Investigations of the inflow turbulence effect on rotational augmentation by means of CFD

Galih Bangga, Yusik Kim, Thorsten Lutz, Pascal Weiheing and Ewald Krämer
Institute of Aerodynamics and Gas Dynamics (IAG), University of Stuttgart, Pfaffenwaldring 21 Stuttgart 70569, Germany
E-mail: bangga@iag.uni-stuttgart.de

Abstract. The present studies are addressed to gain more insights into the inflow turbulence effect on rotational augmentation using computational fluid dynamics. Three different cases were simulated and analysed focusing on the three-dimensional effects in the inboard blade region of a 10 MW generic wind turbine rotor. The evidence of rotational augmentation was presented and compared to two-dimensional simulations of the blade sections at consistent inflow conditions. Inflow turbulence has a very strong impact on the instantaneous blade loads and standard deviations, but the effect on the mean values is small. The amplitudes of the blade load fluctuations are amplified under turbulent inflow conditions and these are related to the blade passing frequency and the specified turbulence length scale at the inlet. Detailed examinations of these phenomena were performed and are presented in the present manuscript.

1. Introduction
Three-dimensional (3D) effects or rotational augmentation at the inboard sections of horizontal axis wind turbines (HAWT) have been widely studied and reported in literature. This phenomenon can be defined as the lift increase of a rotating blade section compared to 2D airfoil at consistent inflow conditions. The effects were firstly described by Himmelskamp [1] for propeller aerodynamics who found that the lift coefficient of the rotating blade section was substantially higher than of the airfoil in the 2D conditions. Later measurements by Milborrow and Ross [2], Ronsten [3], and Bruining et al. [4] confirmed this observation. Savino and Nyland [5] made use of balanced wind vanes to evaluate the flow direction on the surface of a full-scale rotor. It was observed that the chordwise flow was occurring upstream of the separation line while a strong radial flow was observed within the separated flow area.

The Coriolis and centrifugal forces were claimed as the reason for such phenomena. The centrifugal force transported the separated flow near the root, creating a strong radial flow component and reducing the boundary layer thickness [6, 7], called centrifugal pumping. The Coriolis force was believed to act in chordwise direction, delaying the occurrence of separation. Later, it was shown by Du and Selig [8] that the centrifugal pumping effect was not as strong as it had been thought before; instead, the Coriolis force had a stronger contribution in separation delay.

The design of wind turbine blades typically relies on simple aerodynamic models based on the blade-element momentum theory (BEM). The methods were well known to give reasonable results near the turbine design point but under-predict the forces in stalled conditions [9].
3D corrections for the engineering models were then developed in HAWT calculations. Snel et al. [10], Du and Selig [11], Lindenburg [12] and Chaviaropoulos and Hansen [13] proposed semi-empirical models which were mainly based on the ratio of the blade sectional chord length to radius \((c/r)\). Later, Bak et al. [14] gave a good overview of these models and introduced a new correction model based on the surface pressure coefficient. Despite many efforts spent for correcting the loads of wind turbines, to the authors' knowledge, there are no available 3D correction models incorporating the inflow turbulence level at the moment.

The turbulence level has been shown to have a strong influence on the aerodynamic behavior of 2D airfoils (see e.g., [15, 16]), enhancing the maximum lift coefficient and postponing the occurrence of stall. Sicot et al. [17] demonstrated that the study of wind turbine blade aerodynamics needs to take into account the coupled effects of rotation and turbulence [17]. Increasing the turbulence level tends to delay flow separation on the rotating blade as well as increase its chordwise pressure gradient [17]. Understanding this issue is important because wind turbines are exposed to turbulence and subjected to rotational motion during operation.

On this basis, the present work aims to gain a deeper insight into the impact of inflow turbulence on rotational augmentation by evaluating CFD results of a wind turbine rotor. The rotational effects are studied by comparing the results of the 3D rotating blade with the 2D simulations at consistent inflow conditions. With the presence of turbulence, the sectional forces are examined and compared to laminar inflow conditions. Finally, the load fluctuations and turbulence intensity in the vicinity of separated flow are presented.

2. Test Cases and Numerical Setup

The study was conducted on the generic 10 MW AVATAR (Advance Tools for Large Rotor) wind turbine blade [18]. The computational study was performed for the tip speed ratio \(TSR = 9.07\). This corresponds to the wind speed \(U_{\text{inf}} = 10.5\, m/s\) and rotational speed \(n = 9.02\, \text{rpm}\). The designated rated wind speed for this turbine is \(U_{\text{inf}} = 10.75\, m/s\). The pitch angle was set to zero. The blade was developed based on the DTU 10 MW reference wind turbine from the INNWIND.EU project with a lower induction factor along the blade radius [18].

Three different cases were simulated. For Case 1, flow periodicity from one blade to the other blade is assumed. By doing so, only one blade was modelled in the simulation. Laminar inflow condition at the inlet was considered in Case 1. This setup was used to demonstrate the effect of rotation on the blade. 2D simulations at consistent inflow conditions, i.e., at the same angle of attack (AoA), Reynolds (Re) and Mach numbers (Ma), were carried out for comparison. The other cases (Case 2 and Case 3) incorporated the full 3D rotor blades without tower. The inflow condition for Case 2 is laminar and for Case 3 is turbulent. A uniform inflow without shear was imposed at the inlet for all cases. The summary of the simulated cases is presented in Table I.

The CFD code FLOWer [19] from the German Aerospace Center was utilized. The code is continuously developed in the institute for the application in wind turbine aerodynamics [20,21]. The cell-centered based finite volume solver was used on a block-structured grid, generated

| 2D Case | Case 1 | Case 2 | Case 3 |
|---------|--------|--------|--------|
| 2D / 120° / Full (F) Model | 2D | 120° | F | F |
| Laminar (L) / Turbulent (T) Inflow | L | L | L | T |
| Turb. Model | URANS | URANS | IDDES | IDDES |
using the Gridgen™ and Pointwise™ grid generators, consisting of four grid components (blade; spinner + nacelle; connector; and background grids). The meshes were built separately independent of each other. The communication between the independent grid meshes was made possible by the Chimera (overset) method which interpolates the information from one grid to the other grid(s) on the overlapping region, Figure 1. The blade mesh (Figure 2a) was C-H type topology and consisted of 280 x 128 x 192 cells in chordwise, normal and spanwise directions, respectively. Spatial discretization studies according to Celik et al. [22] have been conducted in [23], showing the independency of the predicted sectional loads towards increasing blade cells number. For Cases 2 and 3, the spinner was different than in Case 1, and no nacelle was applied in simulations, Figure 2b and 2b.

Temporal discretization studies have been done for Case 1. 3° blade rotation per physical timestep was found to sufficiently predict the mean forces along the blade radius. At least 45 inner iterations were applied every physical timestep, resulting in the drop of density residual to $10^{-6}$. A smaller timestep, 1°, was used for Case 2 and Case 3 as it was the requirement of the AVATAR project for the turbulence inflow case. However, the loads and sectional forces are not sensitive within the time steps used ($\leq 3°$) as long as a sufficient number of inner iterations were applied. The time integration was carried out by an explicit hybrid multi-stage Runge-Kutta scheme. Dual time-stepping and third level multigrid were used.

In Case 1 and in 2D case, the (URANS) SST $k-\omega$ turbulence model with fully turbulent boundary layer was employed. For Cases 2 and 3, the IDDES turbulence model [24] based on the same RANS model as for Case 1 was used to capture resolved turbulence in the upstream region and its impact on flow characteristics in the inner sections of the rotor. Inflow turbulence was synthesized using the Mann model [25], and it was imposed via momentum source term [26] on a 2D plane at 10 radii upstream distance of the rotor. Mesh convergence studies for background turbulence have been conducted for the current study. A relatively mild turbulence intensity, TI = 3.88%, was applied as a first case, and higher levels of turbulence will be applied within the AVATAR project Task 2.4 coordinated by the University of Stuttgart. The turbulence length scale as an input for the Mann box was set to $L = 89.53m$. 

Figure 1: Grid setup showing blade (pink); spinner and nacelle (green); connector (red); and background grids (blue).
3. CFD Results

The objective of the performed simulations was to receive information on how the wind turbine loads are effected by 3D effects and turbulence inflow. Therefore, the CFD simulations of a turbine isolating the 3D and turbulence inflow effects have been analyzed.

3.1. 3D Effects

The rotation is expected to influence the blade loads especially in the inboard region of the blade. Figure 3a shows distribution of the lift coefficient ($C_l$) from 3D simulations (blue line) compared to 2D simulations along the blade radius at consistent inflow conditions. The angles of attack ($AoA$s) of the 3D rotating rotor were extracted using the reduced axial velocity method proposed by Johansen and Sørensen [27].

It can be seen that the 3D effect, i.e., rotational augmentation, has a strong influence in the inboard blade region up to $r/R < 0.4$. The lift coefficient of the 3D case shows remarkably higher value compared to the 2D airfoil simulations at the same inflow conditions ($AoA, Re$ and $Ma$). It is observed that 2D lift shows negative value at certain radial stations reaching minimum at
Figure 3: (a) $C_l$ distribution along the blade radius. The 3D sectional lift coefficient is higher than in the 2D conditions, especially in the blade root region. (b) Limiting streamlines on the blade surface, a strong radial flow is observed in the blade root due to centrifugal pumping.

Figure 4: Chordwise velocity profiles comparison at two radial stations, non-dimensionalized by the local inflow velocity. Wall distance is non-dimensionalized by the chord length. Solid lines: 3D results, dashed lines: 2D results. The 3D effects enhance the velocity in streamwise direction adjacent to the blade surface.

Massive flow separation was observed on the suction and pressure sides of this airfoil due to excessive airfoil thickness ($t/c \approx 52\%$). The decambering effect is expected to cause this phenomenon since the positive angle of attack is observed. On the other hand, separation is delayed in 3D conditions due to the centrifugal pumping and the Coriolis acceleration within the blade boundary layer.

The rotational augmentation occurs because the flow is massively separated in the blade inboard region. The centrifugal pumping transports this low momentum flow towards the outer blade region so that a strong radial flow is observed in figure 3b. The trend of the 3D $C_l$ curve is similar with the two-dimensional results implying that the lift augmentation characteristic is strongly dependent on the airfoil behaviour in the two-dimensional viscous flow state. For example, the local 2D lift increase/decrease at $0.15R/0.25R$ in the 2D case is observed also in
the 3D case.

Being consistent with the preceding studies [1–4, 6–9, 11–14, 17, 23, 27], the rotational augmentation becomes smaller with increasing radial distance from the center of rotation. Figure 3a shows that the predicted 2D $C_l$ is comparable with the 3D case in the blade middle sections. On the other hand, in the blade outer section, the 3D lift coefficient is lower than the 2D $C_l$ in the vicinity of the blade tip (Figure 3a) which may be caused by the three dimensional flow around the tip. Smaller 3D lift coefficient around the tip indicates that the induction is smaller; thus, the wind speed is higher in and downstream of the rotor plane. This leads to increasing angle of attack close to the tip.

In figure 4, chordwise velocity profiles at two radial positions are depicted. The 3D velocity profiles have fuller shapes compared to the 2D case in the proximity of the hub region (at $0.15R$). In consequence, smaller deficits of the flow momentum can be observed compared to the 2D case and the occurrence of separation can be delayed significantly to high AoA. There is a certain chordwise area where the chordwise velocity is positive in the thin near wall layer beneath separated flow and that is exactly in the domain where the Coriolis force as well as radial flow is strongest. This effect alleviates at a larger radius and only a small enhancement of the flow momentum is observed ($r/R=0.6$).

To better understand the Coriolis force acting on the blade, the ratio between the Coriolis to the centrifugal forces was calculated using equation (1) and the results in chordwise direction are plotted in Figure 5.

$$\frac{\vec{F}_{\text{Coriolis}}}{\vec{F}_{\text{centrifugal}}} = \frac{-2(\vec{\Omega} \times \vec{V})}{\vec{\Omega} \times (\vec{\Omega} \times \vec{r})}.$$  

(1)

It can be seen that the strength of the Coriolis force is decreasing with increasing radial distance. Strongest influence is observed at $r/R = 0.2$. The blade suction side shows more pronounced Coriolis force than the blade pressure side. However, on the very thick airfoils near the blade root, the blade pressure side has a high positive pressure gradient so that severe flow separation is also occurring. Thus, the radial flow also develops resulting in the remarkable Coriolis force on the pressure side of the blade at $r/R = 0.2$. Overall, the Coriolis force is larger than the local centrifugal force close to the point of separation.

![Figure 5](image_url)

Figure 5: The ratio between the Coriolis to centrifugal forces acting on the blade sections. The Coriolis acceleration is decreasing with increasing blade radius.
3.2. Turbulence Inflow Effects

This section evaluates the CFD results for laminar (Case 2) and turbulence inflow (Case 3) cases. In Figure 6a, it is demonstrated that the mean values of sectional forces, normal ($F_n$) and tangential ($F_t$) to the rotor plane, are hardly influenced by the inflow turbulence though the forces are slightly enhanced. For examples, the mean value of the tangential force at radial station of $0.2R$ is $0.366 \, kN/m$ for the laminar case and $0.379 \, kN/m$ for the turbulent case, while it is $0.684 \, kN/m$ (laminar) and $0.696 \, kN/m$ (turbulent) for the radial station of $0.8R$. The higher flow momentum for Case 3 causes this small sectional force increase along the blade radius. The momentum increase is observed in streamwise and radial directions, Figure 6b and 6c, especially close to the wall so that a slightly stronger Coriolis force is expected. For

![Image](image1)

Figure 6: (a) Sectional forces of the laminar and turbulence inflow cases. The impact of turbulence on mean loads is weak, but the mean forces are slightly enhanced. (b) Streamwise and (c) radial velocities at $r/R = 0.15$, non-dimensionalized by the local inflow velocity. Wall distance is non-dimensionalized by the chord length. Solid lines: turbulence inflow, dashed lines: laminar inflow. The flow momentum in chordwise and radial directions is increased by the inflow turbulence effect.

![Image](image2)

Figure 7: Tangential force history at three radial stations, non-dimensionalized by the mean value. It should be noted that the scale of the pictures is different. The amplitude of the force is significantly amplified in the case with the turbulence inflow.
low turbulence level, it can be seen that the impact of the inflow turbulence on the averaged sectional force is little. Nevertheless, higher turbulence levels may show more pronounced effects. Furthermore, to better evaluate the impact on rotational augmentation, comparison of the load at the same AoA is required. However, the velocity averaging method [27] is not suitable for strong unsteady flow condition like inflow turbulence. The extraction method developed by Shen et al. [28] is useful in predicting the angle of attack for unsteady flows because no spatial averaging is performed, but to the best of our knowledge this model is not yet validated for inflow turbulence case. On this basis, no angle of attack extraction is performed for Case 2 and Case 3 and only mean value is compared.

From Figure 6a, it can be depicted that turbulence promotes stronger load fluctuations than in the laminar inflow case. It can be seen that the tangential force has a large contribution on this fluctuation and it is decreasing with radius. To get a better view on it, the tangential force history over the azimuth angle is plotted in Figure 7a and 7b for the laminar and turbulence inflow cases, respectively. From both figures, it is clear that the blade inboard section produces a stronger load oscillation with higher relative amplitude. Background turbulence seems to influence this behaviour by enhancing the amplitude of the tangential load oscillation along the whole blade radius, compared to the laminar case which does not show any sign of load oscillation.

Figure 8: Turbulence intensity field near the blade sections for laminar and turbulent cases. Distinct differences are observed near the leading edge and within the separated flow area.
in the outer blade stations. This is expected because the main cause of load oscillation for the laminar inflow case is flow separation. Since the flow over the blade outer region is attached, no strong oscillation is observed. In contrast, the load oscillation in the turbulence inflow case comes from the incoming velocity itself, therefore it results in the load fluctuation with high amplitude even though the flow is attached. Combined with the separated flow near the root, the turbulence effect creates much stronger oscillations than in the laminar inflow case. For example, at the radial position of 0.15\( R \) the relative amplitude of \( F_1 \) in Case 3 is 10 times higher than in Case 2.

Figure 8 demonstrates this phenomenon. The turbulence intensity contours are plotted for the laminar and turbulent cases at two radial positions. Distinct differences can be seen on the leading edge region. The turbulence intensity is very high due to inflow turbulence in the both observed radial stations. This may lead to unsteady effects such as dynamic stall [29–31] as the blade sections see continuous changes of the wind velocity due to turbulence, i.e., the angle of attack also changes because the rotational speed is constant. The aerodynamic loads fluctuate with a distinctive frequency, see Figure 9. A narrow frequency range corresponds with not only the blade rotation frequency, \( F_0 = 0.15Hz \), but also the incoming unsteady turbulence signals with the frequency of \( U_{inf}/L = 0.12Hz \). This frequency signal occurs only within a short period and at a specific point, i.e., at a specific radial position. Longer time signals analyses are required to obtain more broadband spectra for the specific point which are computationally too expensive for nowadays CFD. Spatial averaging of the frequency spectra are commonly performed to obtain such spectra. Figure 9c shows the response spectra of the total tangential force from blade 1. It is shown that the spectra are much more broadband compared to the frequency spectra of a specific point, depicted in Figures 9a and 9b, because they contain several frequency responses from different radial positions. In Figure 9c, the amplitude of the tangential force is also increased by the background turbulence, becoming more pronounced with decreasing frequency.

Apart from that, the turbulence also shows a strong influence within the separated flow region demonstrated by increasing turbulence intensity in Figure 8, but in a smaller order of magnitude compared to the turbulence intensity increase around the leading edge. For a specific point at a short time period (Figures 9a and 9b), flow field fluctuation within separated area is characterized
by the high frequency signals and the fluctuation around the leading edge corresponds to the lower frequency around $1F_0$. It can be seen that the relative amplitude of $F_t$ and $F_n$ at $1F_0$ for the turbulent inflow case is much higher than for the laminar inflow at the inboard and outboard blade sections. Meanwhile, they are similar for the high frequency peak ($3F_0$).

4. Conclusion

CFD studies on the impact of inflow turbulence on rotational augmentation have been performed. Three different cases were simulated for the same tip speed ratio. The first case was simulated using the URANS turbulence model to demonstrate the rotational augmentation of the studied blade. The two other cases were pure-rotor and were simulated with and without turbulence inflow using IDDES with the same turbulence model as in the first case. The turbulence intensity as high as 3.88% was generated using the Mann turbulence box and it was imposed to the domain via forcing term.

From the first case, the three-dimensional (3D) effects are observed. The 3D lift coefficient ($C_l$) in the blade inboard sections is substantially higher than the two-dimensional (2D) $C_l$. The Coriolis force accelerates the flow in chordwise direction and its strength increases with decreasing blade radius. The Coriolis force becomes stronger than the centrifugal force near the separation point where the flow starts to detach from the blade surface in the blade inboard region.

The turbulence has a weak influence on the sectional loads increase which is originated by the stronger flow momentum in the turbulent case, especially in streamwise and radial directions. The inflow turbulence strongly influences the instantaneous value of sectional loads which is mainly caused by the fluctuation of the wind speed, increasing the turbulence intensity near the leading edge for the whole blade. The load amplitude is amplified significantly at a frequency close to the blade passing frequency ($F_0$) and is expected to be determined by the specified turbulence length scale at the inlet.

The present study is limited only for the flow at mild turbulence level. Higher turbulence levels may result in the more pronounced impact. Deeper evaluations of the background turbulence impact on rotational augmentation are necessary. Examinations of the resulting loads need to consider the sectional-instantaneous, not only the averaged, angle of attack to obtain a clear insight into the physical phenomena. This will be carried out in subsequent studies. In addition, because the blade loads fluctuate periodically over the blade revolutions, investigations incorporating the higher turbulence levels and the unsteady effects are encouraged.

Acknowledgements

The authors gratefully acknowledge these following institutions for the supports: Ministry of Research, Technology and Higher Education of Indonesia for the funding through Directorate General of Higher Education (DGHE) scholarship, the AVATAR project for a good cooperation by providing the blade geometry and test cases necessary for the study, the High Performance Computing Center Stuttgart (HLRS) and Leibniz Supercomputing Center (LRZ) for providing computational time in the CFD simulations.

References

[1] Himmelskamp H 1945 Profile investigations on a rotating airscrew Ph.D. thesis Universität Göttingen
[2] Milborrow D and Ross J 1984 Airfoils characteristics of rotating blades IEA LS-WECS, 12th Meeting of Experts, Copenhagen
[3] Ronsten G 1992 Static pressure measurements on a rotating and a non-rotating 2.375 m wind turbine blade. Comparison with 2D calculations J. Wind Eng. Ind. Aerodyn. 39 105–118
[4] Bruining A, Van Bussel G, Corten G and Timmer W 1993 Pressure distribution from a wind turbine blade; field measurements compared to 2-dimensional wind tunnel data Tech. rep.
[5] Savino J and Nyland T 1985 Wind turbine flow visualisation studies Tech. rep. NASA Lewis Research Center
[6] Carcangiu C E, Sørensen J N, Cambuli F and Mandas N 2007 CFD-RANS analysis of the rotational effects on the boundary layer of wind turbine blades J. Phys.: Conf. Series 75 012031
[7] Ramos-García N, Sørensen J N and Shen W Z 2014 A strong viscous-inviscid interaction model for rotating airfoils Wind Energy 17 1957–1984
[8] Du Z and Selig M 2000 The effect of rotation on the boundary layer of a wind turbine blade Renewable Energy 20 167–181
[9] Shen W Z and Sørensen J N 1999 Quasi-3D Navier-Stokes model for a rotating airfoil Journal of Computational Physics 150 518–548
[10] Snel H, Houwink R, Bosschers J, Piers W J, van Bussel G J W and Bruining A 1993 Sectional prediction of 3-D effects for stalled flow on rotating blades and comparison with measurements EWEC 3959
[11] Du Z and Selig M S 1998 A 3-D stall-delay model for horizontal axis wind turbine performance prediction AIAA paper 21
[12] Lindenburg C 2003 Investigation into rotor blade aerodynamics Tech. rep. ECN Report: ECN-C-03-025
[13] Chaviaropoulos P and Hansen M O 2000 Investigating three-dimensional and rotational effects on wind turbine blades by means of a quasi-3D Navier-Stokes solver Journal of Fluids Engineering 122 330–336
[14] Bak C, Johansen J and Andersen P B 2006 Three-dimensional corrections of airfoil characteristics based on pressure distributions Proceedings of the European Wind Energy Conference pp 1–10
[15] Devinan P, Laverne T and Hureau J 2002 Experimental study of wind-turbine airfoil aerodynamics in high turbulence J. Wind Eng. Ind. Aerodyn. 90 689–707
[16] Schneemann J, Knebel P, Milan P and Peinke J 2010 Lift measurements in unsteady flow conditions Proceedings of EWEC 2010
[17] Sicot C, Devinan P, Loyer S and Hureau J 2008 Rotational and turbulence effects on a wind turbine blade. Investigation of the stall mechanisms J. Wind Eng. Ind. Aerodyn. 96 1320–1331
[18] Lekou D et al. 2015 Avatar deliverable d1.2 reference blade design Tech. rep. ECN Wind Energy
[19] Kroll N, Rossow C C, Becker K and Thiele F 2000 The MEGAFLOW project Aerospace Science and Technology 4 223–237
[20] Meister K, Lutz T and Krämer E 2014 Simulation of a 5MW wind turbine in an atmospheric boundary layer J. Phys.: Conf. Series vol 555 p 012071
[21] Bangga G, Lutz T and Krämer E Numerical investigation of unsteady aerodynamic effects on thick flatback airfoils German Wind Energy Conference 12, Bremen
[22] Celik I B, Ghia U, Roache P J et al. 2008 Procedure for estimation and reporting of uncertainty due to discretization in CFD applications Journal of Fluids Engineering 130
[23] Bangga G, Lutz T and Krämer E 2015 An examination of rotational effects on large wind turbine blades EAWE PhD Seminar 11, Stuttgart
[24] Shur M L, Spalart P R, Strelets M K and Travin A K 2008 A hybrid RANS-LES approach with delayed-DES and wall-modelled LES capabilities Int. J. Heat Fluid Flow 29 1638–1649
[25] Mann J 1994 The spatial structure of neutral atmospheric surface-layer turbulence Journal of Fluid Mechanics 273 141–168
[26] Troldborg N, Sørensen J N, Mikkelsen R and Sørensen N N 2014 A simple atmospheric boundary layer model applied to large eddy simulations of wind turbine wakes Wind Energy 17 657–669
[27] Johansen J and Sørensen N N 2004 Aerfoil characteristics from 3D CFD rotor computations Wind Energy 7 283–294
[28] Shen W Z, Hansen M O and Sørensen J N 2009 Determination of the angle of attack on rotor blades Wind Energy 12 91–98
[29] Huyer S A, Simms D and Robinson M C 1996 Unsteady aerodynamics associated with a horizontal-axis wind turbine AIAA journal 34 1410–1419
[30] Hutomo G, Bangga G and Sasongko H 2016 CFD studies of the dynamic stall characteristics on a rotating airfoil Applied Mechanics and Materials 836 109–114
[31] Larsen J W, Nielsen S R and Krenk S 2007 Dynamic stall model for wind turbine airfoils Journal of Fluids and Structures 23 959–982