

PAPER • OPEN ACCESS

High Fidelity Computational Fluid Dynamics Assessment of Wind Tunnel Turbine Test

To cite this article: M. Salman Siddiqui *et al* 2019 *J. Phys.: Conf. Ser.* **1356** 012044

View the [article online](#) for updates and enhancements.



IOP | ebooks™

Bringing you innovative digital publishing with leading voices to create your essential collection of books in STEM research.

Start exploring the collection - download the first chapter of every title for free.

High Fidelity Computational Fluid Dynamics Assessment of Wind Tunnel Turbine Test

M. Salman Siddiqui¹, Trond Kvamsdal^{1,2}, Adil Rasheed²

¹ Department of Mathematical Sciences, Norwegian University of Science and Technology, NO 7491 Trondheim, Norway

² Department of Mathematics and Cybernetics, SINTEF Digital, NO 7465, Trondheim, Norway

E-mail: Muhammad.Siddiqui@ntnu.no, Trond.Kvamsdal@ntnu.no,
Adil.Rasheed@sintef.no

Abstract. We present, what to our best knowledge, is the most accurate numerical investigation of the wind tunnel tests carried out over a model wind turbine (known as *NTNU Blind Test*) at the Norwegian University of Sciences and Technology. We show numerical benchmarking of wake measurements against experimental data and similar investigations performed previously by researchers using Computational Fluid Dynamics (CFD) simulations. We have made a full 3D model of the wind turbine and used Sliding Mesh Interface (SMI) approach to handling the rotation of the rotor. The simulations are done with the use of OpenFoam and the $k - \omega$ Shear Stress Transport model to resolve turbulence using the Reynolds Average Navier-Stokes (RANS) technique. We present the numerically simulated spatial distribution of the flow field across the wake at zero angles of yaw for horizontal lines downstream of the rotor plane as that was the focus of the *NTNU Blind Test* presented in [1]

1. Introduction

In recent years, we have observed a shift in the upcoming energy trends. The focus has shifted from the traditional oil and nuclear-based energy production units to more cleaner and more sustainable renewable resources [2]. Amongst the potential sources for the generation of clean electricity, wind power is believed to have a promising future owing to its significantly higher outputs and lower costs [3, 4]. The enormous potential for future growth has persuaded the wind energy community to develop turbines which can gain maximum yields from regions of large wind potential [5].

From a technical standpoint, this requires the development of highly sophisticated tools/methods capable of analyzing the flow around the wind turbine efficiently, under a minimal time frame [6, 7]. At present, the three most widely adopted procedures for wind turbine flow analysis are based on analytical, numerical or experimental approaches. The Blade Element Momentum (BEM) technique for the analytical solution of blade loading provides reasonable estimates in a short time [8, 9]. One of the widely-known codes



High Fidelity CFD Assessment of Wind Tunnel Turbine Test

employing BEM, with a focus on the simulation of wind turbines, is FAST [10]. The disadvantage of BEM tools is their inability to predict the distributions of pressure and velocity over the structure of the turbine [11, 12]. On the other hand, experimental tests conducted inside wind tunnels represent the actual flow field and blade loads, albeit on a scaled-down geometry of the turbine [13]. The outputs could then be subsequently upscaled to the real dimensions of the turbine. However, it is challenging to find experiments in which the actual dynamics of flow around the turbines are accurately measured and reported [1, 14, 15, 16].

Numerical investigations based on CFD [4] are an attractive alternative to the analytical and empirical approaches; and owing to the availability of high-performance computers and optimized methods for the solution of governing equations [17, 18], the accuracy of CFD has improved significantly over the last decade. CFD allows one to conduct high-fidelity simulations and provides means of visualizing the flow fields, especially velocity and pressure, over the regions of interest. The CFD simulations for present industrial wind turbines require tremendously high computational resources to realistically account for the coupling of flow and blade dynamics [4, 13]. To manage this challenge, various strategies are formulated to model the interaction of the flow field and the wind turbine rotors. One such approach involves modeling the fully resolved (FR) geometry of the turbine, with simplifications to the geometry (an isolated blade, rotor without nacelle) made depending on the computational power at hand. Performing FR CFD analysis requires a multi-physics toolbox like ANSYS Fluent or OpenFOAM [19, 20]. Alternative approaches couple the two-dimensional lift and drag forces, calculated using analytical methods like BEM, with a numerical flow solver employing, typically, the Actuator Disc Method (ADM) or Actuator Line Method (ALM). This strategy requires lesser computational resources in the modeling of wind turbines and wind farms [21, 22]. Some of the known codes available for modeling using these methods include SOWFA by NREL [23], Ellips3D by DTU [24, 25] and WIRE-LES by EPFL [26, 27].

Herein we present high fidelity simulations of the so-called *NTNU Blind Test* performed at the Norwegian University of Science and Technology (NTNU) [15, 1, 16]. Our computational model accurately represented the wind tunnel test set up, and we discretized the full 3D geometry of the wind turbine (rotor, nacelle, and tower). The rotation of the rotor was handled by means of the Sliding Mesh Interface (SMI) technique and the governing equations for the incompressible Navier-Stokes equations on a moving grid are given in Section 2. The most important features of our high-fidelity computational model are described in Section 3. In Section 4, we present the numerically simulated spatial distribution of the flow field across the wake, downstream of the rotor plane, at zero angles of yaw. Finally, we conclude the present study in Section 5.

*High Fidelity CFD Assessment of Wind Tunnel Turbine Test***2. Governing Equations**

The purpose of present work is to exploit a fully-resolved CFD model that explicitly includes the turbine rotation and minimizes other geometric approximations. The Sliding Mesh Interface (SMI) technique is employed to model rotor rotation in the presence of the turbine nacelle and tower. Concerning the SMI methodology, the mesh surrounding the rotor physically rotates, while the mesh away from the rotor is held fixed – the two mesh regions are coupled through interfaces. In literature the method is reported as highly computationally intensive; nevertheless, it has provided accurate estimates of the flow field for rotating assemblies, especially turbines [28]. The governing system of equations for incompressible Navier-Stokes flow are the momentum and continuity equations, written in tensor notation as follows:

$$\frac{\partial \mathbf{u}}{\partial t} + \nabla \cdot ((\mathbf{u} - \mathbf{u}_g) \otimes \mathbf{u}) + \nabla \cdot (\nu + \nu_t) \nabla (\mathbf{u} + (\nabla \mathbf{u})^T) - \nabla p = \mathbf{f} \quad (1)$$

$$\nabla \cdot (\mathbf{u} - \mathbf{u}_g) = 0 \quad (2)$$

Here \mathbf{u} is the fluid velocity and \mathbf{u}_g is the grid velocity (which is zero for the static part of grid), p the fluid pressure, ν and ν_t are respectively the fluid viscosity and the turbulent eddy-viscosity, and \mathbf{f} applied volume forces.

For the closure of turbulence quantities, $\nu + \nu_t$, which accounts for turbulent stresses resulting from Reynolds averaging, is modeled with k - ω Shear Stress Transport (SST) as follows:

$$\frac{\partial k}{\partial t} + \nabla \cdot ((\mathbf{u} - \mathbf{u}_g) \otimes k) = \tau_{ij} \nabla \mathbf{u} + \nabla \cdot \left[\frac{\nu + \nu_t}{\sigma_k} \right] \nabla (k + (\nabla k)^T) - \omega k \quad (3)$$

$$\begin{aligned} \frac{\partial \omega}{\partial t} + \nabla \cdot ((\mathbf{u} - \mathbf{u}_g) \otimes \omega) &= \frac{\gamma}{\nu_t} \tau_{ij} \nabla \mathbf{u} + \nabla \cdot \left[\frac{\nu + \nu_t}{\sigma_\omega} \right] \nabla (\omega + (\nabla \omega)^T) \\ &- \omega^2 \beta + 2(1 - F_1) \frac{\sigma_{\omega 2}}{\omega} \nabla \cdot k \nabla \cdot \omega \end{aligned} \quad (4)$$

The model damping functions, auxilliary relations and the trip terms as defined in [29]

$$\tau_{ij} = \mu_t \left(2S_{ij} - \frac{2}{3} \frac{\partial u_k}{\partial x_k} \delta_{ij} \right) - \frac{2}{3} k \delta_{ij}, \quad S_{ij} = \frac{1}{2} \left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) \quad (5)$$

$$\nu_t = \frac{a_1 k}{\max(a_1 \omega, \Omega F_2)}, \quad \phi = F_1 \phi_1 + (1 - F_1) \phi_2, \quad F_1 = \tanh(\arg_1^4) \quad (6)$$

$$\arg_1 = \min \left[\max \left(\frac{\sqrt{k}}{\beta^* \omega d}, \frac{500 \nu}{d^2 \omega} \right), \frac{4 \sigma_{\omega 2} k}{C D_{k\omega} d^2} \right] \quad (7)$$

$$C D_{k\omega} = \max \left(2 \sigma_{\omega 2} \frac{1}{\omega} \frac{\partial k}{\partial x_j} \frac{\omega}{\partial x_j}, 10^{-20} \right), \quad F_2 = \tanh(\arg_2^2) \quad (8)$$

$$\arg_2 = \max \left(2 \frac{\sqrt{k}}{\beta^* \omega d}, \frac{500 \nu}{d^2 \omega} \right) \quad (9)$$

The model constants are $\gamma=5/9$, $\beta=0.075$, $C_\mu=0.09$, $\sigma_{k1}=0.85$, $\sigma_{\omega 1}=0.65$, $\beta_1=0.075$, $\sigma_{k2}=1.00$, $\sigma_{\omega 2}=0.856$, $\beta_2=0.0828$, $\beta^*=0.09$, and $a_1=0.31$.

*High Fidelity CFD Assessment of Wind Tunnel Turbine Test***3. Approach and methods**

To establish a high-fidelity computational model for a wind turbine there are a number of modeling decisions to be made. Herein, we will in particular mention four aspects of the model that are important with respect to the accuracy of the numerical simulations:

- *Geometric models:* Simplified models represent only the rotor, without the nacelle and tower. However, both the nacelle and the tower affect the flow in the wake, so higher accuracy may be expected when they are included.
- *Aerodynamic models:* Two commonly used simplified methods are the Actuator Line Method (ALM) and the Actuator Disc Method (ADM), which are coupled with two-dimensional lift and drag forces, corrected to account for 3D effects, calculated using analytical methods (BEM). However, one may obtain more accurate results by fully resolving the 3D geometry of the rotor blades.
- *Rotating rotor:* For the case with fully-resolved aerodynamic models, there are two common methods for handling the rotating rotor. In the Multiple Reference Frame (MRF) approach the turbine rotor is made to remain stationary, while the source/sink terms (centripetal and centrifugal forces) are added to the systems of governing equations, yielding the desired rotational effects. In the Sliding Mesh Interface (SMI) technique, the mesh surrounding the rotor is physically rotating, whereas the mesh away from the rotor is held fixed. The SMI technique is most accurate as it can properly handle time variation, whereas MRF is by construction not able to handle transient effects.
- *Turbulence models:* The two broad classes of turbulence models are Reynolds Average Navier-Stokes (RANS) and Large Eddy Simulation (LES), where the latter one is considered most accurate and computationally demanding.

Notice, that in the discussions above, we have assumed that the mesh resolution is sufficiently high in the more advanced methods to actually make them more accurate.

In Table 1, we have displayed the choices made, in the present and previous studies, for the four aspects of modeling described above. The present study has for the three first categories chosen to maximize model accuracy, but due to the computational cost, it is limited to a RANS, as opposed to LES, turbulence model. Among the six previous authors, only one has performed LES. However, in that study only simplified geometric and aerodynamic models were used. In our opinion, the best results of the previously published studies were achieved by Manger [1], who adopted the SMI technique with fully resolved 3D geometry using the RANS framework, as we have herein. However, Manger used a CFD mesh resolution with 5.3×10^6 cells, whereas we used 40×10^6 cells in our study.

The Computer Aided Design (CAD) model is based on the wind tunnel setup described in [1]. The CAD-model is constructed to have smooth interfaces between different structural components, especially between the blades and the nacelle, to avoid abrupt changes in the geometry which are not considered ideal from a computational

High Fidelity CFD Assessment of Wind Tunnel Turbine Test

Authors	Geometric modeling			Aerodynamic modeling			Rotor modeling		Turbulence modeling	
	Rotor	Nacelle	Tower	ALM	AD	FR	MRF	SMI	RANS	LES
Lund	X				X			X	X	
Manger	X	X	X			X		X	X	
Hansen	X					X	X		X	
Sørensen	X			X	X					X
Melheim	X				X				X	
Kalvig	X		X		X				X	
Present	X	X	X			X		X	X	

Table 1: An overview of the geometrical complexity, aerodynamic models, techniques for handling rotor rotation and turbulence model employed in the present and previous studies of the wind tunnel test performed at NTNU [1]

point of view. An overview of the computational model is displayed in Figure 1. The mesh is refined in the vicinity of the turbine to capture the highly dynamical flow pattern in that area. As the flow field becomes less dynamic away from the turbine, the mesh resolution is decreased towards the outer walls of the tunnel. Since one of our main interests is to explore the wake in the downstream direction, a region of highly concentrated mesh cells is generated downstream of the turbine to capture the flow dynamics accurately there.

The flow domain is decomposed into a rotating and a stationary part that is solved using the SMI technique as described above. The rotating part contains the rotor and its vicinity as illustrated in Figure 1. We have applied boundary conditions in order to comply as closely as possible to the experimental setup. The inflow, outflow and wall boundary conditions (as defined in OpenFOAM 4.0 [30]) are imposed to inlet, outlet and outer boundaries of the tunnel. The SMI-technique handles the coupling of the inner rotating domain with the static surrounding domain.

The numerical solvers are developed in OpenFOAM-4.0 (OF) [30], which is an open-source tool and based on the finite volume method (FVM). For these transient simulations, a first-order implicit scheme is employed to handle the time-dependent term. The convection term of the Navier-Stokes is discretized using the bounded Gauss upwind scheme, whereas for the diffusion term a Gauss linear correction is used. The resulting linear system of equations is solved using the Geometric Agglomerated Algebraic Multigrid (GAMG) method. The simulations are run until the convergence criteria is achieved for GAMG i.e., the linear algebra residuals fall below 10^{-6} .

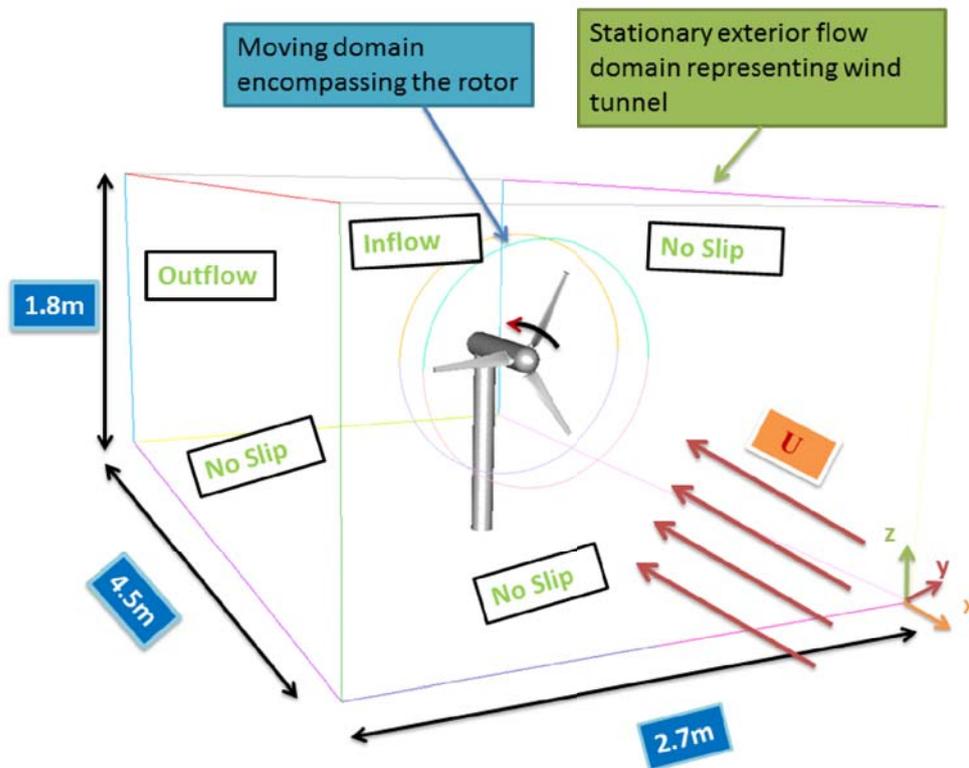
High Fidelity CFD Assessment of Wind Tunnel Turbine Test

Figure 1: *NTNU Blind Test* (TSR=6): Overview of computational domain and imposed boundary conditions. Wind tunnel dimensions used in the numerical simulations are set to $2.7\text{m} \times 4.5\text{m} \times 1.8\text{m}$ which is the same dimensions as the wind tunnel at NTNU [1]

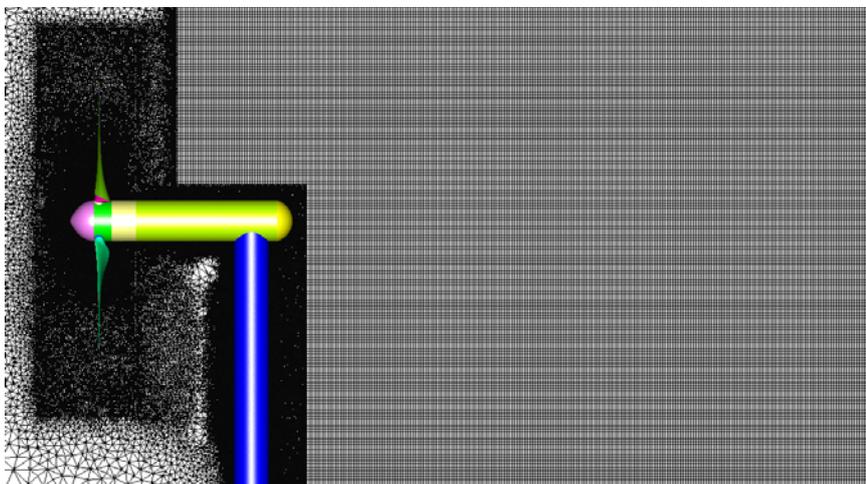


Figure 2: *NTNU Blind Test* (TSR=6): Computational mesh comprises of 40×10^6 cells. Mesh block downstream with refined cells for correct evaluation of wake profiles.

*High Fidelity CFD Assessment of Wind Tunnel Turbine Test***4. Results and discussions**

The participants in the *NTNU Blind Test* presented in [1] were asked to provide predictions of the spatial distribution across the wake, at zero angle of yaw, for three positions downstream of the rotor plane, $X/D = 1, 3$ and 5 , and three operating conditions (i.e. tip speed ratios), $\lambda = 3, 6$ and 10 . In order to provide data for three turbine rotor rotations, our fully resolved 3D simulations using SMI and RANS required 25 days on the high-performance supercomputer *Vilje* using 512 cores with 2.6 GHz Intel(R) Xeon(R) E5-2670 processor having 4Mb cache memory. We were therefore only able to do one of the three different operating conditions and present herein the case with inflow velocity (U_{ref}) of 10m/s and turbine rotating speeds of $\Omega = 134$ rad/s, which corresponds to the tip speed ratio $\lambda = 6$ and $Re = 1 \times 10^5$ at the blade tip.

The comparison of the wake velocity deficit with experiments and the results obtained by Manger [1] is displayed in Figure 3, where we plot $U = 1 - U_{wake}/U_{ref}$ over the horizontal line ($y/R = -2$ to $y/R = +2$) located at $X/D = 1$ and $X/D = 3$ downstream of the turbine. Given the inclusion of the nacelle and tower in the present simulations, the overall shape of the wake generated by the rotor is not axisymmetric and manifests an uneven expansion of the wake in the horizontal direction. We notice that adjacent to the tower a kink in the profile, which is believed to occur because of the wake generated by the rotor, is captured by our fully resolved 3D simulation using SMI to handle the rotation of the rotor. Our simulations have captured well even the small scales generated by the tower as shown in Figure 4 and compare significantly better to the experiments than the results reported by Manger [1]. The differences are assumed to be caused by the fact that we have eight times more CFD cells in our simulations than those performed by Manger.

5. Conclusion

We have herein presented high-fidelity simulations of the wind tunnel tests of a model wind turbine (known as the *NTNU BlindTest*) carried out at the Norwegian University of Science and Technology in 2011 by Krogstad et al [1]. Our computational model accurately represented the wind tunnel test setup and we discretized the full 3D geometry of the wind turbine (rotor, nacelle, and tower) using 40×10^6 finite volume cells. The rotation of the rotor was handled by means of the Sliding Mesh Interface (SMI) technique and we employed the Reynolds Average Navier-Stokes (RANS) framework with the $k - \omega$ SST turbulence model. The simulation was done using OpenFOAM-4.0 (OF) [30] and the high-performance supercomputer *Vilje*.

The focus of the *NTNU Blind Test* was to provide predictions of the spatial distribution across the wake at zero angles of yaw for horizontal lines downstream of the rotor plane, and our results compared very well with the experimental data. To our knowledge, the results presented herein are the most accurate simulations conducted for the *NTNU Blind Test*, and may indeed be described as *high-fidelity*.

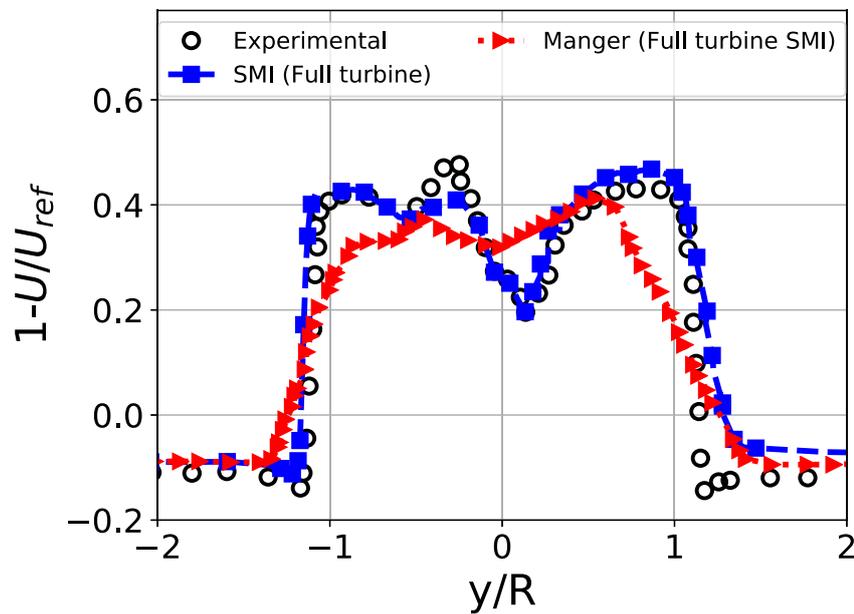
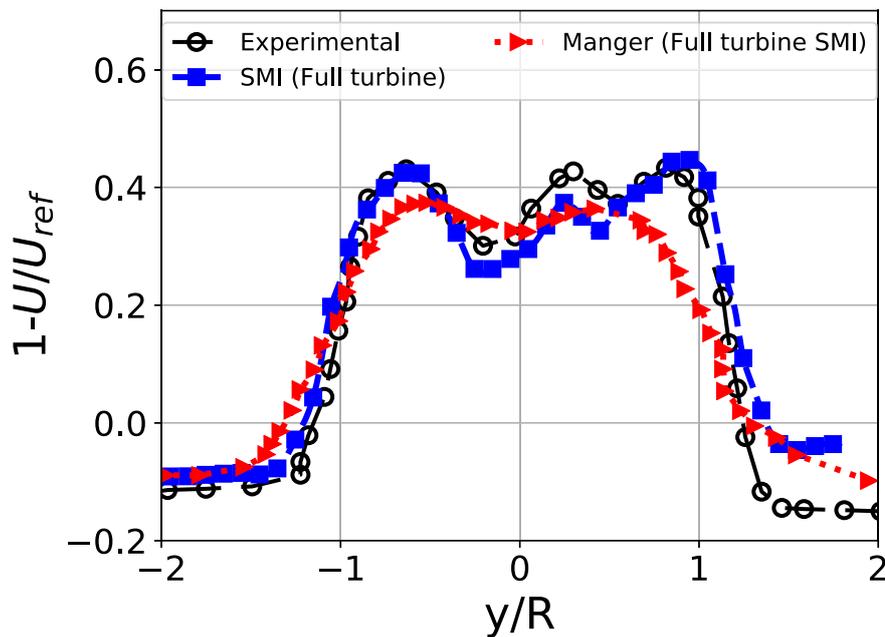
High Fidelity CFD Assessment of Wind Tunnel Turbine Test(a) $x/D=1$ (b) $x/D=3$

Figure 3: *NTNU Blind Test* (TSR=6): Comparison of the wake velocity deficit at horizontal lines downstream of the turbine ((a) $x/D = 1$ and (b) $x/D = 3$) obtained in the present study with those obtained by numerical simulations by Manger and experimental wind tunnel tests by Krogstad et al [1].

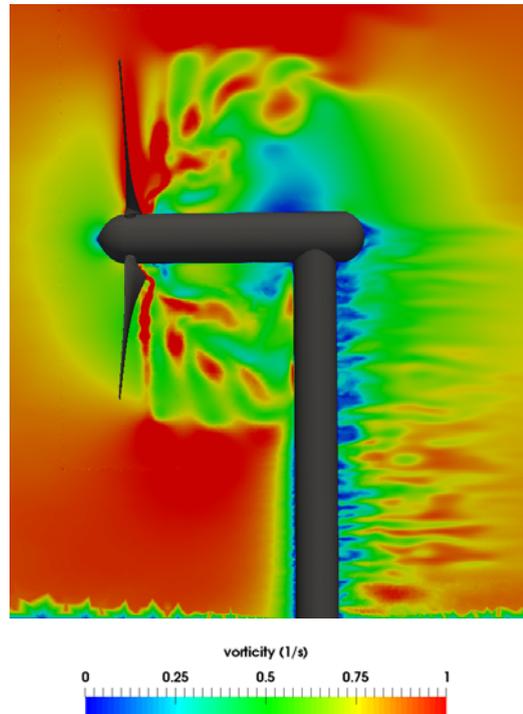
High Fidelity CFD Assessment of Wind Tunnel Turbine Test

Figure 4: *NTNU Blind Test* (TSR=6): Contours of velocity magnitude over a cross-section aligned with the inlet velocity through the center of the turbine. Tip vortices are evident along with smaller flow disturbances emanating from the turbine tower. Fully resolved 3D turbine geometry and the use of the SMI technique for handling the rotation of the rotor captures well even small-scale flow structures and the turbulent flow field around the wind turbine.

We think that such high-fidelity solutions can serve to generate snapshots for Reduced Order Methods (ROM) based on Proper Orthogonal Decomposition. Such methods have been tested before in study [31, 32] for benchmark flow problems (such as flow around an airfoil [33], cylinder [32], etc.) and have shown encouraging results for low Reynolds number flow. We are now in the process of enhancing this previous attempt to realistic Reynolds number flow and will address the challenge of providing ROM for the *NTNU Blind Test* model turbine.

Acknowledgments

The authors acknowledge the financial support from the Norwegian Research Council and the industrial partners of *NOWITECH: Norwegian Research Centre for Offshore Wind Technology*, *OPWIND: Operational Control for Wind Power Plants* and *FSI-WT* (Grant No.:216465). Furthermore, the authors greatly acknowledge the *Norwegian Metacenter for Computational science* (NOTUR-reference number: NN9322K/1589) for giving us access to the *Vilje* high-performance computer at NTNU.

*High Fidelity CFD Assessment of Wind Tunnel Turbine Test***References**

- [1] Krogstad P A and Eriksen P E 2013 *Renewable Energy* **50** 325 – 333
- [2] Hansen M, Sørensen J, Voutsinas S, Sørensen N and HA M 2006 *Progress in Aerospace Sciences* **42** 285–330
- [3] Qin C, Saunders G and Loth E 2017 *Applied Energy* **201** 148 – 157
- [4] Siddiqui M S, Rasheed A, Tabib M and Kvamsdal T 2019 *Renewable Energy* **132** 1058 – 1075
- [5] Herraes I, Akay B, van Bussel G J W, Peinke J and Stoevesandt B 2016 *Wind Energy Science* **1** 89–100
- [6] Thé J and Yu H 2017 *Energy* **138** 257 – 289
- [7] Siddiqui M S, Durrani N and Akhtar I 2015 *Renewable Energy* **74** 661–670
- [8] Bai C J and Wang W C 2016 *Renewable and Sustainable Energy Reviews* **63** 506–519
- [9] Siddiqui M S, Rasheed A, Kvamsdal T and Tabib M 2017 *Journal of Physics: Conference Series* **854** 012043
- [10] Jonkman J M, Butterfield S, Musial W and Scott G 2009, Tech Rep NREL/TP-500e38060 *National Renewable Energy Laboratory*
- [11] Yang H, Shen W, Xu H, Hong Z and Liu C 2014 *Renewable Energy* **70** 107–115
- [12] Monteiro J P, Silvestre M R, Piggott H and Andre J C 2013 *Journal of Wind Engineering and Industrial Aerodynamics* **123** 99–106
- [13] Siddiqui M S, Rasheed A, Kvamsdal T and Tabib M 2017 *Energy Procedia* **137** 460 – 467
- [14] Bottasso C L, Campagnolo F and Petrovic V 2014 *Journal of Wind Engineering and Industrial Aerodynamics* **127** 11–28
- [15] Krogstad P A and Sætran L 2012 *Wind Energy* **15** 443–457
- [16] Krogstad P A, Sætran L and Adaramola M S 2015 *Journal of Fluids and Structures* **52** 65–80
- [17] Miller A, Chang B, Issa R and Chen G 2013 *Renewable and Sustainable Energy Reviews* **25** 122–134
- [18] Khalid S, Rabbani T, Akhtar I, Durrani N and Siddiqui M S 2015 *ASME Journal of Computational and Nonlinear Dynamics* **10** 041012(9)
- [19] Jasak H 1996 *Imperial College, University of London*
- [20] Tabib M, Rasheed A, Siddiqui M S and Kvamsdal T 2017 *Energy Procedia* **137** 477 – 486
- [21] Stevens R J, Tossas L A M and Meneveau C 2018 *Renewable Energy* **116** 470 – 478
- [22] Stevens R J, Graham J and Meneveau C 2014 *Renewable Energy* **68** 46 – 50
- [23] Fleming P A, Gebrad P M, Lee S, van Wingerden J W, Johnson K and Churchfield M 2014 *Renewable Energy* **70** 211 – 218
- [24] Heinz J C, Sørensen N N and Zahle F 2016 *Wind Energy* **19** 2205–2221
- [25] Ivanell S, Sørensen J N, Mikkelsen R and Henningson D *Wind Energy* **12** 63–80
- [26] Paik J, Sotiropoulos F and Porté-Agel F 2009 *Heat and Fluid Flow* **30** 286 – 305
- [27] Bastankhah M and Porté-Agel F 2014 *Renewable Energy* **70** 116 – 123
- [28] Siddiqui M S, Durrani N and Akhtar I 2013 *ASME 2013 Power Conference* (American Society of Mechanical Engineers) p V002T09A020
- [29] Menter F 1994 *AIAA Journal*. *32* 1598–1605
- [30] Jasak H 5-8 January, 2008 *47th AIAA Aerospace Sciences Meeting Including the New Horizons Forum and Aerospace Exposition* **52** (AIAA 2009–341)
- [31] Leblond C, Allery C and Inard C 2011 *Computer Methods in Applied Mechanics and Engineering* **200** 2507 – 2527
- [32] Fonn E, Tabib M, Siddiqui M S, Rasheed A and Kvamsdal T 2017 *Elsevier Energy Procedia* **137** 452 – 459
- [33] Siddiqui M S, Fonn E, Kvamsdal T and Rasheed A 2019 *Energies* **12** 1271