sectional area compare to 12% at 0 degree. Overall, this shows the impact of the wind drection on the shroud turbine performace and the effectiveness of of such system in urban areas. The system manages to attain 1.5 maximum speed ratio under specific conditions which contribute to further enhance the turbine efficiency.

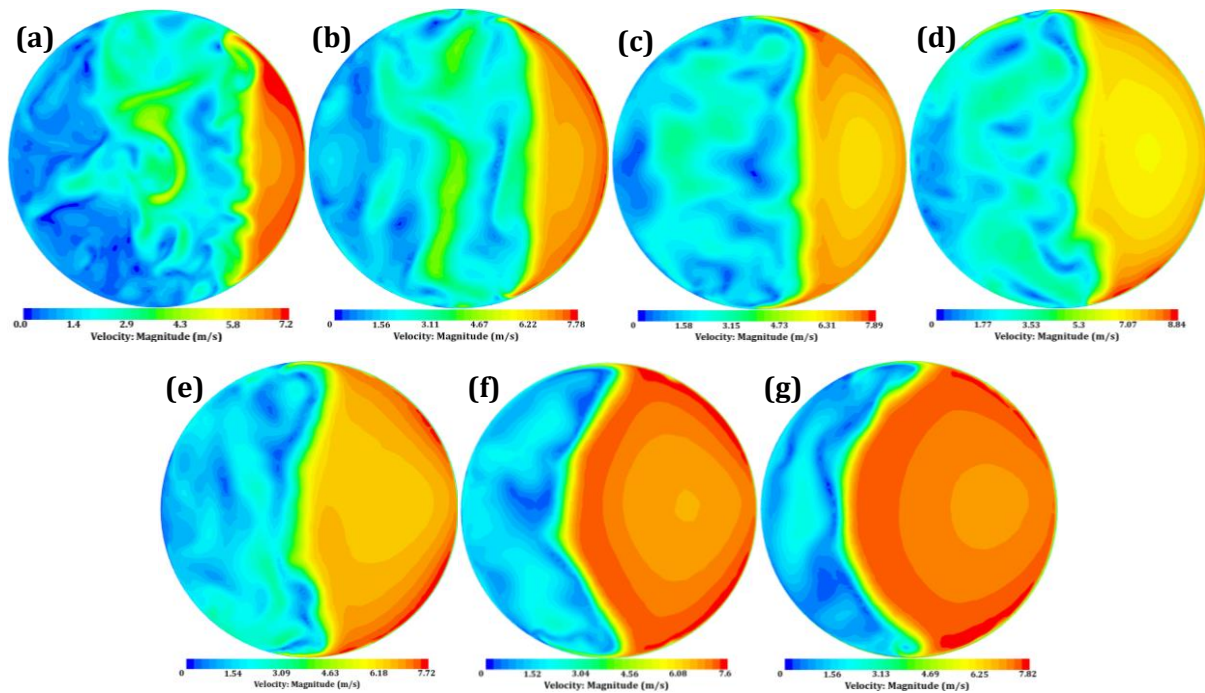


Figure 12: velocity contour at the outlet of the shroud at different wind direction: (a) $\theta=0^\circ$; (b) $\theta=10^\circ$; (c) $\theta=20^\circ$; (d) $\theta=30^\circ$; (e) $\theta=40^\circ$; (f) $\theta=50^\circ$; (g) $\theta=60^\circ$.

3.3.2 Pressure coefficient distribution on the shroud.

Figure 13 shows the distribution of the pressure coefficient. The IDDES SST $k-\omega$ model recorded C_p values up to 1 at the front and internal bottom, with a low of -1.3356 on the external sides. The LES WALE model has also predicted high internal C_p values up to 1 and a low of -2.2901. Both models have predicted the highest C_p values at the front and bottom right of the shroud, with IDDES SST $k-\omega$ and LES WALE reaching approximately 1. The LES WALE model provided the lowest external C_p values, while significant low and negative C_p values were about 80% of the internal shroud volume. Both models demonstrated a more gradual pressure gradient.

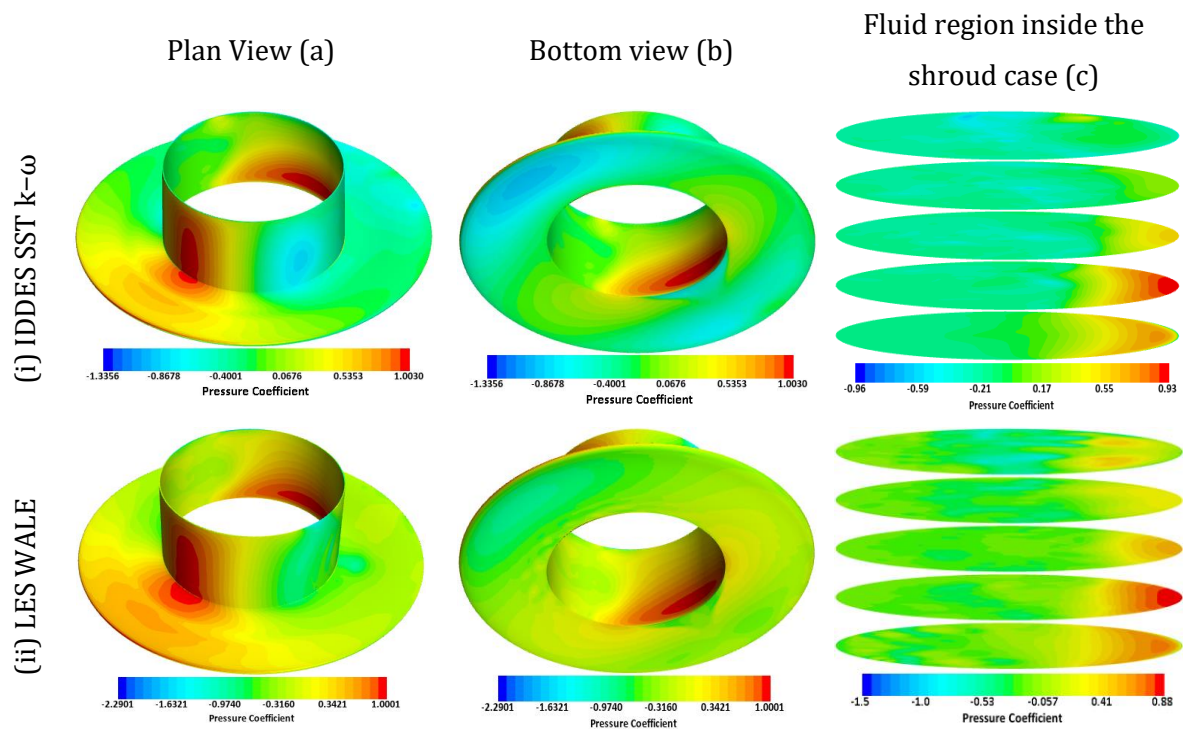


Figure 13. Pressure coefficient around the shroud (a, b) and in the fluid region inside the shroud Case (c).

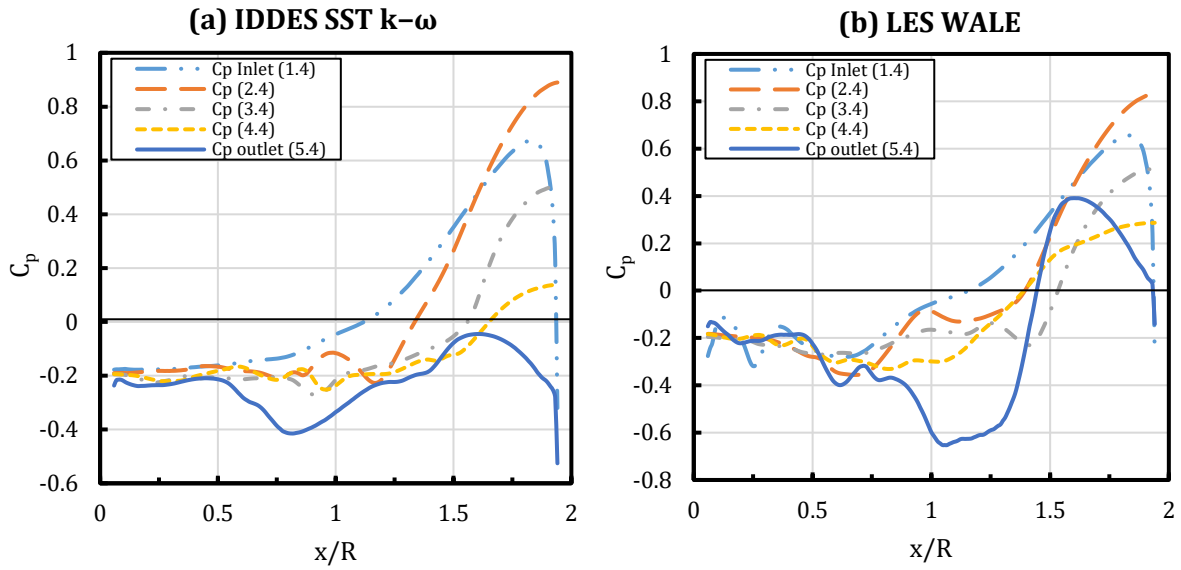


Figure 14. Internal pressure coefficient distribution on radial line probes 1.4, 2.4, 3.4, 4.4, 5.5 along the y-axis: (a) IDDES SST $k-\omega$; (b) LES WALE.

The IDDES SST $k-\omega$ model, as shown in Figure 14 (a), exhibited low negative C_p values over approximately 50% of the case volume and a maximum C_p of 0.9 on line probe 2.4 on the right side, with fluctuations in the centre region and decreased C_p with height. Figure 14 (b), the LES WALE model predicted slightly fluctuating C_p values decreasing over about 40% of the shroud volume before increasing steadily, with the highest C_p also on line probe 2.4 on the right side; the outlet plane features the lowest negative C_p of about -0.65 at the centre, rising sharply before a sudden drop.

The shroud shows consistent pressure coefficient (C_p) trends, with low C_p values on the left and high positive C_p values on the right. The most significant C_p differences occur between line probe 2.4 and outlet line probe 5.3, indicating flow acceleration on the right and turbulence on the left.

3.3.3 C_p comparison with experimental data

Figure 15 (a) shows the coefficient of pressure (C_p) distribution relative to the wing base chord length (X/C). C_p decreases sharply on the suction side up to 30% of the chord, reaching a minimum of -0.91, and then increases smoothly toward the trailing edge. Both models show a similar C_p trend up to 66% of the chord, though LES WALE shows fluctuations at the trailing edge, indicating turbulence near the wing base-shroud joint. IDDES SST $k-\omega$ showed a lower relative error of 4.5%, compared to 4.9% for LES WALE. Figure 15 (b) illustrates C_p distribution within the shroud case at $3/4$ of its height. Both models, supported by experimental data, predict two

distinct regions, with C_p decreasing toward the middle of the shroud and increasing nearly 25% of the case diameter on the right. The turbulence on the left side matches both model predictions.

The IDDES SST $k-\omega$ model best matches experimental C_p along the wing base chord. In contrast, LES WALE best predicts C_p along the shroud's inner diameter, emphasising the importance of using both models in this study.

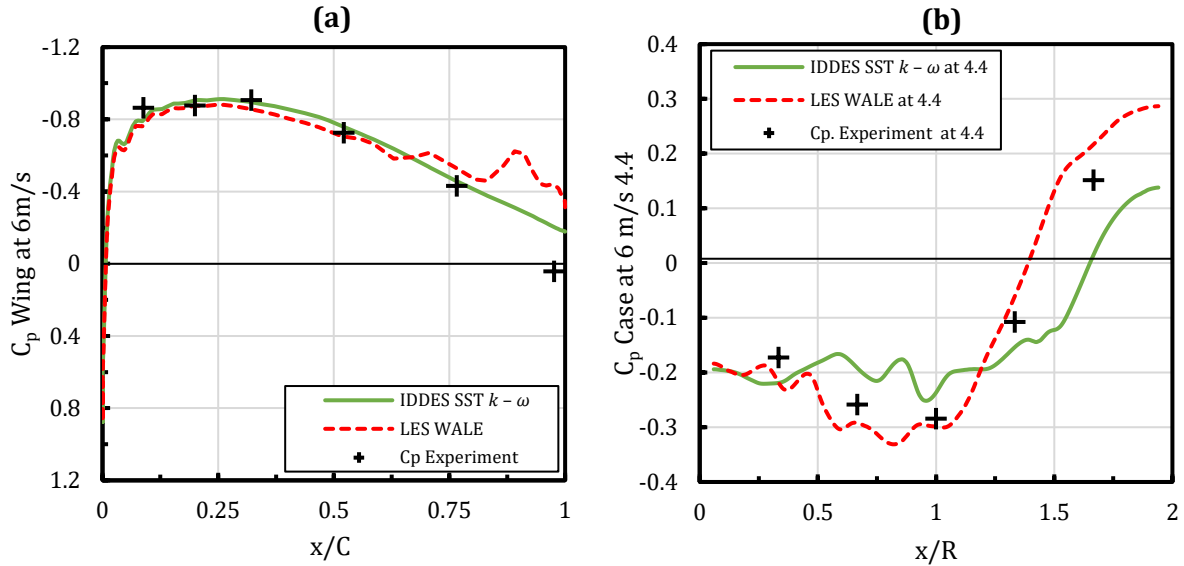


Figure 15. Experimental and numerical distribution of C_p at 6m/s ($Re = 145000$): (a) across the bottom-suction side of the wing base against the chord length; (b) C_p across the shroud case diameter; velocity ratio along the radial line probe at different height case positions.

3.3.4 Skin friction coefficient

This analysis is crucial for understanding the viscous forces exerted by the fluid on the surface of the shroud, thus allowing better optimisation of the aerodynamic design and improving the shroud's flow-converting performance.

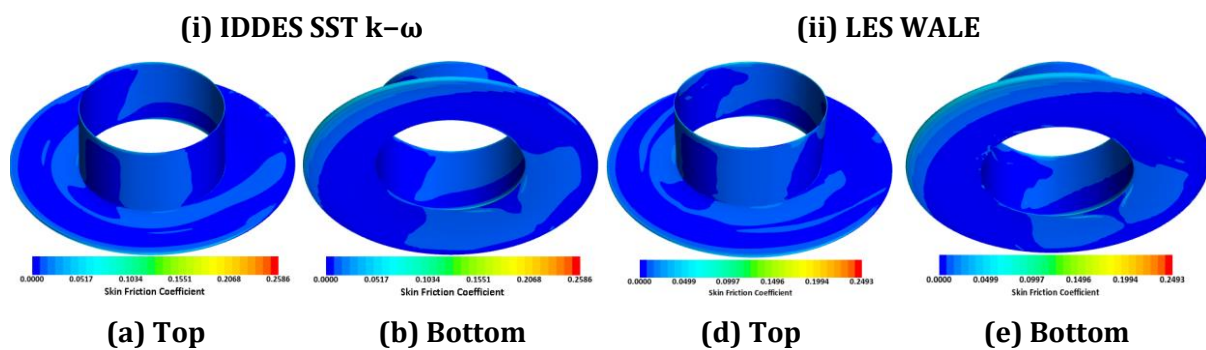


Figure 16. Skin friction coefficient distribution on the shroud surface.

Figure 16 shows that both models have relatively predicted similar general representations of the skin friction on the top surface of the shroud and very distinct features of the bottom side of the shroud (suction) side. The LES WALE model did the lowest high C_f of 0.2493, indicating attached flow along the bottom side of the wing base for the first 25% of its chord length. It appeared that C_f gradually decreased towards the entrance of the shroud. The LES WALE model revealed a subtle increase in C_f just before flow entry, indicating reattachment or acceleration, with C_f being relatively high on the right internal side of the case but dropping sharply to near zero at 25% of the case height and across the left internal sides, covering just over 60% of the cylindrical internal surface area.

As shown in Figure 16, the flow stays attached to the bottom side of the wing (suction side) as the skin friction coefficient decreases gradually on the area. However, the LES WALE has shown that the flow accelerated and attached near the entrance of the case, predicting a small region at about 80% of the wing chord length of flow with slight separation and reattachment. Internally, the sharp difference in C_f on the surface of the case further reinforced the presence of two distinct regions of flow: a high-speed and uniform flow on the right side (about 30%) of the case and a more prominent region of flow turbulence on the left side (about 70%).

3.3.5 Flow visualisation in the wind tunnel

Figure 17 (a) The illustration in the dashed red line shows an approximate S shape curve. The angle of the line of diversions in the lower part of the S characteristic profile was 14 degrees. Subsequently, the flow progressed into an elbow on the left side of the uniform flow. The smoke region represented the region of relatively uniform flow, and the clearer region on the left side was the flow dominated by small-scale turbulences. The flow progressed into an almost vertical stream of uniform flow on 30% of the case height, then exited the case.

Small strings were attached to tiny parallel lines in the direction of the diameter. Figure 17 (b) shows tufts pointing down on the left side and pointing up on the right side, as illustrated by the blue and red arrows, respectively. This behaviour of the tufts also revealed high turbulence with small-scale vortices in the region, as indicated by vibrating tufts. Smoke and tuft analyses in Figures 15(a)-(b) at six m/s corroborate with the S-shaped flow profile depicted in the IDDES SST $k-\omega$ models in Figure 9, highlighting uniform regions and turbulence characteristics.

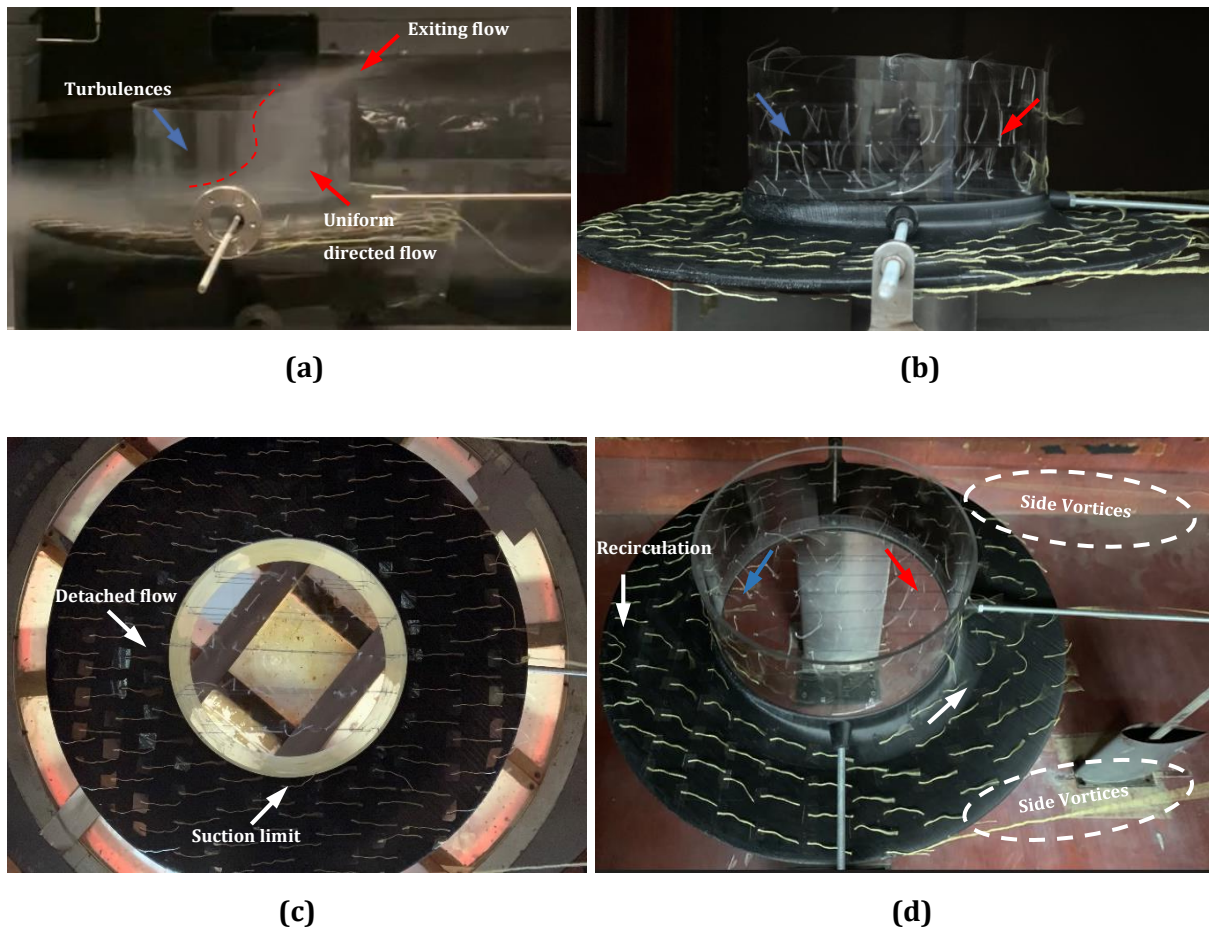


Figure 17. Flow visualisation at 6m/s of the shroud using transparent acrylic case: (a) smoke front view, (b) Tuft front view, (c) the bottom-suction side of the shroud, (d) the top of the shroud.

The flow on the wing base remains attached over most of the area, with localised detachment observed near the entrance of the case at approximately 80% of the wing base chord length, as indicated by vibrating tufts. Figure 17 (c). Flow detachment is also evident from the sides of the case, with turbulence developing at the rear of the wing base on the right side. On the top view of the shroud Figure 17 (d) . Three distinct flow regions are observed: detachment and reattachment at about 25% of the wing base chord length, full detachment and turbulence behind the shroud just after the vertical diameter, and small-amplitude rotation of tip vortices at the wing base end. A comparison with the wind tunnel results has showed that the LES WALE model most accurately depicted flow detachment and reattachment at approximately 80% of the chord length. Thus, the LES WALE model provided better predictions for skin friction coefficient and wing base side vortices, as shown in Figure 17.

3.4 Performance of wind turbine

The investigation of the ideal position of the turbine rotor blade between the inlet and the outlet during the experiment showed the turbine starts at lower free stream air speed when placed at the outlet. In addition, the flow does not have enough impulse at lower speed to start the rotor at the inlet position than at the outlet position. The rotor hub was found to have a significant impact on the starts of the rotor at the inlet. Thus, the outlet position of the rotor blade was employed for the rest of the investigation.

Further wind tunnel performance investigations were performed, due to the presence of a stagnation point on the lower right side of the shroud as shown in Figure 9, leading to a localized flow exiting the system, resulting in a minor energy loss. It was found that this reverse flow did not significantly affect the turbine's rotational performance hence its stability as they were almost no detectable change in the rotational speed of the rotor blade. The shroud's aerodynamic design ensured effective redirection of a great potential of the incoming flow toward the turbine blades, maintaining optimal energy transfer. Furthermore, the turbine's demonstrated self-starting capability and operational efficiency at low wind speeds further validated the system's ability to mitigate localized losses, preserving its overall performance.

Figure 18 (a) shows the relationship between the wind speed and the rotating speed of the turbine blade which identifies the linear relationship between the wind speed and the RPM. Figure 18 (b) shows that power was generated by the turbine fitted in the shroud. At 6 m/s, the turbine blade rotated at the angular speed of 840 RPM, which produced a power ratio of about 23 W/m². The turbine started rotating and producing power at a free stream air speed of 2.1 m/s though the shroud could channel air into the shroud case at free stream as low as 1.5m/s. The power ratio increased rapidly as the blade rotated faster in this range of free stream speed. The power output of this novel wind turbine exhibited an initial gradual increase with RPM, followed by a sharp rise as RPM surpasses approximately 800. This behaviour indicates a non-linear relationship between RPM and power, which becomes increasingly pronounced at higher RPM values.

Figure 18 (c) shows the relationship between power ratio output (W/m²) and the tip-speed ratio λ for a wind turbine. The power ratio increases steadily for λ values up to 2. When $\lambda > 2$, the turbine output increases sharply, indicating a more than proportional increase in power with increasing λ . This sharp rise in power output suggests that the wind turbine becomes significantly more efficient as the blade tip speed increases relative to the wind speed. The curve reaches its highest point when λ approaches 2.5, with the maximum power ratio value just above 60 W/m².

Figure 18 (d) illustrates the distribution for the power coefficient with respect to the tip speed ratio (TSR) which shows a sharp initial increase up to TSR 0.5, then progresses into a steady rise before soaring from TSR 2.2. This reflected the efficiency of the turbine and the self-starting capability. Figure 18 (d) also shows that the novel shrouded system reaches C_{power} value higher than the Betz's limit of 0.59 demonstrating it superior efficiency. It also shows that the turbine is performing better at low tips speed compared to many current shrouded wind turbines [31].

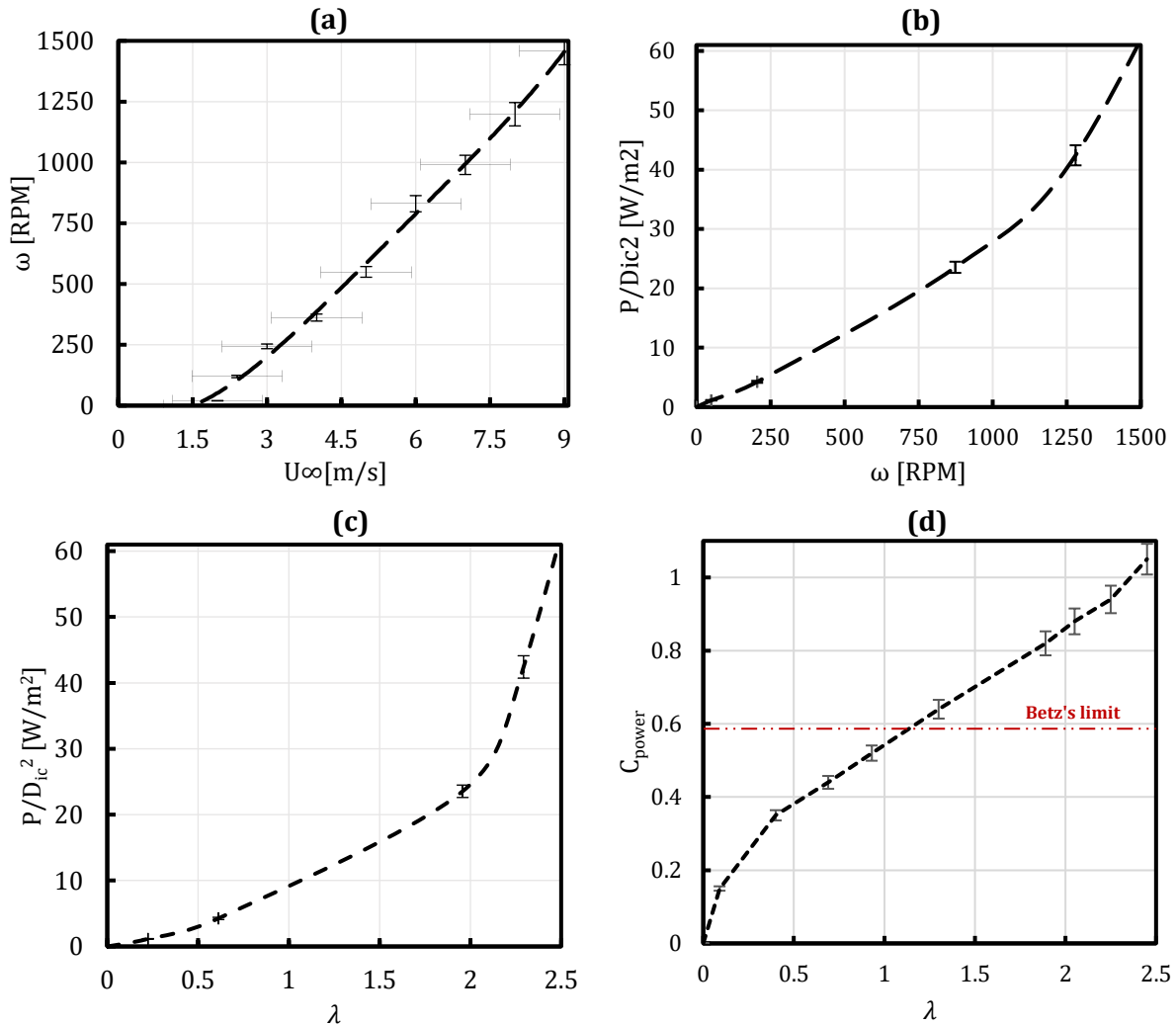


Figure 18. Wind turbine performance Characteristics: (a) Turbine rotational speed across free stream range, (a) Ratio of wind power to shroud case diameter squared across turbine rotation speed vs λ , (c) Ratio of wind power to shroud case diameter squared across tip speed ratio; (d) Coefficient of power to the tip speed ratio.

The optimal efficiency was achieved at higher RPMs, particularly beyond 1000 RPM, where the power ratio output exceeded 28 W/m² and approached 61 W/m² sharply. It is evident that this novel wind turbine design has a significant efficiency improvement at higher rotational speeds,

highlighting its potential for enhanced energy capture under varying wind conditions and converting efficiently wind energy to electrical power at higher speeds.

The power ratio output demonstrated a marked increase with the tip-speed ratio λ , particularly within the range of 2 to 2.5, indicating that the wind turbine start attains high efficiency at these λ values. Notably, at approximately 2.1 m/s, $\lambda = 0.25$ and at 6 m/s λ equals about 2. This pronounced rise in power ratio output at higher λ values implies that the turbine operates with maximum efficiency within a specific λ range. Therefore, the investigation showed that this initial optimal λ range demonstrates the turbine-fitted shroud relevance at low airspeed and indicates the crucial area for maximising power output and overall efficiency.

4 Conclusions

The aerodynamic characteristics and performance of the flow around a new shrouded vertical axis wind turbine have been investigated with turbulence models and a wind tunnel for more efficient wind energy capture systems. Based on the above results and discussion, the following conclusions can be drawn:

- i) The innovative shrouded vertical axis wind turbine effectively worked in typical low Reynolds number air flow in the urban area and all models effectively predicted the suction and acceleration of external flow into the shroud including high velocity, the point of flow separation. It also works better at high wind direction angle reaching maximum speed of 1.5.
- ii) Both IDDES SST k- ω and LES WALE models demonstrated a strong agreement with experimental data, accurately predicting velocity fluctuations and mass flow rate. While LES WALE exhibited slightly superior performance in characterising flow detachment, skin friction coefficient (C_f), and pressure coefficient (C_p) in the complex shroud region, IDDES SST k- ω excelled in predicting velocity ratio, lift coefficient (C_l), and a favourable balance of accuracy and computational efficiency. It models boundary layers and regions of uniform flow using RANS, reducing computational demand. It resolves large-scale turbulence separated regions using LES, ensuring accurate predictions of flow mixing and velocity ratios. Its adaptive nature minimizes the errors introduced in regions of complex transition.
- iii) The shroud effectively diverted free-stream airflow at speeds as low as 1.5 m/s for energy harvesting. The resulting accelerated flow within the shroud exhibited a characteristic S-shaped profile, culminating in a stagnation point that caused a portion of the flow to exit the shroud at the inlet.

- iv) This innovative shrouded turbine demonstrated exceptional self-starting capabilities, initiating operation at a low cut-in speed of 2.1 m/s with power output ranging from 1.14 W/m² to 60 W/m² and increasing nonlinearly with increasing wind speed. The optimal efficiency was achieved within a tip-speed ratio range of 2 to 2.5. The turbine ensures high efficiency in omnidirectional wind conditions, overcoming the directional limitations of conventional shrouded.
- v) The findings underscore the strategic placement of the turbine rotor blade at the outlet and the potential for substantial efficiency gains and enhanced energy harvesting through this novel vertical-shrouded turbine design, contributing significantly to SDG 7: Affordable and Clean Energy. The proposed solution offers improved performance, reduced maintenance, and a visually integrated design suitable for urban and rural applications.

Although, the attributes of this shrouded turbine system have been established, the present study has some limitations. Firstly, as the turbine system was numerical simulated for one wind speed regime, further investigation should be done to determine the performance and characteristic under varying wing speed. Secondly, more study could be done utilizing other high performance low Reynold number aerofoil. Finally, further wind tunnel experiments should be done in a higher speed to draw the complete power coefficient to tip speed ratio of this novel shrouded vertical axis wind turbine.

Declaration of competing interest

The authors wish to confirm that there are no known conflicts of interest associated with this publication and there has been no significant financial support for this work that could have influenced its outcome.

Acknowledgements

The authors would like to acknowledge the University of Hertfordshire, which supported this study.

References

- [1] R. Özkan and M. S. Genç, "Aerodynamic design and optimization of a small-scale wind turbine blade using a novel artificial bee colony algorithm based on blade element momentum (ABC-BEM) theory," *Energy Conversion and Management*, vol. 283, 2023.

- [2] UN, "The Sustainable Development Goals Report 2023: Special edition," United Nations, 2023.
- [3] S. Boadu and E. Otoo, "A comprehensive review on wind energy in Africa: Challenges, benefits," *Renewable and Sustainable Energy Reviews*, vol. 191, 2024.
- [4] P. Ying, Y. Chen, Y. Xu and Y. Tian, "Computational and experimental investigations of an omni-flow wind turbine," *Applied Energy*, vol. 146, pp. 74-83, 2015.
- [5] T. Abbasi, M. Premalatha, T. Abbasi and S. Abbasi, "Wind energy: Increasing deployment, rising environmental concerns," *Renewable and Sustainable Energy Reviews*, vol. 31, pp. 270-288, 2014.
- [6] Y. Teff-Seker, O. Berger-Tal, Y. Lehnardt and N. Teschner, "Noise pollution from wind turbines and its effects on wildlife: A cross-national analysis of current policies and planning regulations," *Renewable and Sustainable Energy Reviews*, vol. 168, 2022.
- [7] S. Li, Y. Li, C. Yang, Q. Wang, B. Zhao, D. Li, R. Zhao, T. Ren, X. Zheng, Z. Gao and W. Xu, "Experimental investigation of solidity and other characteristics on dual vertical axis wind turbines in an urban environment," *Energy Conversion and Management*, vol. 229, 2021.
- [8] A. S. Saad, I. I. El-Sharkawy, S. Ookawara and M. Ahmed, "Performance enhancement of twisted-bladed Savonius vertical axis wind turbines," *Energy Conversion and Management*, vol. 209, 2020.
- [9] K. Calautit and C. Johnstone, "State-of-the-art review of micro to small-scale wind energy harvesting technologies for building integration," *Energy Conversion and Management: X*, vol. 20, 2023.
- [10] P. Zhao, Y. Jiang, S. Liu, T. Stoesser, L. Zou and K. Wang, "Investigation of fundamental mechanism leading to the performance improvement of vertical axis wind turbines by deflector," *Energy Conversion and Management*, vol. 247, 2021.
- [11] B. Bazooyar and H. G. Darabkhani, "Design, manufacture and test of a micro-turbine renewable energy combustor," *Energy Conversion and Management*, vol. 213, 2020.
- [12] A. Hijazi, A. ElCheikh and M. Elkhoury, "Numerical investigation of the use of flexible blades for vertical axis wind turbines," *Energy Conversion and Management*, vol. 299, 2024.

- [13] S. M. H. D. Perera, G. Putrus, M. Conlon, M. Narayana and K. Sunderland, "Wind Energy Harvesting and Conversion Systems: A Technical Review," *Energies*, vol. 15, no. 24, p. 9299, 2022.
- [14] M. S. Siddiqui, M. H. Khalid, R. Zahoor, F. S. Butt, M. Saeed and A. W. Badar, "A numerical investigation to analyze effect of turbulence and ground clearance on the performance of a roof top vertical-axis wind turbine," *Renewable Energy*, vol. 164, pp. 978-989, 2021.
- [15] M. M. A. Bhutta, N. Hayat, A. U. Farooq, Z. Ali, S. R. Jamil and Z. Hussain, "Vertical axis wind turbine – A review of various configurations and design techniques," *Renewable and Sustainable Energy Reviews*, vol. 16, pp. 1926-1939, 2012.
- [16] N. Verma and B. D. Baloni, "Influence of Reynolds number on the performance of small horizontal axis wind turbine with fixed speed operation," *Energy Sources, Part A: Recovery, Utilization, and Environmental Effects*, vol. 44, no. 2, pp. 4016-4031, 2022.
- [17] J. Sumanik-Leary, . M. Delor, M. Little, M. Bellamy, A. Williams and S. Williamson, *Engineering in Development: Energy*, London: Engineers Without Borders UK, 2014.
- [18] A. Dilimulati, T. Stathopoulos and M. Paraschivoiu, "Wind turbine designs for urban applications: A case study of shrouded," *Journal of Wind Engineering & Industrial Aerodynamics*, vol. 175, pp. 179-192, 2018.
- [19] P. Ying, Y. K. Chen and Y. G. Xu, "An aerodynamic analysis of a novel small wind turbine based on impulse turbine principles," *Renewable Energy*, vol. 75, pp. 37-43, 2015.
- [20] D. Qian, Y. Li, F. Niu and Z. O'Neill, "Nationwide savings analysis of energy conservation measures in buildings," *Energy Conversion and Management*, vol. 188, pp. 1-18, 2019.
- [21] M. Ghasemian, Z. N. Ashrafi and A. Sedaghat, "A review on computational fluid dynamic simulation techniques for Darrieus vertical axis wind turbines," *Energy Conversion and Management*, vol. 149, pp. 87-100, 2017.
- [22] F. Arpino, M. Scungio and G. Cortellessa, "Numerical performance assessment of an innovative Darrieus-style vertical axis wind turbine with auxiliary straight blades," *Energy Conversion and Management*, vol. 171, pp. 769-777, 2018.
- [23] L. Santamaría, J. M. F. Oro, K. M. A. Díaz, A. Meana-Fernández, B. Pereiras and S. Velarde-Suárez, "Novel methodology for performance characterization of vertical axis wind

turbines (VAWT) prototypes through active driving mode," *Energy Conversion and Management*, vol. 258, 2022.

- [24] K. H. Wong, W. T. Chong, N. L. Sukiman, Y.-C. Shiah, S. C. Poh, K. Sopian and W.-C. Wang, "Experimental and simulation investigation into the effects of a flat plate deflector on vertical axis wind turbine," *Energy Conversion and Management*, vol. 160, pp. 109-125, 2018.
- [25] Y. Ohya and T. Karasudani, "A Shrouded Wind Turbine Generating High Output Power with Wind-lens Technology," *energies*, vol. 3, p. 634-649, 2010.
- [26] A. Grant, C. Johnstone and N. Kelly, "Urban wind energy conversion: The potential of ducted turbines," *Renewable Energy*, vol. 33, pp. 1157-1163, 2008.
- [27] D. Allaei and Y. Andreopoulos, "INVELOX: Description of a new concept in wind power and its performance evaluation," *Energy*, vol. 69, pp. 336-344, 2014.
- [28] M. M. Ghorani, B. Karimi, S. M. Mirghavami and Z. Saboohi, "A numerical study on the feasibility of electricity production using an optimized wind delivery system (Invelox) integrated with a Horizontal axis wind turbine (HAWT)," *Energy*, vol. 268, 2023.
- [29] S. Siahpour, F. N. Khakiani, V. Fazlollahi, A. Golozar and F. A. Shirazi, "Morphing Omni-directional Panel Mechanism: A novel active roof design for improving the performance of the wind delivery system," *Energy*, vol. 217, 2021.
- [30] M. M. Ngouani Siewe, Y. K. Chen, R. Day and O. David-West, "Low-Speed Aerodynamic Analysis Using Four Different Turbulent Models of Solver of a Wind Turbine Shroud," in *Energy and Sustainable Futures*, 2021.
- [31] A. Ilhan, B. Sahin and M. Bilgili, "A review: diffuser augmented wind turbine technologies," *International Journal of Green Energy*, vol. 19, no. 1, p. 1-27, 2021.
- [32] M. S. Selig and B. D. McGranahan, "Wind tunnel aerodynamic tests of six airfoils for use on small wind turbines," *AIAA*, vol. 1188, 2004.
- [33] D. Fulton, P. Gallagher, W. Grannan, J. Martin, N. Mastej and B. Wujek, "The Bunny: A Simulated Commercial Air Transportation Study," 1993.

- [34] C. Attie, A. ElCheikh, J. Nader and M. Elkhoury, "Performance enhancement of a vertical axis wind turbine using a slotted deflective flap at the trailing edge," *Energy Conversion and Management*, vol. 273, 2022.
- [35] A. Tummala, R. Kishore Velamati , D. Kumar Sinha, V. Indraja and V. Hari Krishna, "A review on small scale wind turbines," *Renewable and Sustainable Energy Reviews*, vol. 56, pp. 1351-1371, 2016.
- [36] J. Thé and H. Yu, "A critical review on the simulations of wind turbine aerodynamics focusing on hybrid RANS-LES methods," *Energy*, vol. 138, pp. 257-289, 2017.
- [37] Y. Celik, D. Ingham, L. Ma and M. Pourkashanian, "Design and aerodynamic performance analyses of the self-starting H-type VAWT having J-shaped aerofoils considering various design parameters using CFD," *Energy*, vol. 251, 2022.
- [38] M. Maali Amiri, M. Shadman and S. F. Estefen, "URANS simulations of a horizontal axis wind turbine under stall condition using Reynolds stress turbulence models," *Energy*, vol. 213, 2020.
- [39] R. Zangeneh, "Assessing IDDES-Based Wall-Modeled Large-Eddy Simulation," *Fluids*, vol. 6, no. 7, p. 246, 2021.
- [40] H. Lei, D. Zhou, Y. Bao and Z. Han, "Three-dimensional Improved Delayed Detached Eddy Simulation of a two-bladed vertical axis wind turbine," *Energy Conversion and Management*, vol. 133, pp. 235-248, 2017.
- [41] P. Spalart, M. Strelets and A. Travin, "Direct numerical simulation of largeeddy-break-up devices in a boundary layer," *International Journal of Heat and Fluid Flow*, vol. 27, p. 902–910, 2006.
- [42] L. Xiao, Z. Xiao, Z. Duan and S. Fu, "Improved-Delayed-Detached-Eddy Simulation of cavity-induced transition in hypersonic boundary layer," *International Journal of Heat and Fluid Flow*, vol. 51, pp. 138-150, 2015.
- [43] M. L. Shur, P. R. Spalart, M. K. Strelets and A. K. Travin, "A hybrid RANS-LES approach with delayed-DES and wall-modelled LES capabilities," *International Journal of Heat and Fluid Flow*, vol. 29, no. 6, pp. 1638-1649, 2008.

- [44] R. S. Mirsane and F. Torabi, "An innovative method of investigating the wind turbine's inflow speed in a wind farm due to the multiple wake effect issue," *Energy Conversion and Management*, vol. 269, 2022.
- [45] L. J.L. Stival, J. R. Brinkerhoff, J. M. Vedovotto and F. Oliveira de Andrade, "Wake modeling and simulation of an experimental wind turbine using large eddy simulation coupled with immersed boundary method alongside a dynamic adaptive mesh refinement," *Energy Conversion and Management*, vol. 268, 2022.
- [46] K. Wijesooriya, D. Mohotti, K. Chauhan and D. A. S. G. Dias-da-Costa, "Numerical investigation of scale resolved turbulence models (LES, ELES and DDES) in the assessment of wind effects on supertall structures," *Journal of Building Engineering*, vol. 25, 2019.
- [47] J. B. Barlow, A. Pope and W. H. Rae, *LOW-SPEED WIND*, NewYork: John Wiley & Sons, 1999.
- [48] T. Syawitri, Y. Yao, B. Chandra and J. Yao, "Comparison study of URANS and hybrid RANS-LES models on predicting vertical axis wind turbine performance at low, medium and high tip speed ratio ranges," *Renewable Energy*, vol. 168, pp. 247-269, 2021.
- [49] M. Ghasemian and A. Nejat, "Aerodynamic noise prediction of a Horizontal Axis Wind Turbine using Improved Delayed Detached Eddy Simulation and acoustic analogy," *Energy Conversion and Management*, vol. 99, pp. 210-220, 2015.
- [50] A. Dinh Le, P. Nguyen Thi Thu, V. Ha Doan, H. The Tran, M. Duc Binh and V.-T. Truong, "Enhancement of aerodynamic performance of Savonius wind turbine with airfoil-shaped blade for the urban application," *Energy Conversion and Management*, vol. 310, 2024.
- [51] M. Somoano and F. Huera-Huarte, "The dead band in the performance of cross-flow turbines: Effects of Reynolds," *Energy Conversion and Management*, vol. 172, pp. 277-284, 2018.
- [52] R. S. Mirsane, M. Rahimi and F. Torabi, "Development of a novel analytical wake model behind HAWT by considering the nacelle effect," *Energy Conversion and Management*, vol. 301, 2024.
- [53] A. Rezaeiha, H. Montazeri and B. J. Blocken, "CFD analysis of dynamic stall on vertical axis wind turbines using Scale-Adaptive Simulation (SAS): Comparison against URANS and hybrid RANS/LES," *Energy Conversion and Management*, vol. 196, pp. 1282-1298, 2019.

- [54] Q. Li, J. Xu, Y. Kamada, M. Takao, S. Nishimura, G. Wu and C. Cai, "Experimental investigations of airfoil surface flow of a horizontal axis wind turbine with LDV measurements," *Energy*, vol. 191, 2020.
- [55] V. Kornilov, "Three-dimensional turbulent near-wall flows in streamwise corners: Current state and questions," *Progress in Aerospace Sciences*, vol. 94, pp. 46-81, 2017.
- [56] H. Pan, H. Li, T. Zhang, A. A. Laghari, Z. Zhang, Y. Yuan and B. Qian, "A portable renewable wind energy harvesting system integrated S-rotor and H-rotor for self-powered applications in high-speed railway tunnels," *Energy Conversion and Management*, vol. 196, pp. 56-68, 2019.
- [57] E. Fatahian, F. Ismail, M. H. H. Ishak and W. S. Chang, "An innovative deflector system for drag-type Savonius turbine using a rotating cylinder for performance improvement," *Energy Conversion and Management*, vol. 257, 2022.
- [58] N. K. Delany and N. E. Sorensen, "LOW-SPEED DRAG OF CYLINDERS OF VARIOUS SHAPES," NATIONAL ADVISORY COMMITTEE FOR AERONAUTICS (NACA), Washington, 1953.
- [59] S. Bengt, "Tubes, Crossflow over," *Thermopedia*, 2011.
- [60] H. C.-H. Ng, H. L. F. Cregan, J. M. Dodds, R. J. Poole and D. J. C. Dennis, "Partially filled pipes: experiments in laminar and turbulent flow," *Journal of Fluid Mechanics*, vol. 848, pp. 467 - 507, 2018.
- [61] R. Röhrig, S. Jakirlić and C. Tropea, "Comparative computational study of turbulent flow in a 90° pipe elbow," *International Journal of Heat and Fluid Flow*, vol. 55, pp. 120-131, 2015.
- [62] A. Kalpakli and R. Örlü, "Turbulent pipe flow downstream a 90° pipe bend with and without superimposed swirl," *International Journal of Heat and Fluid Flow*, vol. 41, pp. 103-111, 2013.
- [63] C. Mockett, M. Fuchs and F. Thiele, "Progress in DES for wall-modelled LES of complex internal flows," *Computers & Fluids*, vol. 65, pp. 44-45, 2012.
- [64] T. Lund, "The use of explicit filters in large eddy simulation," *Computers & Mathematics with Applications*, vol. 46, pp. 603-616, 2003.

- [65] R. Zangeneh, "Assessing IDDES-Based Wall-Modeled Large-Eddy Simulation (WMLES) for Separated Flows with Heat Transfer," *Fluids*, vol. 6, no. 7, p. 246, 2021.
- [66] W. J. RODI, "Status of large eddy simulation : results of a workshop," *J. of Fluids Engineering*, vol. 119, pp. 248-262, 1997.