6th European Conference on Computational Mechanics (ECCM 6)
7th European Conference on Computational Fluid Dynamics (ECFD 7)
11 – 15 June 2018, Glasgow, UK

EFFECTS OF TURBULENCE MODELLING ON THE PREDICTIONS OF THE PRESSURE DISTRIBUTION AROUND THE WING OF A SMALL SCALE VERTICAL AXIS WIND TURBINE

MOHAMED M. ELSAKKA[1,2], DEREK B. INGHAM[3], LIN MA[4], AND MOHAMED POURKASHANIAN[5]

[1] Energy2050, University of Sheffield
Sheffield, UK
mmmel-sakka1@sheffield.ac.uk

[2] Faculty of Engineering, Port Said University
Port Said, Egypt
elsakka@eng.psu.edu.eg

[3] Energy2050, University of Sheffield
Sheffield, UK
d.ingham@sheffield.ac.uk

[4] Energy2050, University of Sheffield
Sheffield, UK
lin.ma@sheffield.ac.uk

[5] Energy2050, University of Sheffield
Sheffield, UK
m.pourkashanian@sheffield.ac.uk

Key words: *VAWT, CFD, Turbulence model, Moderate Reynolds Number.*

Abstract. The aerodynamics of the flow around Vertical Axis Wind Turbines (VAWTs) has become a very important research topic. The combination of the blade rotational motion and the free stream wind leads to an oscillating Angle of Attack (AOA) around the turbine blade. The small-scale VAWTs typically operate at low and moderate Reynolds numbers and low Tip Speed Ratios (TSRs) and this leads to a higher amplitude in the oscillating AOA and complex aerodynamics that includes boundary layer flow transition, separation, and dynamic stall. This directly contributes to the pressure distribution around the blade and the flow turbulence plays an important role. The aim of this paper is to compare two different turbulence models in a 2D Computational Fluid Dynamics (CFD) prediction of the pressure distribution, and the overall turbine power generation of a small scale VAWT, and compare against the experimental data that is available in the literature. The turbine investigated is a small two-bladed VAWT operating at an average Reynolds number of approximately $2.5*10^5$. A commercial solver is used for the CFD calculations, while the average non-dimensional wall distance is maintained at about $y^+=1$ in order to resolve the details of the boundary layer. The SST transition and k-ω SST turbulence models are implemented and their results are compared to the experimental data taken at the mid-span of the turbine at different azimuthal locations. For the turbine investigated, the CFD results, based on the two turbulence models, in general show a good agreement with the experimental data and in particular in the upstream section of the turbine. The *k*-*ω* SST model gives reasonable predictions to the pressure distribution over the blade at almost all blade locations from low to high angles of attack. The agreement with experimental data is very good for the suction side of the blade. The SST transition model is able to capture the small laminar separation bubbles that occur near to the leading edge of the aerofoil and predict well the pressure distribution on the suction side when the boundary layer flow is attached. However, at high AOAs where stall occurs, the SST transition results appear to overestimate the flow separation, while the *k*-*ω* SST results fit better with the experimental data. In terms of the instantaneous power coefficients, both models can predict reasonably well the measurement data although the SST transition model appears to predict results which are slightly closer to the experimental data than the *k*-*ω* SST model when the blade is moving across the upwind side of the turbine where majority of the power is generated. It is expected that further investigations are needed on the prediction of flow at stalled flow conditions and the blade wake interactions in order to accurately predict the performance of a VAWT.

1 INTRODUCTION

In recent years, there has been a notable increase in the number of investigations on VAWTs and this has given the VAWT technology a new rebirth. While the Horizontal Axis Wind Turbines (HAWTs) have acquired the major share of the wind power market, the VAWT concept is estimated to play an increasing role in the next 2–3 decades [1]. In addition, the VAWTs feature potential advantages, especially for the urban environment and the offshore with floating platforms as it can receive wind from any direction [2]. However, VAWTs suffer from lower efficiencies in comparison with the HAWT design [3] due to its complex aerodynamics. The small-scale VAWTs typically operate at low and moderate Reynolds numbers in the order of 10^5 and low Tip Speed Ratios (TSRs). Furthermore, the combination of the blade rotational motion around its vertical axis and the free stream wind velocity leads to a continuous variation and oscillation of the Angle of Attack (AOA) around the turbine blade. This leads to a higher amplitude in the oscillating AOA and complex aerodynamics that includes boundary layer flow transition, separation, and dynamic stall and separation. This has a direct impact on the performance of the wind turbine.

The 2D CFD analysis, based on the Reynolds-averaged Navier–Stokes (RANS), is widely used in order to investigate the complex aerodynamics of the VAWTs because of its reasonable accuracy and moderate computational cost [4]. Previous research has shown that turbulence modelling is critical to the prediction of the aerodynamics and force generation of the turbine blade, in particular, with the occurrence of flow transition and separation. *k*-*ω* SST and SST transition are two popular turbulence models that are employed in the literature. The *k*-*ω* SST model has been developed to be more accurate in the prediction of the flow with separation and high adverse pressure gradients [5]. The correlation-based SST transition model has shown to give a good agreement with several experimental data from aerodynamic and turbomachinery tests [6]. Several studies have utilized the *k*-*ω* SST model [7]–[10] and SST transition [11]–

[13] in the 2D CFD predictions of the VAWT power curves that rely on the average power coefficient over the cycle. Since the accurate prediction of the pressure distribution over the turbine blade is very important which dictates the boundary layer flow and turbine power generation, the aim of this paper is to test the two models against detailed experimental data. For this purpose, the experimental data of Li et al. [1] has been selected to validate the current CFD simulations. Their experimental data includes the instantaneous power coefficient and the pressure distribution around the blade at several azimuthal angles. These data are considered at the blade mid-span and they are assumed to be relevant to the 2D simulations that neglect the 3D effect. The test case under consideration is for a two-bladed VAWT with a 1.7 m diameter. The turbine blades have a chord length of 0.225 m and a span of 1.02 m and with a NACA0015 profile while each blade is pitched out at 6° as shown in

Figure **1**. The turbine was tested at a wind velocity of 7 m/s and TSR of 2.29 with an opened circular type wind tunnel having a jet diameter of 3.6 m and the main test section of 6.2 m. The average Reynolds Number based on blade chord is approximately 2.5×10^5 and this is considered to be in the transitional range. Both the *k-ω* SST and SST transition turbulence models are selected to investigate the flow characteristics around the blade in the 2D CFD simulations.

2 COMPUTATIONAL MODEL

The two-dimensional version of ANSYS FLUENT with double precession is utilized for the simulation of the mid-plane of the VAWT. The coupled pressure-based algorithm is used for the velocity-pressure coupling. The second-order discretization scheme is used for the momentum and turbulence model equations while the second-order implicit formulation is used for the temporal discretization.

The computational domain and the associated boundary conditions are shown in Figure 2. The domain extends to five turbine-diameters in the lateral direction in order to eliminate the effect of the side boundaries on the turbine performance while it extends to eight turbine-diameters downstream of the rotational axis to resolve the turbine wake. The lateral sides of the domain are set as symmetrical boundary conditions while the downstream edge of the domain is modelled as a zero-pressure outlet. The domain is divided into several subdomains to account for the two blades, rotor, and stationary domain zones. The sliding mesh method is utilized to model the rotational motion of the turbine by means of a circular non-conformal mesh interface. The upstream edge of the domain is considered as a velocity inlet with a velocity of 7 m/s and turbulence intensity of 0.5% taken from the experimental data.

Dedicated mesh and time step independency studies are carried out and a mesh with 7×10^5 elements has been selected for the analysis with a temporal resolution of 540 time steps per cycle. Although the simulations include at least five complete cycles, only the results of the fifth cycle are considered for the post-processing in order to ensure that a time-periodic solution is obtained. Figure 3 shows the mesh topology of the computational domain that features a structure mesh over the entire domain. The mesh is refined around the blades to have an average $y^+\approx1$.

3 RESULTS AND DISCUSSIONS

The straight blade Darrieus VAWTs are mainly lift driven devices that rely on aerofoil-shaped blades to generate the driving forces. The aerodynamic force that acts on an immersed body can be classified into viscous forces and pressure forces. However, for the turbine blades the pressure forces are predominant. The lift force is mainly developed due to the pressure differences between the suction and pressure sides of the aerofoil-profiled blade. This makes the study of the pressure distribution around the VAWT blades very important. The rotational cycle of the turbine could be divided into upstream and downstream parts. Most of the power generation occurs when the blade is travelling in the upstream part of the cycle. It is noticed that the downstream part is associated with the complex incident flow structure due to the strong blade wake interactions. In the present study, the 2D CFD predictions of the pressure coefficient distributions around the blades, based on both the k-ω SST and SST transition turbulence models, are analysed at 12 different azimuthal locations with a 30° increment.

Figure 4 shows a comparison between the experimental data and the 2D CFD predictions of the pressure coefficients of the blade on the upstream part of the cycle, while Figure 5 shows the corresponding comparisons on the downstream part of the cycle. In general, the two models investigated show a good agreement with the experimental data, in particular at the upstream section of the turbine and when the flow is attached to the blade. The k-ω SST model gives reasonable predations of the pressure distribution over the blade at almost all blade locations from low to high angles of attack. It is observed that the agreement with experimental data is very good for the suction side of the blade. At a few azimuthal locations of the blade, the model slightly over predicted the magnitude of the pressure at the pressure side of the blade, and this occurred in the SST transition model as well. However, the SST transition model is able to capture the formation of the small Laminar Separation Bubbles (LSBs) that occurs near to the leading edge of the aerofoil at several azimuthal locations, including $\theta = 0°$, 120° , 210° , 240° , 300° , and 330°, although the predicted locations of the LSBs are slightly shifted towards the leading edge. The prediction of the LSBs indicates that the solver predicts the laminar separation and the turbulent reattachment in the near wall layers. Overall, the SST transition model predicted the pressure distribution well on the suction side when the boundary layer flow is attached. At certain azimuthal angles, e.g. $\theta = 150°$ and 240°, where the blade encounters high AOAs and stall most probably occurs, the SST transition model appears to overestimate the flow separation on the suction side of the blade, while the k-ω SST results fit better with the experimental data. Both the SST transition and k-ω SST predictions show good agreements with the experimental data on the pressure side of the turbine blade overall. Nevertheless, discrepancies between the model predictions and experimental data for the suction peak at the leading edge are observed, as widely reported in the literature.

Figure 6 shows a comparison between experimental data, based on the measured pressure distribution at the mid-span of the blade, and the 2D CFD prediction of the instantaneous power coefficient for a single blade. Both models can predict reasonably well the measurement data although the SST transition model results appear to be closer to the experimental data than the *k-ω* SST model when the blade is moving across the upwind side of the turbine where the majority of the power is generated. It may be noticed that the *k-ω* SST and SST transition results are almost identical in the azimuthal angle range between θ= 0° and 90° and agrees well with the experimental data. The over prediction of the power around θ= 90° is mostly due to the discrepancies in the pressure on the pressure side and the leading edge of the blade. Noticeable differences between the two model predictions between θ= 90° and 150° are observed. This is the region where the turbine blade encounters high AOAs and the blade stalls and the SST transition model predicted larger flow separation regions than the *k-ω* SST model, thus resulting in a different pressure distribution over the suction side of the blade. In the range between θ= 180° and 360°, the blade travels in the downstream part of the turbine into the wake sheded from the blade from the upstream cycle and severe blade-wake interactions are expected It is very difficult to model the flow in this region accurately. It may be noticed that the maximum deviation of the SST transition results occurs at θ= 270° where the blade passes across the wake of the turbine shaft.

4 CONCLUSIONS

The 2D CFD predictions of the detailed pressure and power coefficients, based on both the *k-ω* SST and SST transition turbulence models, have been investigated. The analysis, based on the SST transition model, is able to capture the formation of the LSBs and predicts the pressure distributions well when the flow is attached. The *k-ω* SST model, on the other hand, predicted the pressure distribution more closely to the measurement at the stalled condition, as well as for the attached flow. The predicted pressure coefficients show good agreements overall with the experimental data for both the SST transition and *k-ω* SST models on the pressure side of the turbine blade. However, the predictions of the SST transition model overestimates the separation on the suction side at certain azimuthal angles. The predicted pressure coefficients, based on the *k-ω* SST model, appears to have a consistent trend. The 2D CFD predictions of the instantaneous power coefficients, based on both the turbulence models, show a fair agreement with the experimental data. It is expected that further investigations are needed for the prediction of flow at stalled flow conditions and the blade wake interactions in order to accurately predict the performance of a VAWT.

5 FUTURE WORK

Turbulence is itself a 3D process. The comparison between the 2D and 3D behaviours of the different turbulence models on CFD simulations of VAWTs is considered as a potential future work.

ACKNOWLEDGEMENT

Mohamed M. Elsakka would like to express his gratitude to the Egyptian Cultural Affairs and Missions Sector along with Port Said University for their financial support.

REFERENCES

[1] M. R. Islam, S. Mekhilef, and R. Saidur, “Progress and recent trends of wind energy technology,” *Renewable and Sustainable Energy Reviews*, vol. 21, pp. 456–468, May 2013.

[2] D. Marten, A. Bianchini, G. Pechlivanoglou, F. Balduzzi, C. N. Nayeri, G. Ferrara, C. O. Paschereit, and L. Ferrari, “Effects of Airfoil’s Polar Data in the Stall Region on the Estimation of Darrieus Wind Turbine Performance,” *Journal of Engineering for Gas Turbines and Power*, vol. 139, no. 2, pp. 1–9, Sep. 2016.

[3] W. Tjiu, T. Marnoto, S. Mat, M. H. Ruslan, and K. Sopian, “Darrieus vertical axis wind turbine for power generation II: Challenges in HAWT and the opportunity of multi-megawatt Darrieus VAWT development,” *Renewable Energy*, vol. 75, pp. 560–571, Mar. 2015.

[4] A. Bianchini, F. Balduzzi, P. Bachant, G. Ferrara, and L. Ferrari, “Effectiveness of two-dimensional CFD simulations for Darrieus VAWTs: a combined numerical and experimental assessment,” *Energy Conversion and Management*, vol. 136, pp. 318–328, Mar. 2017.

[5] F. R. Menter, M. Kuntz, and R. Langtry, “Ten Years of Industrial Experience with the SST Turbulence Model,” *Turbulence Heat and Mass Transfer 4*, vol. 4, pp. 625–632, 2003.

[6] F. R. Menter, R. B. Langtry, S. R. Likki, Y. B. Suzen, P. G. Huang, and S. Völker, “A Correlation-Based Transition Model Using Local Variables—Part I: Model Formulation,” *Journal of Turbomachinery*, vol. 128, no. 3, p. 413, 2006.

[7] C. Xisto, J. Páscoa, and M. Trancossi, “Numerical analysis of design parameters with strong influence on the aerodynamic efficiency of a small-scale self-pitch VAWT,” in *Volume 1: Advances in Aerospace Technology*, 2015, pp. 1–11.

[8] R. Firdaus, “Numerical and experimental studies of a small vertical-axis wind turbine with variable-pitch straight blades,” Dissertation, Graduate School of Natural Science & Technology, Kanazawa University, 2015.

[9] M. Yi, Q. Jianjun, and L. Yan, “Airfoil design for vertical axis wind turbine operating at variable tip speed ratios,” *The Open Mechanical Engineering Journal*, vol. 9, pp. 1007–1016, 2015.

[10] L. Zhang, Y. Pei, Y. Liang, F. Zhang, Y. Wang, J. Meng, and H. Wang, “Design and implementation of straight-bladed vertical axis wind turbine with collective pitch control,” *Mechatronics and Automation (ICMA), 2015 IEEE International Conference on*, pp. 1258–1263, 2015.

[11] R. Lanzafame, S. Mauro, and M. Messina, “2D CFD modeling of H-Darrieus wind turbines using a transition turbulence model,” *Energy Procedia*, vol. 45, pp. 131–140, 2014.

[12] B. Paillard, J. A. Astolfi, and F. Hauville, “URANSE simulation of an active variable-pitch cross-flow Darrieus tidal turbine: Sinusoidal pitch function investigation,” *International Journal of Marine Energy*, vol. 11, pp. 9–26, 2015.

[13] H. F. Lam and H. Y. Peng, "Study of wake characteristics of a vertical axis wind turbine by two- and three-dimensional computational fluid dynamics simulations," *Renewable Energy*, vol. 90, pp. 386–398, 2016.

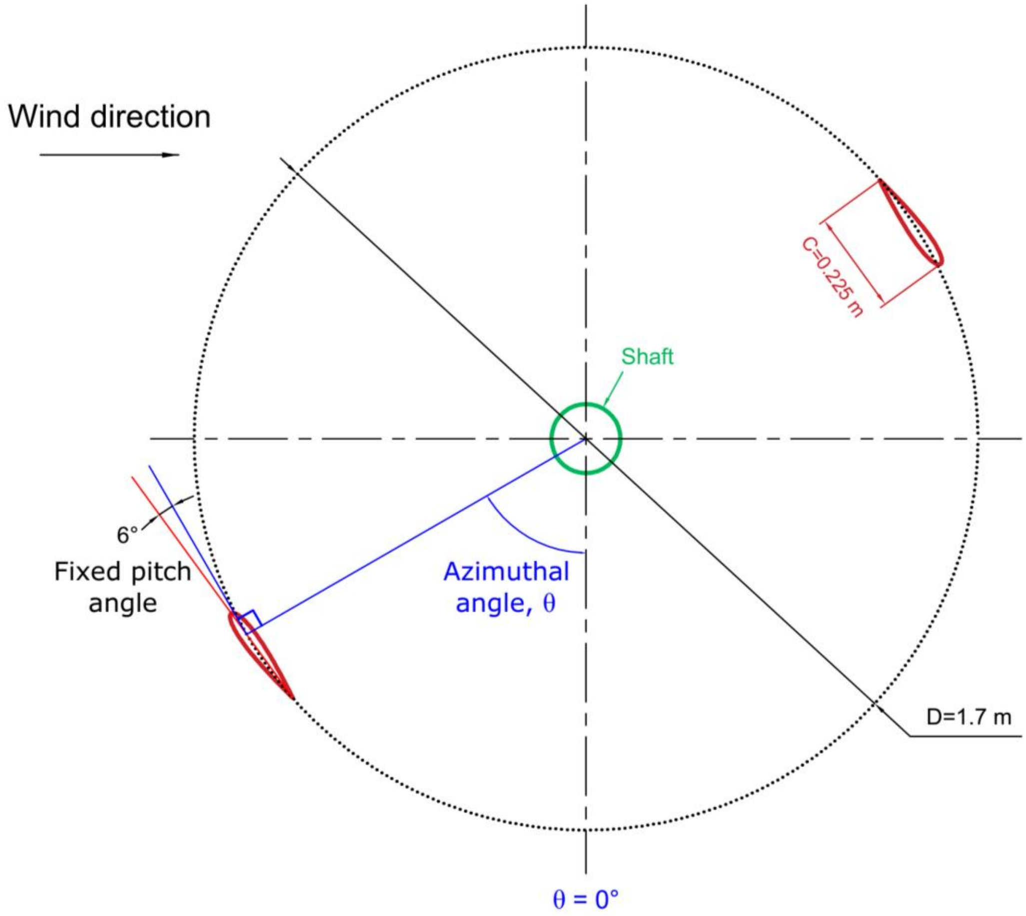


Figure 1 A schematic diagram of the turbine at arbitrary azimuthal position.

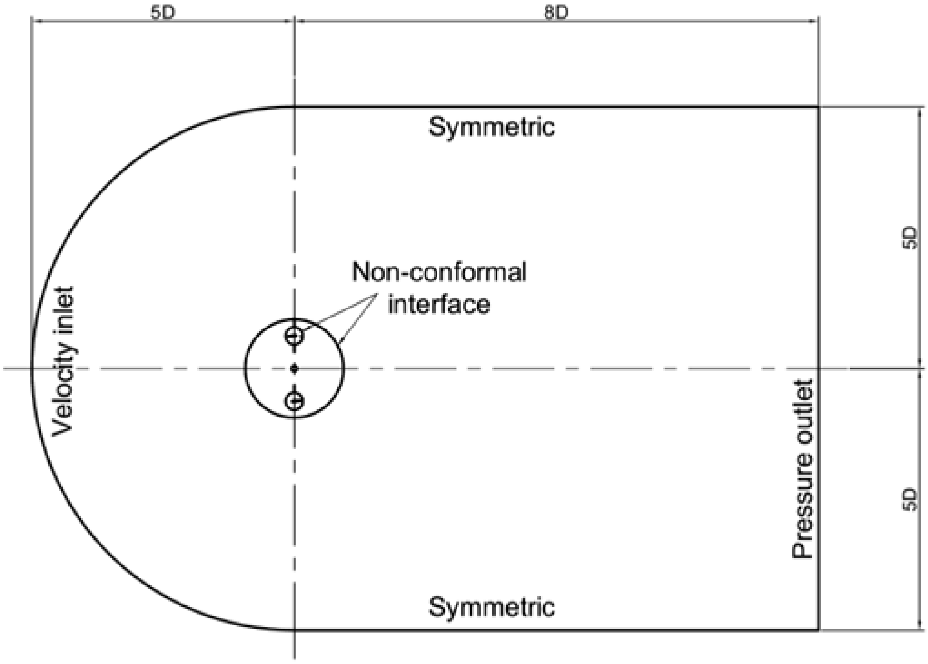


Figure 2 The mesh topology of the computational domain.

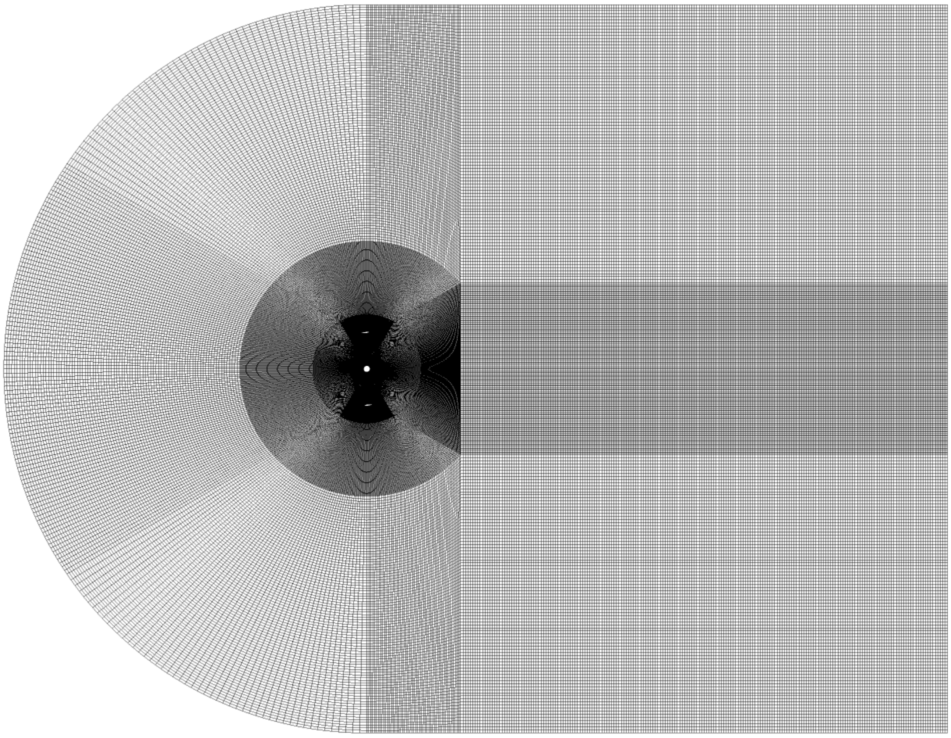
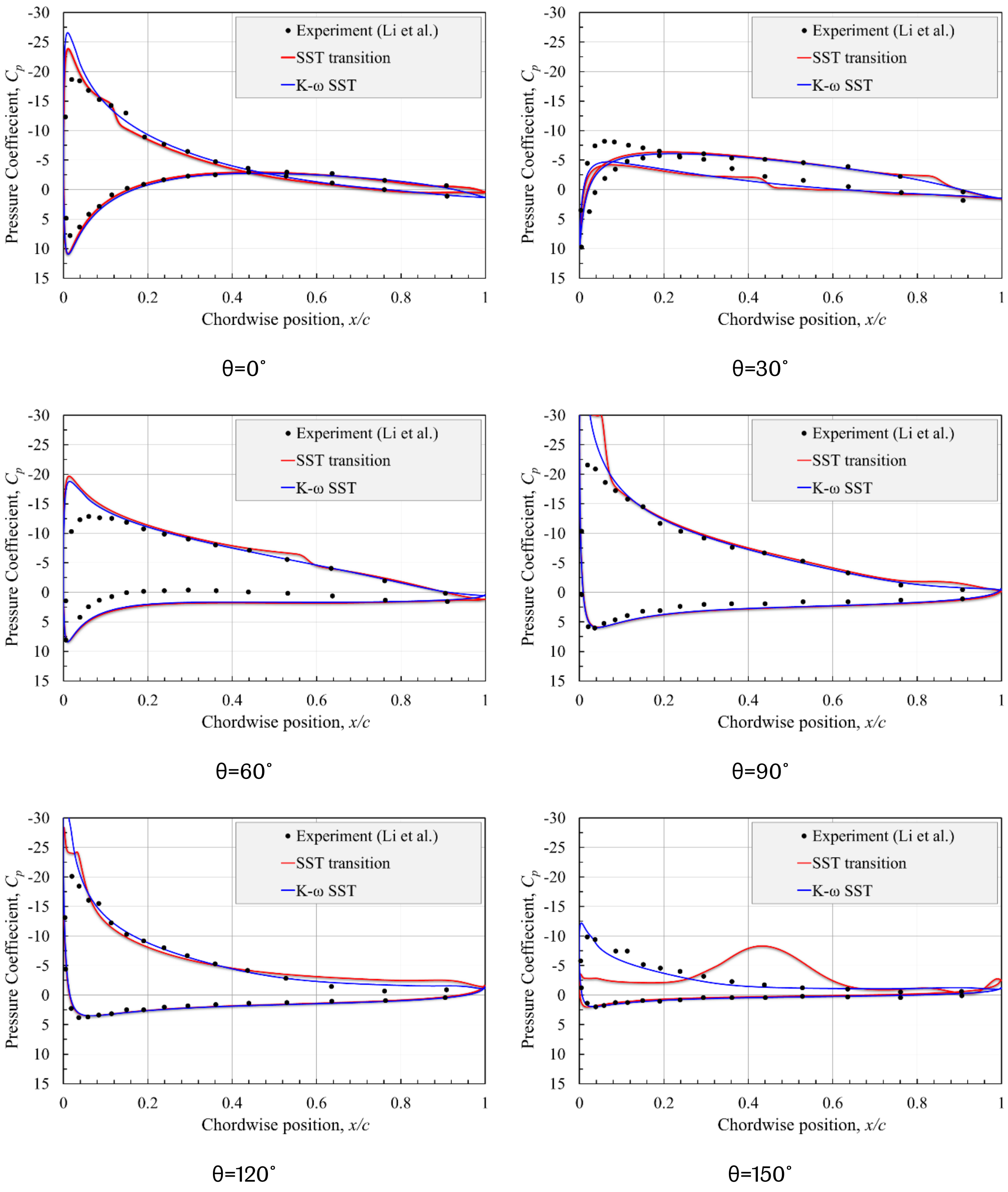


Figure 3 The mesh topology of the computational domain.



θ=0°

θ=30°

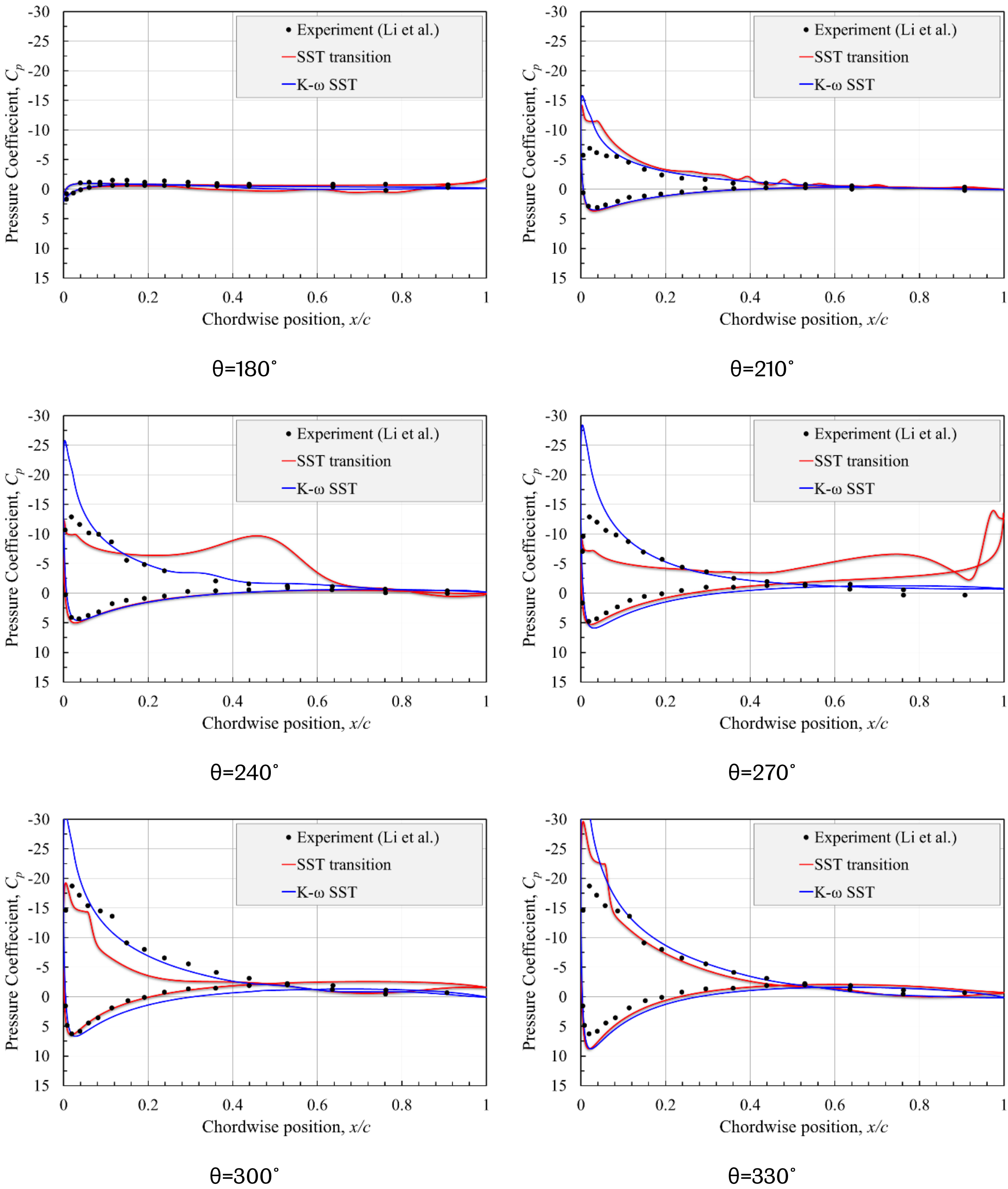
θ=60°

θ=90°

θ=120°

θ=150°

Figure 4 The pressure coefficient around the blade at different azimuthal angles (on the upstream part of the cycle).



θ=180°　θ=210°

θ=240°　θ=270°

θ=300°　θ=330°

Figure 5 The pressure coefficient around the blade at different azimuthal angles (on the downstream part of the cycle).

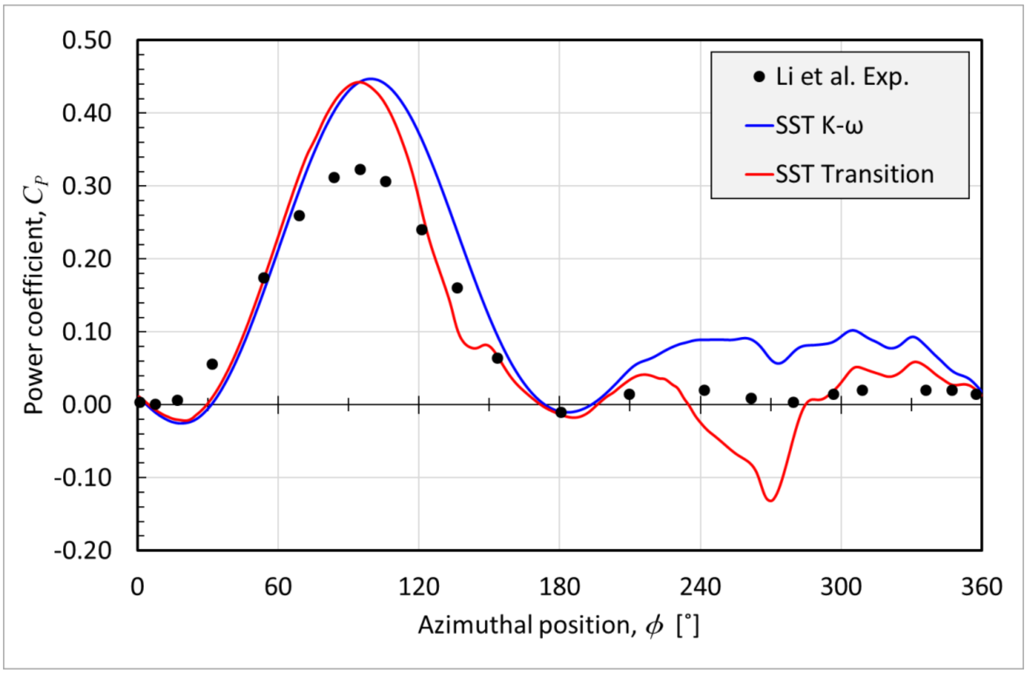


Figure 6 A comparison between the experimental and the numerical single blade instantaneous power coefficient over the fifth cycle.