Escola de Primavera de Transição e Turbulência Blumenau 2022

ABCM
13th Spring School on Transition and Turbulence
September 19th-23rd, 2022, Blumenau, SC, Brazil

EPTT-2022-0008
Assessment of Turbulence Models for the Simulation of the Flow Through a Megawatt Scale Wind Turbine Rotor

Marielle de Oliveira
marielledeoliveira@usp.br
Rodolfo C. Puraca
rodolfo.puraca@usp.br
Bruno S. Carmo
bruno.carmo@usp.br
Department of Mechanical Engineering, Escola Politécnica - University of São Paulo - Brazil

Abstract. *The increase of the size of wind turbines to deliver power at megawatt scale, particularly for offshore application, brings a number of engineering challenges. The numerical modeling of these systems, considering the wind turbine geometry in full scale, is a valuable tool for design and performance analysis. To properly model the interaction between the turbine and the wind we need a proper turbulence model. This paper presents a comparison of two of the most used turbulence models, the Unsteady Reynolds Averaged Navier-Stokes (URANS) k-ω SST and the two-equation Detached Eddy Simulation (DES) applied in blade-resolved simulations of the NREL 5 MW reference wind turbine, in order to predict the rotor performance when it operates in optimal wind-power conversion efficiency, for a wind speed of 10 m/s at hub height. The power production, generated thrust, and forces distribution along the blade span were estimated. The computational analyses were carried out using a Computational Fluid Dynamics (CFD) methodology employing the Finite Volume Method (FVM) implemented in the OpenFOAM software. A numerical verification was conducted by comparing the CFD results against values obtained using the blade element momentum theory, implemented in OpenFAST. The performance of each turbulence model was assessed considering the computational cost and accuracy of the results. Both turbulence models presented satisfactory results when comparing with the results from OpenFAST, for the same environmental condition investigated. However the wake internal gradient present different patterns. For the DES model it was possible to observe with higher resolution the effects of the blade in the near wake region. In addition, a different behavior of the flow that detaches along the blade span and transitions to the wake external pattern was also observed when comparing the turbulence models.*

Keywords: *NREL 5 MW Wind Turbine Rotor, Blade-resolved Simulations, Iterative PISO Solver, Turbulence Models, Computational Cost Analysis*

1. INTRODUCTION

With the continued expansion of the wind energy industry over the past decade, which aims to contribute significantly to the global energy transition, besides the increase of wind power in emerging markets such as China, India, and Brazil, the growth of wind turbines operating in offshore installations was also noticed (Council, 2022; El Bassam, 2021). Consequently, due to the development of wind energy in offshore areas, which can present outstanding wind resources (Ostachowicz *et al.*, 2016; Karimirad, 2014), the changes in the size of wind turbines to higher scales present intrinsic challenges.

The offshore sites present environmental conditions such as atmospheric boundary layer and turbulence varying spatially, which directly affects the prediction of the aerodynamic loads and wake behaviour of the wind turbines. Therefore, along with the arising of the new generation of wind turbines which include higher costs associated in both fabrication and installation process, the need of better tools to accurately predict the loads acting in the offshore wind turbines (OWT) become an important task.

Up to date, the experimental campaigns which were conducted to obtain information about the unsteady three-dimensional aerodynamic behaviour of horizontal-axis wind turbine (HAWT), such as presented by Hand *et al.* (2001a,b), have shown that the aerodynamic loads and that 3D effects are prevalent, resulting in a complex system to be accurately represented experimentally. Even though the data have been used to validate and enhance engineering models, due to the ever increasing power capacity of OWTs to scales of 3 MW, 6 MW (Hayes *et al.*, 2021), and more recently DTU 10 MW, GE's Haliade-X 14 MW, and IEA 15 MW (Bak *et al.*, 2013; Jiang, 2021; Gaertner *et al.*, 2020), the development of high fidelity numerical models capable of capturing the influence of these three-dimensional effects to better predict the OWT performance is necessary as a reliable tool in the OWT design (Hand *et al.*, 2001b; Zhang *et al.*, 2019). Among the numerical options to deal with these effects, computational fluid dynamics (CFD) has been applied through different methods, and has shown to be a mature approach to investigate the unsteady aerodynamic behaviour of the flow around wind turbine blades and generated wakes (Sanderse *et al.*, 2011; Thé and Yu, 2017).

The numerical investigations available in the literature which considered a CFD approach to investigate the flow around a wind turbine rotor blade with a blade-resolved modeling (Sorensen and Hansen, 1998; Duque *et al.*, 1999, 2003; Zhang *et al.*, 2019) showed that the numerical solution of the Navier-Stokes (N-S) equations, which needs special treatment to proper represent the turbulence effects, still requires efforts since the solution through the direct numerical simulations (DNS) to analyse the wind turbine performance is yet not feasible.

A different approach is given by the large-eddy simulation (LES) method, in which the equations are solved taking into account a filtered velocity field, so the larger scales of the turbulent motion are represented, whereas the smaller scales of the turbulent motions, also called as subgrid-scales (SGS), are modeled (Pope, 2001; Wilcox *et al.*, 1998). As part of the turbulence modeling technique considered in the numerical investigations of engineering problems, the Reynolds-Average Navier-Stokes (RANS) procedure is vastly applied, due to the solution of the Reynolds equations which determines the mean velocity field (Pope, 2001). In sequence, the Unsteady Reynolds-Averaged Navier-Stokes (URANS) nomenclature started to be used, since the RANS models are unsteady even when considering steady boundary conditions (Spalart, 2000). In addition, as a result of this mathematical procedure, the Reynolds stress tensor requires the use of turbulence models to be evaluated (Wilcox *et al.*, 1998; Pope, 2001).

Among the ample variety of turbulence models, to represent the aerodynamic loads under the influence of considerably adverse pressure gradient, the URANS approach is commonly linked to the two-equation k-ω SST turbulence model (Menter, 1992, 1993, 1994). To date, the URANS k-ω SST turbulence model has been used in the modeling of wind turbines conducted in the OpenFOAM software for a small-scale HAWT and presented good agreement in terms of the wind turbine performance coefficient between the CFD results and the calibrated experimental tests (Rocha *et al.*, 2014).

More qualitative and quantitative agreement between the results from the CFD modeling of a similar problem, considering the same turbulence modeling and experimental tests, were found also in the prediction of the velocity profiles in the wake region in the MEXICO project (Sørensen *et al.*, 2014). More recently, the URANS k-ω SST turbulence model was successfully applied to represent the turbulence effects in the numerical modeling of the NREL 5 MW wind turbine in full scale, including the tower influence, to represent the flow around the blades and in the wake region. Since there was no experimental data available, the authors conducted a verification procedure benchmarking the blade-resolved results against the results obtained with the OpenFAST software for the same environmental conditions, and presented good agreement in terms of the power production, generated thrust and distributed forces along the blade span (de Oliveira *et al.*, 2022).

Usually, the preference for URANS-based models instead of LES or DNS approaches is related to the computational costs which is largely determined by the resolution requirements (Pope, 2001). Even though LES is advantageous when comparing to URANS in the modeling of anisotropic turbulent flow, in which large-scale structures are dominant, in the numerical analysis of the flow around wind turbines, the model is recommended to be applied only in the wake region (Sanderse *et al.*, 2011), due to the fact that the LES approach present difficulties to determine the flow properties in the wall region of the boundary layer (Wilcox *et al.*, 1998; Spalart, 2000).

In this regard, since LES even when implemented with a proper wall-region modeling is not viable to predict unsteady aerodynamic loads (Spalart, 1997), the hybridization of LES into a improved approach such as the Detached-Eddy Simulation (DES) model allow the numerical modeling of the turbulence effects to a manageable computational demand even for flows at high Reynolds numbers (Shur *et al.*, 1999; Nikitin *et al.*, 2000; Spalart, 1997).

Whereas the application of DES in aerodynamics is promising due to the possibility of applying the URANS-based models in the large areas of the boundary layer while in the regions in which the momentum transfer is dominated by large structures LES is efficiently applied (Spalart, 2000), few investigations have implemented the DES approach in the blade-resolved CFD simulations of a megawatt scale wind turbine, due the difficulties of proper connecting the numerical model setup with the more suitable spatial and temporal discretization. For example, in the investigations performed by Lawson *et al.* (2019), the authors used DES to represent the flow around the NREL 5 MW wind turbine blades and wake region. However, the quasi-steady regime for the power and thrust, which typically requires around 5-6 rotor revolutions to be established, was not achieved even for the coarsest mesh tested, due to the mesh strategy employed, which presented a high computational demand, indicating that more efforts were still required to better comprehend the efficiency of the method when comparing the results accuracy and computational costs.

With the increase in the wind turbines scale such as 15 MW, the numerical modeling in full scale becomes even more challenging, and the need of a optimized turbulent model to represent the unsteady aerodynamics load more evident. Therefore, in order to cover these needs, the target of this paper is to present a comparison between the URANS k-ω SST and DES-two equations turbulence models applied in the blade-resolved CFD simulations to adequately predict the aerodynamic loads of the baseline NREL 5 MW wind turbine rotor in full scale (Jonkman *et al.*, 2009), under the operating condition of optimal wind-power conversion efficiency.

Since these simulations are computationally expensive and challenging to set up the turbulence model properly, in the present investigation we are not taking into account the tower influence, nevertheless we hope the discussion and results presented in the next sections regarding the rotor-only investigation, considering different turbulence models, can be used to help in the modeling and simulation of other large wind turbines.

2. METHODOLOGY

To conduct the numerical investigations and capture the effects of the different turbulence models tested, we considered the same numerical discretization schemes and also the same spatial and temporal discretization strategies for each case. The investigations were carried out considering the open source OpenFOAM software and the performance of the wind turbine rotor-only simulations was evaluated in terms of power production, generated thrust, distributed forces along the blade span and wind profile in different positions in the wake region. As a verification procedure, the CFD results were benchmarked against the results obtained with OpenFAST (National Renewable Energy Laboratory, 2021), which implements the blade element momentum method, considering the same NREL 5 MW wind turbine rotor-only, at the same environmental conditions. Finally a computational cost analysis was conducted between the two different turbulence models to allow us to understand the performance of each one considering the numerical arrangement chosen.

In this section we describe the governing equations considered to model the problem and also some details about the solver parameters and discretizations schemes employed in the CFD investigation.

2.1 Governing Equations

As the problem being investigated is represented by a transient three-dimensional incompressible flow, the governing set of equations is given respectively by the conservation of mass and conservation of momentum equations, as:

$$\nabla \cdot \mathbf{U} = 0, \tag{1}$$

$$\frac{\partial \mathbf{U}}{\partial t} + \nabla \cdot (\mathbf{U}\mathbf{U}) = -\nabla p + \nabla \cdot (\nu \nabla \mathbf{U}) + \mathbf{f}, \tag{2}$$

where t is time, ν is the kinematic viscosity, **U** is the velocity vector, p is the kinematic pressure and **f** represents the body forces.

In this work we discretize the governing equation given by Eqs. 1 and 2 considering the finite volume method (FVM) in which detailed information can be found in Versteeg and Malalasekera (2007); Patankar (2018). In this regard, the discretization of the non-linear term given by the convective term in Eq. (2) leads to

$$\nabla \cdot (\mathbf{U}\mathbf{U}) = \sum_f \mathbf{S}(\mathbf{U}_f)(\mathbf{U}_f) = \sum_f F(\mathbf{U}_f), \tag{3}$$

$$\nabla \cdot (\mathbf{U}\mathbf{U}) = a_P \mathbf{U}_P + \sum_N a_N \mathbf{U}_N, \tag{4}$$

where the coefficients a_P and a_N are related to the values interpolated at the faces of the control volume P and its neighbors indicated by N, and are functions of the velocity **U**. $\mathbf{U}_f$ is the control volume velocity at the face cell, and **S** is the area vector pointing out of the volume cell with magnitude equal to the face area, while F represents the term $\mathbf{S} \cdot (\rho U)_f$, which is the mass flux through a general face.

Due to the complexity of the non-linear solvers and consequent computation effort required, a linearisation of the convective term is recommended. Considering the type of flow being investigated, the calculation of the F term is performed using a numerical upwind-based method, to guarantee the boundedness of the solution by preserving positive coefficients in the linear algebraic equation matrices (Jasak, 1996; Versteeg and Malalasekera, 2007).

The discretized form of the continuity equation, Eq. (1), is given by

$$\nabla \cdot \mathbf{U} = \sum_f \mathbf{S}\mathbf{U}_f = 0. \tag{5}$$

More details about the step-by-step in the discretization process by the FVM can be found in de Oliveira *et al.* (2022); Versteeg and Malalasekera (2007). Therefore, the discretized form of the incompressible Navier-Stokes equations are given by

$$a_P \mathbf{U}_P = \mathbf{H}(\mathbf{U}) - \sum_f \mathbf{S}(p)_f, \tag{6}$$

$$\sum_f \mathbf{S}\Big[\Big(\frac{1}{a_P}\Big)_f (\nabla p)_f\Big] = \sum_f \mathbf{S}\Big(\frac{\mathbf{H}(\mathbf{U})}{a_P}\Big)_f, \tag{7}$$

with the calculation of the face fluxes F given by

$$F = \mathbf{S}\mathbf{U}_f = \mathbf{S}\Big[\Big(\frac{\mathbf{H}(\mathbf{U})}{a_P}\Big)_f - \Big(\frac{1}{a_P}\Big)_f (\nabla p)_f\Big]. \tag{8}$$

2.2 Turbulence Modeling

To complete the numerical arrangement to conduct the blade-resolved CFD simulations, an additional set of transport equations to represent the turbulence are required in order to obtain an approximate solution for the Navier-Stokes set of equations. According to Wilcox *et al.* (1998), an ideal turbulence model should minimize the complexity of the flow field in order to capture the features of the most significant part of the physical system. As aforementioned mentioned in the section 1, the main objective of our work is to compare different turbulence modeling approaches in the blade-resolved simulation of a megawatt scale wind turbine rotor, focusing on the estimate of the rotor performance through the prediction of the power production, generated thrust, distributed forces along the blades, wind profile at different positions of the wake region and the comparison of the flow structures obtained by each approach.

Even though the URANS method along with the two-equations k-ω SST turbulence model is the most common and vastly used method to represent the physics of similar investigation (Rocha *et al.*, 2014; Sorensen and Hansen, 1998; Robertson *et al.*, 2015; de Oliveira *et al.*, 2022), with the increase in the wind turbine scale to cover the offshore application needs, the DES model has been put forward as a promising solution since it improves the accuracy of the results prediction, with less computational cost than LES (Zhang *et al.*, 2015).

2.2.1 URANS k-ω SST Approach

The URANS approach comes from a statistical averaging procedure applied to the Navier-Stokes equations, from which the nonlinear Reynolds stresses tensor term and consequent closure problem arise requiring the turbulence models to establish a sufficient number of equations to solve all the flow properties (Wilcox *et al.*, 1998; Pope, 2001).

One way to obtain a solution for the Reynolds stresses tensor in means of known quantities is using the mean velocity gradient. In this regard, the most popular approach is to use the turbulent-viscosity hypothesis, introduced by Boussinesq (Pope, 1975) which, according to the hypothesis, the deviatoric part of the Reynolds stress tensor is proportional to the mean rate of strain (Pope, 2001).

Therefore, the relation between the turbulent tensor and the turbulent viscosity, also called as eddy viscosity, imposes the idea that the transfer of momentum by diffusion in molecular level is similar to the transfer of momentum in a turbulent flow due to the turbulent fluctuations. The evaluation of the kinematic eddy viscosity, ν_t, can be made by solving the transport equations with the use of auxiliary relations. However, the most common method to obtain the kinematic eddy viscosity is considering a function which correlates it with the specific turbulent kinetic energy and its specific dissipation rate, such approach stands out for the so called two-equations turbulent model (Wilcox *et al.*, 1998; Pope, 2001).

In this work for the URANS turbulence modeling approach we employed the two-equations k-ω SST model from Menter (1994), due its ability of predict flows with strong adverse pressure gradient with higher performance when compared to the variation of the k-ω models from Wilcox *et al.* (1998), and the baseline from Menter (1993). Thus, by considering the k-ω SST model, a new set of governing equations is obtained. In the flow region close to the rotor walls, low-Reynolds corrections are applied due to the near-wall region treatment.

2.2.2 DES k-ω SST Approach

According to Spalart (2000), the limitations of the RANS approach to represent the physics in the boundary layer region relies in the outer region, where LES captures well the straining, cross-flow and curvature effects, although with a considerable computational demand over RANS. Therefore, by thanking into account the idea of entrust the RANS approach in the attached eddies region of the boundary layer, which refers to the region close to the walls, while LES is applied in the separated region, also called as the detached eddies region, the hybridization of the LES and RANS models brings out the DES hybrid approach (Spalart, 1997). In addition, DES is an attractive solution for external flows, since the model is simple to be implemented, and preset stability for both URANS turbulence models, such as, one and two-equations (Robertson *et al.*, 2015). However, the main challenge in the use of DES model is regarding the user skills in the determination of a suitable mesh resolution (Spalart, 1997).

Therefore, based on the results obtained by the authors in previous investigations about the blade-resolved simulations of the NREL 5 MW wind turbine, including the tower interference (de Oliveira *et al.*, 2022), we designed a strategical spatial discretization, in order to capture the influence of the hybrid model over the RANS model in terms of the results accuracy and less computational demand, to be able to perform blade-resolved simulations of a megawatt scale wind turbine rotor with a more robust turbulence model. Thus, we employed for DES the same turbulence model used for the URANS approach model, the k-ω SST turbulence model, which is also the reason the approach is called as DES 2-Equations.

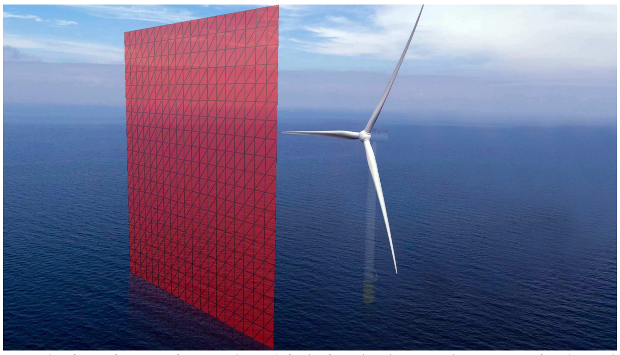


Figure 1: Visualization of the case being investigated, which includes a 5 MW wind turbine rotor in full scale, (without the tower an nacelle parts,) operating under a uniform non-turbulent wind profile.

2.3 Near-wall region treatment

As well mentioned in the previous section, the features of the turbulent flow changes considerably by getting closer to the wall region, in which an appropriated model is required for the estimation of the kinematic eddy viscosity ν_t. Thus, a strategical mesh refinement must be considered to estimate the turbulent flow close to the wall, which must satisfy the requirement of the turbulence model based on the y^+ variable. In the current investigations, for both turbulence models being investigated, k and ω were modeled by the low Reynolds wall functions, which represents a model which can switch between the viscous and logarithmic regions of the boundary layer according to the position of y^+, and also avoid the buffer layer region at the same time.

Following the same methodology, the kinematic eddy viscosity ν_t was calculated using the Spalding wall function model (Spalding, 1961), which can also switch between viscous and logarithmic regions based on the value of y^+.

3. NUMERICAL SIMULATIONS

In this section the setup and parameters considered in each numerical investigations are presented. The main objective of the CFD simulations were to predict the performance of the NREL baseline 5 MW offshore wind turbine rotor in full scale, in terms of power production, generated thrust, blade loading and wake aerodynamics pattern analysis considering two different turbulence models. The rotor geometry in full scale is composed by three blades and the hub, more detailed information regarding the rotor design are available in Jonkman *et al.* (2009).

Fig. 1 illustrates the case being investigated, which consists of a 5 MW wind turbine rotor for offshore application (without the tower interference), placed on an site of operation, under the influence of a uniform non-turbulent wind profile.

The simplifications which were made in the case being modeled, such as, the consideration of a uniform wind profile and the absence of the tower and nacelle parts in the wind turbine geometry, are typical of rotor-only CFD investigations (Duque *et al.*, 1999, 2003; Zhang *et al.*, 2019).

3.1 Computational Domain and Boundary Conditions

The rotor and hub geometries were built using the software Solid Edge and imported into OpenFOAM, whereas all other parts of the computational domain were built around the rotor geometry using the snappyHexMesh utility. Fig. 2 illustrates the computational domain dimensions in meters and the boundary conditions, which were defined based in Hsu and Bazilevs (2012); de Oliveira *et al.* (2022).

For both turbulence model investigation, the dimensions of the computational domain were the same, $480\,\mathrm{m}$ wide, $640\,\mathrm{m}$ long, $480\,\mathrm{m}$ high, and the rotor region of $160\,\mathrm{m}$ to settle the rotor diameter which is considered as $124\,\mathrm{m}$ to take into account the hub distance between the blades. The same boundary conditions were also considered in the turbulence models investigation for both cases, in which for the inflow the boundary condition for the velocity was of Dirichlet type, given by a prescribed uniform wind profile of 10 m/s, which was chosen based on Jonkman *et al.* (2009). According to the authors, at this wind speed the rotor operates in optimal wind-power conversion efficiency. Still at the inflow, the boundary condition for the pressure was a null gradient (Neumann condition). For the turbulent quantities, Dirichlet conditions were employed, with prescribed values estimated based on the most critical Reynolds number (at the blade tip), through the turbulence Reynolds number parameter (Re_L) as suggested by Pope (2001). Based on that, the turbulence length scale for this region was 0.175 m, the turbulence kinetic energy $k = 3.2651\,\mathrm{m^2s^{-2}}$, and the dissipation rate, $\omega = 20.5649\,\mathrm{s^{-1}}$, while the kinematic eddy viscosity was calculated based on the internal field everywhere. As the wind profile is uniform for all the side walls, the boundary conditions for the velocity were symmetric plane condition, which corresponds to null normal velocity and zero normal gradient for the tangential velocity, pressure and turbulent quantities. For the rotor walls,

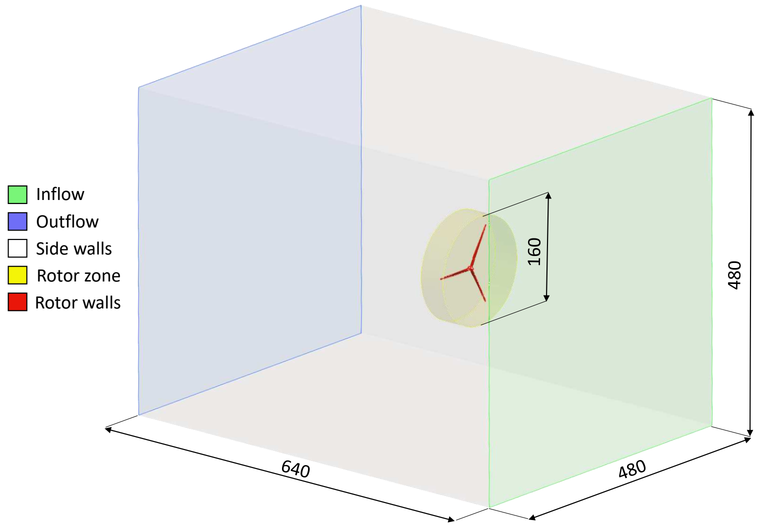


Figure 2: NREL baseline 5 MW offshore wind turbine rotor, computational domain dimensions (in meters) and boundary conditions.

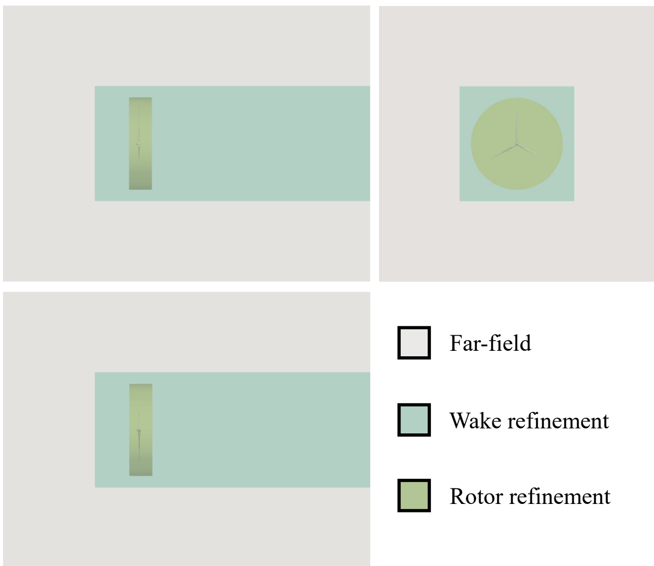


Figure 3: Strategic partition of the computational domain to apply different mesh refinements.

no slip condition was imposed. Since the mesh around this region is dynamic, a uniform rotor velocity of 1.1649 rad/s was prescribed, which is the rotor speed for a wind speed of 10 m/s, and Neumann boundary condition is applied for the pressure as a null gradient, while the turbulence properties receive the proper wall function treatment according with the y^+ value in the near wall region. At the outflow, Dirichlet condition was applied for the pressure as a fixed value equal to zero, and for the velocity and turbulence quantities had Neumann condition as null gradient.

3.2 Spatial discretization

To perform the turbulence model investigation, for both blade-resolved simulations the same spatial discretization were considered. The mesh was built considering a strategy similar to the one which was employed by de Oliveira *et al.* (2022) in Mesh-2, since it was adequate to present both accuracy in the predicted results at a accessible computational cost. One of the most important steps during the mesh design is related with the division of the the computational domain in which the different stages of refinement are applied.

In this regard, for the mesh being used in the present investigations, the computational domain was decomposed in three main regions as presented in Fig. 3, where in the far-field region the finite volume cells are of the size of $25\,\mathrm{m}$ which decreases into $1.6\,\mathrm{m}$ in the wake region and to $0.5\,\mathrm{m}$ in the rotor region. In the rotor region the cells size are refined from $0.5\,\mathrm{m}$ to $0.0625\,\mathrm{m}$ close to the blades and into $0.001\,\mathrm{m}$ at the first cell attached to the blades wall in order to respect the y^+ parameter within the adequate range for the application of the turbulence model at the near-wall region.

Regarding the mesh communication between the static and dynamic parts of the mesh, an arbitrary mesh interface (AMI) methodology was considered based in (Farrell and Maddison, 2011). Therefore, the spatial discretization strategy applied resulted in a mesh composed by $25, 314, 125$ finite volume cells, with a maximum aspect ratio of 75, skewness of 3.9 and non-orthogonality of 64.4.

3.3 Numerical schemes

The same numerical arrangement regarding the discretization schemes were employed for both turbulence models. The divergence terms were discretized using a second-order upwind scheme, chosen based on the modeling of similar problems to compute the convective fluxes. Central differences were employed for the Laplacian terms, and the second order Gauss scheme was adopted with linear Gaussian integration for the gradient terms. The set of linear equations was solved based on Muratova *et al.* (2020); Moukalled *et al.* (2016), using the geometric-algebraic multi-grid (GAMG) algorithm for the symmetric matrices, and the preconditioned bi-conjugate gradient (PBiCG) with the DILU preconditioner for the non-symmetric matrices.

Regarding the temporal discretization, the second order implicit backward scheme was employed, along with the limited CFL number equal to 1, which was controlled by an adaptive time step to guarantee stability during the iterative process.

3.4 Solver information

The iterative PISO with face flux correction, as presented in (de Oliveira *et al.*, 2022), was chosen as solver, in which 5 sub-iterations and 2 corrections for pressure was performed in each time step for both turbulence models being considered. For both DES and URANS turbulence models, the solution was considered converged when the residuals of the set of estimated variables was equal or less than 10^{-6} based in the convergence parameters for transient problems suggest by Versteeg and Malalasekera (2007). For each case, the initial conditions for the transient problem for all properties were obtained considering the steady state solution for the problem after 500 iterations, calculated with the steady form of the SIMPLE algorithm solver. The computations were carried out in the Brazilian supercomputer SDumont. To run the simulations, in each case the mesh was partitioned into 240 sub-domains using scotch decomposition, using 10 nodes, where each node had two 12 core Intel Xeon Cascade Lake Gold 6252 processors, 3.7 GHz, and 256 Gb of RAM.

4. RESULTS AND DISCUSSION

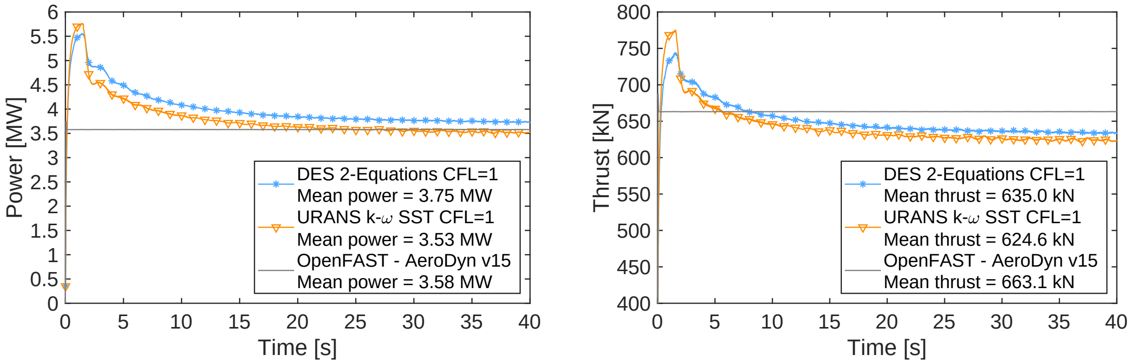
First, the results obtained by the two different turbulence models are presented in terms of power production and generated thrust. Next, a comparison between the computations of the distributed normal and tangential forces are presented. In sequence, the flow pattern and the vortical structures captured by the URANS and DES are illustrated along with the analysis of the wind velocity profile at different positions of the wake region. Finally, a comparison in terms of computational cost is shown in order to understand the performance of each turbulence model considered in the blade-resolved simulations to predict the unsteady aerodynamic loads of a megawatt scale wind turbine rotor.

4.1 Verification with OpenFAST

In order to understand the accuracy in the obtained results by the CFD simulations, the verification and validation are the main recommended methods to quantify the errors and uncertainties (Versteeg and Malalasekera, 2007). However, in our case there is no experimental data available, so we performed a verification procedure by making a comparison between the results obtained with each turbulence model against the results obtained with a different numerical method for the same environmental conditions, which was the OpenFAST v2.5.0 code by NREL (National Renewable Energy Laboratory, 2021). OpenFAST is a code certified by Germanischer Lloyd (GL) National Renewable Energy Laboratory (2005), and calibrated by Coulling *et al.* (2013), which uses the blade element momentum theory and tip corrections to calculate the aerodynamic loads of three-bladed HAWT, including different environmental conditions in the time domain.

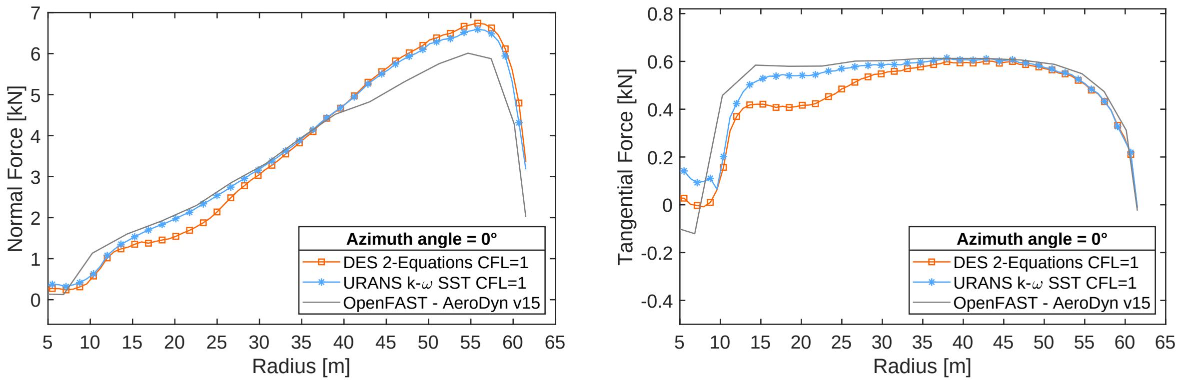
Figure 4 illustrates a comparison of the wind turbine power production and the generated thrust for both turbulence models tested, the DES 2-Equations, and the URANS $k-\omega$ SST. Both turbulence models presented similar results in terms of integral power and thrust. The mean value for the power was of 3.75 MW by DES and 3.53 MW by the URANS model, while for the thrust the mean value was of 635 kN, 624.6 kN for both DES and URANS respectively. This similarity between in results by boh turbulence models were also obtained by (Mittal *et al.*, 2016) for a comparable CFD simulation, where the author investigated the same turbulence models for a reduced scale wind turbine. Probably the similarity in the performance results is due to the modeling of the region close to the blades wall, since both turbulence models solve the same equations.

In addition, the distribution of the mean forces along the blades were also investigated and compared. As presented in Fig. 5, both methodologies presented similar behaviour in the normal and tangential forces prediction along the blade span, for the positions of the blade being analyzed. The $0°$ azimuth angle represents the blade aligned in the z-direction



(a) Power. (b) Thrust.

Figure 4: Generated power and thrust comparison between the DES 2-Equations and URANS $k-\omega$ SST turbulence models, benchmarked against OpenFAST results.



(a) Normal Force. (b) Tangential Force.

Figure 5: Distributed forces along the blade span comparison between the DES 2-Equations and URANS $k-\omega$ SST turbulence models, benchmarked against OpenFAST results.

orthogonal with the flow in the x-direction. The agreement between the results is remarkable.

Besides the quantitative analyses of the normal and tangential forces acting on the blades, we performed a comparison about the instantaneous wind velocity profiles after 7 complete revolution at 5 different positions downstream in the wake region. These results are presented in Fig. 6 along with the instantaneous iso-contours of the mean velocity field for both turbulence models being investigated considering approximately the last 20 seconds of the simulation. It is possible to observe that both turbulence approach led to similar results in terms of the mean velocity field properties. However, differences in the flow pattern ca also be observed in all positions of the wake region.

A computational cost analysis considering the performance of each of the tested turbulence models showed that both approaches took approximated 24 hours to complete the simulation of one time unit. However, the execution time to calculate one time step was on average 18.38 s for the DES 2-Equations, against 18.83 s for the URANS $k-\omega$ SST. Therefore, considering the computational cost comparison, the performance of DES $k-\omega$ SST model was slightly better over the URANS $k-\omega$ SST.

5. CONCLUSIONS

A numerical investigation about the performance of a 5 MW wind turbine rotor was conducted considering for the same numerical arrangement two difference turbulence models, one vastly applied in the investigation of similar blade-resolved simulations, and another yet being implemented in the simulations of the new generation of wind turbines modeled in its full scale. In this paper, the rotor-only blade-resolved simulations were performed considering the NREL 5 MW reference wind turbine for offshore applications. The CFD simulations provided significant data from which the performance of the NREL 5 MW offshore wind turbine rotor in full scale was evaluated, in terms of, power production, generated thrust, blade loading and wake aerodynamics pattern analysis considering the two different turbulence models tested.

In terms of the computations of the power production, generated thrust and distributed mean tangential and normal forces along the blade span, both turbulence models presented similar and satisfactory results. However significant amount

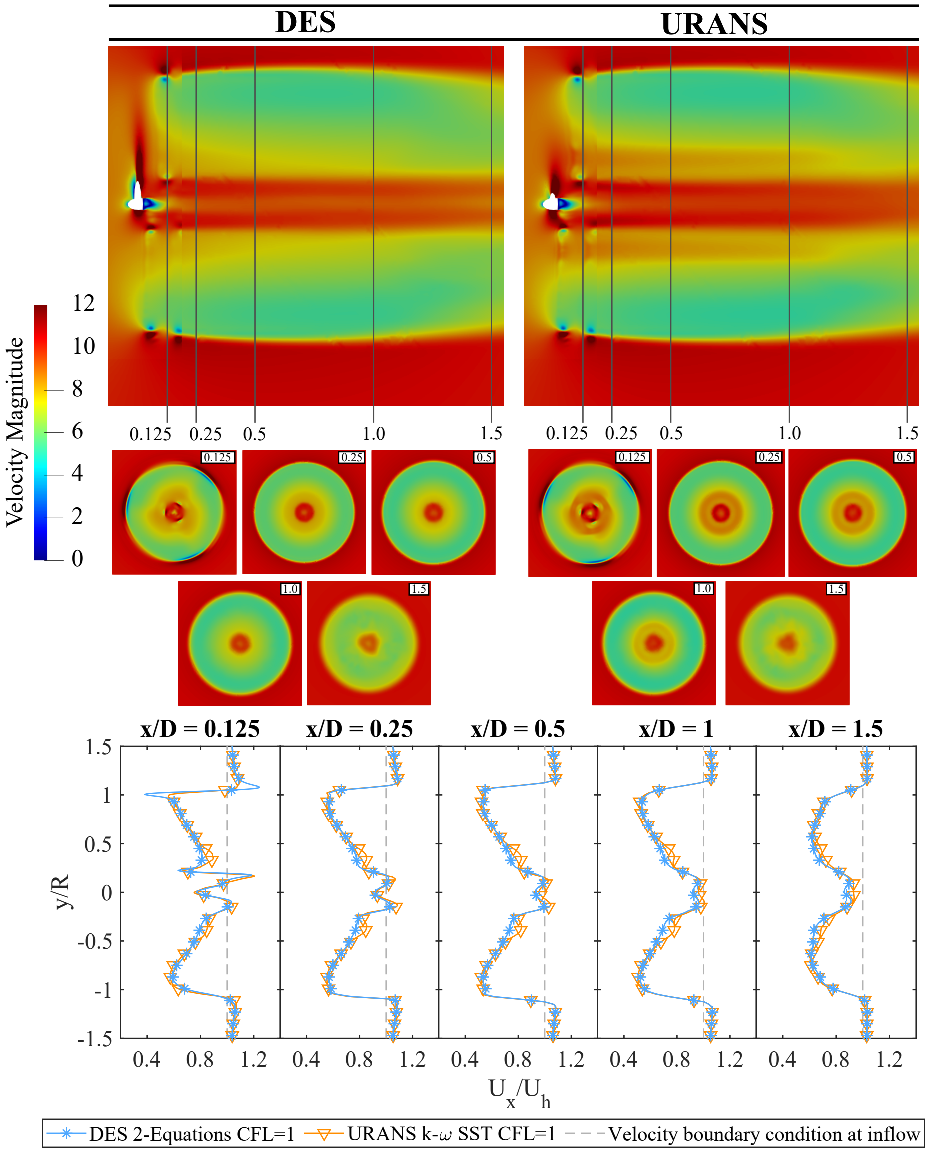
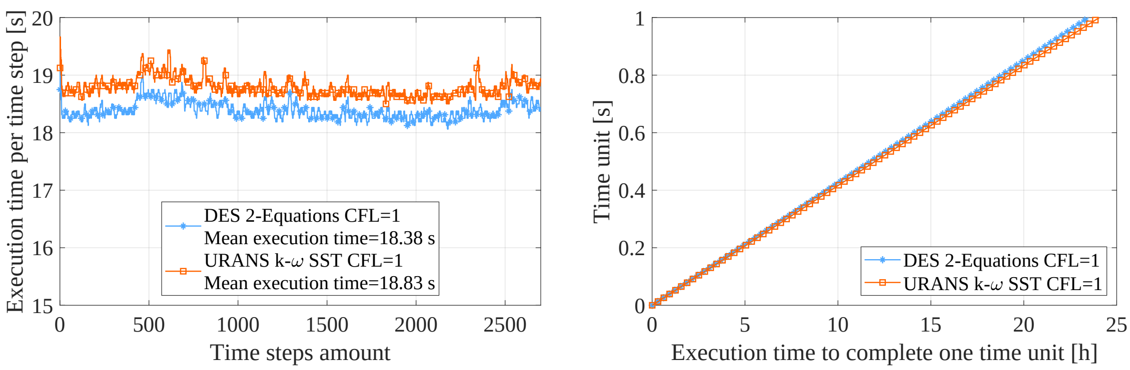


Figure 6: Comparison of the instantaneous wind velocity profiles after 7 complete rotor revolutions at 5 x/D different positions downstream in the wake region along with the iso-contours of the mean velocity field at the same positions for both turbulence models being investigated.



(a) Execution time per time step. (b) Execution time to complete one time unit of simulation.

Figure 7: Computational demand comparison between the DES 2-Equations and URANS $k-\omega$ SST turbulence models.

of flow structures with indicated with higher definition the flow behaviour were captured considering the DES $k-\omega$ SST over RANS $k-\omega$ SST with less computational demand, becoming an attractive solution to be implemented in the modeling of the new generation of larger wind turbines.

6. ACKNOWLEDGEMENTS

M. de Oliveira acknowledges FAPESP (Fundação de Amparo à Pesquisa do Estado de São Paulo), the São Paulo Research Foundation, for the PhD grant - Process number 2018/26207-4. R. C. Puraca acknowledges FUSP/Petrobras for the PhD grant Project number 2019/00171-6. B. S. Carmo acknowledges the support from FAPESP, Proc. 2019/01507-8, for this research, and thanks the Brazilian National Council for Scientific and Technological Development (CNPq) for financial support in the form of a productivity grant, number 312951/2018-3. The authors also acknowledge the grant from the National Laboratory of Scientific Computing (LNCC), CADASE project, which allowed the use of the Santos Dumont supercomputer to run the simulations that generated the results reported in this paper. This work is part of the European Commission Project "High Performance Computing for Wind Energy (HPCWE)" with agreement no. 828799.

7. REFERENCES

Bak, C., Zahle, F., Bitsche, R., Kim, T., Yde, A., Henriksen, L.C., Hansen, M.H., Blasques, J.P.A.A., Gaunaa, M. and Natarajan, A., 2013. "The dtu 10-mw reference wind turbine". In *Danish wind power research 2013*.

Coulling, A.J., Goupee, A.J., Robertson, A.N., Jonkman, J.M. and Dagher, H.J., 2013. "Validation of a fast semi-submersible floating wind turbine numerical model with deepcwind test data". *Journal of Renewable and Sustainable Energy*, Vol. 5, No. 2, p. 023116.

Council, G.W.E., 2022. "Gwecl global wind report 2022". *Global Wind Energy Council: Brussels, Belgium*.

de Oliveira, M., Puraca, R.C. and Carmo, B.S., 2022. "Blade-resolved numerical simulations of the NREL offshore 5 MW baseline wind turbine in full scale: A study of proper solver configuration and discretization strategies". *Energy*, p. 124368.

Duque, E., Van Dam, C. and Hughes, S., 1999. "Navier-stokes simulations of the nrel combined experiment phase ii rotor". In *37th Aerospace Sciences Meeting and Exhibit*. p. 37.

Duque, E.P., Burklund, M.D. and Johnson, W., 2003. "Navier-stokes and comprehensive analysis performance predictions of the nrel phase vi experiment". *J. Sol. Energy Eng.*, Vol. 125, No. 4, pp. 457–467.

El Bassam, N., 2021. "Wind energy". In *Distributed Renewable Energies for Off-Grid Communities*, Elsevier, pp. 149–163.

Farrell, P.E. and Maddison, J.R., 2011. "Conservative interpolation between volume meshes by local Galerkin projection". *Computer Methods in Applied Mechanics and Engineering*, Vol. 200, pp. 89–100. doi:10.1016/j.cma.2010.07.015.

Gaertner, E., Rinker, J., Sethuraman, L., Zahle, F., Anderson, B., Barter, G.E., Abbas, N.J., Meng, F., Bortolotti, P., Skrzypinski, W. *et al.*, 2020. "Iea wind tcp task 37: definition of the iea 15-megawatt offshore reference wind turbine". Technical report, National Renewable Energy Lab.(NREL), Golden, CO (United States).

Hand, M.M., Simms, D.A., Fingersh, L.J., Jager, D.W. and Cotrell, J.R., 2001a. "Unsteady aerodynamics experiment phase v: Test configuration and available data campaigns". Technical report, NREL. doi:10.2172/787980. URL `https://www.osti.gov/biblio/787980`.

Hand, M.M., Simms, D.A., Fingersh, L.J., Jager, D.W., Cotrell, J.R., Schreck, S. and Larwood, S.M., 2001b. "Unsteady aerodynamics experiment phase vi: Wind tunnel test configurations and available data campaigns". Technical report, NREL. doi:10.2172/15000240. URL `https://www.osti.gov/biblio/15000240`.

Hayes, L., Stocks, M. and Blakers, A., 2021. "Accurate long-term power generation model for offshore wind farms in europe using era5 reanalysis". *Energy*, Vol. 229, p. 120603.

Hsu, M.C. and Bazilevs, Y., 2012. "Fluid–structure interaction modeling of wind turbines: simulating the full machine". *Computational Mechanics*, Vol. 50, No. 6, pp. 821–833.

Jasak, H., 1996. *Error analysis and estimation for the finite volume method with applications to fluid flows*. Ph.D. thesis, Imperial College London (University of London).

Jiang, Z., 2021. "Installation of offshore wind turbines: A technical review". *Renewable and Sustainable Energy Reviews*, Vol. 139, p. 110576.

Jonkman, J., Butterfield, S., Musial, W. and Scott, G., 2009. "Definition of a 5-mw reference wind turbine for offshore system development". Technical report, National Renewable Energy Lab.(NREL), Golden, CO (United States).

Karimirad, M., 2014. *Offshore energy structures: for wind power, wave energy and hybrid marine platforms*. Springer.

Lawson, M.J., Melvin, J., Ananthan, S., Gruchalla, K.M., Rood, J.S. and Sprague, M.A., 2019. "Blade-resolved, single-turbine simulations under atmospheric flow". Technical report, National Renewable Energy Lab.(NREL), Golden, CO (United States).

Menter, F.R., 1992. "Improved two-equation k-omega turbulence models for aerodynamic flows". Technical report, NASA Ames.

Menter, F.R., 1994. “Two-equation eddy-viscosity turbulence models for engineering applications”. *AIAA journal*, Vol. 32, No. 8, pp. 1598–1605.

Menter, F., 1993. “Zonal two equation kw turbulence models for aerodynamic flows”. In *23rd fluid dynamics, plasmadynamics, and lasers conference*. p. 2906.

Mittal, A., Sreenivas, K., Taylor, L.K., Hereth, L. and Hilbert, C.B., 2016. “Blade-resolved simulations of a model wind turbine: effect of temporal convergence”. *Wind Energy*, Vol. 19, No. 10, pp. 1761–1783.

Moukalled, F., Mangani, L., Darwish, M. *et al.*, 2016. *The finite volume method in computational fluid dynamics*, Vol. 113. Springer.

Muratova, G., Martynova, T., Andreeva, E., Bavin, V. and Wang, Z.Q., 2020. “Numerical solution of the navier–stokes equations using multigrid methods with hss-based and sts-based smoothers”. *Symmetry*, Vol. 12, No. 2, p. 233.

National Renewable Energy Laboratory, 2005. “Wind turbine design codes certified”. URL `https://www.nrel.gov/news/press/2005/357.html`. (accessed: 10.01.2022).

National Renewable Energy Laboratory, 2021. “Openfast documentation”. URL `https://github.com/openfast`. (accessed: 17.08.2021).

Nikitin, N., Nicoud, F., Wasistho, B., Squires, K. and Spalart, P.R., 2000. “An approach to wall modeling in large-eddy simulations”. *Physics of fluids*, Vol. 12, No. 7, pp. 1629–1632.

Ostachowicz, W., McGugan, M., Schröder-Hinrichs, J.U. and Luczak, M., 2016. *MARE-WINT: new materials and reliability in offshore wind turbine technology*. Springer Nature.

Patankar, S.V., 2018. *Numerical heat transfer and fluid flow*. CRC press.

Pope, S., 1975. “A more general effective-viscosity hypothesis”. *Journal of Fluid Mechanics*, Vol. 72, No. 2, pp. 331–340.

Pope, S.B., 2001. “Turbulent flows”.

Robertson, E., Choudhury, V., Bhushan, S. and Walters, D.K., 2015. “Validation of openfoam numerical methods and turbulence models for incompressible bluff body flows”. *Computers & Fluids*, Vol. 123, pp. 122–145.

Rocha, P.C., Rocha, H.B., Carneiro, F.M., da Silva, M.V. and Bueno, A.V., 2014. “k–ω sst (shear stress transport) turbulence model calibration: A case study on a small scale horizontal axis wind turbine”. *Energy*, Vol. 65, pp. 412–418.

Sanderse, B., Van der Pijl, S. and Koren, B., 2011. “Review of computational fluid dynamics for wind turbine wake aerodynamics”. *Wind energy*, Vol. 14, No. 7, pp. 799–819.

Shur, M., Spalart, P., Strelets, M. and Travin, A., 1999. “Detached-eddy simulation of an airfoil at high angle of attack”. In *Engineering turbulence modelling and experiments 4*, Elsevier, pp. 669–678.

Sorensen, N. and Hansen, M., 1998. “Rotor performance predictions using a navier-stokes method”. In *1998 ASME Wind Energy Symposium*. p. 25.

Sørensen, N.N., Bechmann, A., Réthoré, P.E. and Zahle, F., 2014. “Near wake reynolds-averaged navier–stokes predictions of the wake behind the mexico rotor in axial and yawed flow conditions”. *Wind Energy*, Vol. 17, No. 1, pp. 75–86.

Spalart, P.R., 1997. “Comments on the feasibility of les for wings, and on a hybrid rans/les approach”. In *Proceedings of first AFOSR international conference on DNS/LES*. Greyden Press.

Spalart, P.R., 2000. “Strategies for turbulence modelling and simulations”. *International journal of heat and fluid flow*, Vol. 21, No. 3, pp. 252–263.

Spalding, D., 1961. “Mass transfer through laminar boundary layers—1. the velocity boundary layer”. *International Journal of Heat and Mass Transfer*, Vol. 2, No. 1-2, pp. 15–32.

Thé, J. and Yu, H., 2017. “A critical review on the simulations of wind turbine aerodynamics focusing on hybrid rans-les methods”. *Energy*, Vol. 138, pp. 257–289.

Versteeg, H.K. and Malalasekera, W., 2007. *An introduction to computational fluid dynamics: the finite volume method*. Pearson education.

Wilcox, D.C. *et al.*, 1998. *Turbulence modeling for CFD*, Vol. 2. DCW industries La Canada, CA.

Zhang, Y., Deng, S. and Wang, X., 2019. “Rans and ddes simulations of a horizontal-axis wind turbine under stalled flow condition using openfoam”. *Energy*, Vol. 167, pp. 1155–1163.

Zhang, Y., van Zuijlen, A. and van Bussel, G., 2015. “Massively separated turbulent flow simulation around non-rotating mexico blade by means of rans and ddes approaches in openfoam”. In *33rd AIAA Applied Aerodynamics Conference*. p. 2716.

8. RESPONSIBILITY NOTICE

The authors are the only responsible for the printed material included in this paper.