Near-Wake Flow Simulations for a Mid-Sized Rim Driven Wind Turbine

Bryan E. Kaiser[1] and Svetlana V. Poroseva[2]
University of New Mexico, Albuquerque, New Mexico, 87131

Erick Johnson[3]
Montana State University, Bozeman, Montana, 87185

Rob O. Hovsapian[4]
Idaho National Laboratory, Idaho Falls, Idaho, 83415

A relatively high free stream wind velocity is required for conventional horizontal axis wind turbines to generate power. This requirement significantly limits the area of land for viable onshore wind farm locations. To expand a potential for wind power generation onshore, new wind turbine designs capable of wind energy harvesting at low wind speeds are in development. The aerodynamic characteristics of such wind turbines are notably different from industrial standards. The optimal wind farm layout for such turbines is also unknown. Accurate and reliable simulations of a flow around and behind new wind turbine designs are required. The current paper investigates the performance of a mid-sized Rim Driven Wind Turbine (U.S. Patent 7399162) developed by Keuka Energy LLC.

Nomenclature

σ	=	*ratio of planform area to swept area (solidity)*
τ	=	*ideal torque*
τ_0	=	*induced torque*
ω	=	*ideal angular velocity of the turbine*
ω_0	=	*prescribed angular velocity of the turbine*
a	=	*axial induction factor*
C_T	=	*thrust coefficient*
C_P	=	*power coefficient*
D	=	*turbine diameter*
f	=	*aerodynamic loss factor*
I	=	*turbine mass moment of inertia*
R	=	*blade length*
Re	=	*Reynolds number*
Tu	=	*turbulence intensity*
U	=	*mean velocity in the axial direction*
U_∞	=	*free stream wind velocity in the axial direction*

I. Introduction

A requirement of a relatively high free-stream wind velocity limits areas suitable for wind energy harvesting by conventional horizontal axis wind turbines (HAWTs). As a mean to overcome this limitation, small to mid-

[1]Graduate student, Mechanical Engineering, MSC01 1105, 1 UNM Albuquerque, NM 87131-0001, AIAA Student Member.
[2]Assistant professor, Mechanical Engineering, MSC01 1105, 1 UNM Albuquerque, NM 87131-0001, AIAA Senior Member.
[3]Assistant Professor, Mechanical Engineering, Montana State University, Bozeman, MT 87185, AIAA Member.
[4]Scientist, Idaho National Laboratory, Idaho Falls, ID 83415-3810.

sized wind turbine designs capable of power generation at low wind speeds have started to receive a renewed interest from industry and consumers. The Keuka rim-driven wind turbine (RDWT) (U.S. Patent 7399162) developed by Keuka Energy LLC is one of such new designs currently in production and testing. The objective of our study is to estimate its performance using computational simulations in a range of wind speeds characteristic of states with a low wind speed (wind class three or below), such as, for example, Florida, New Mexico, and Texas.

The Keuka RDWT is a drag-driven wind turbine designed for wind energy extraction in locations of wind class three or below. The design is passive-stall-controlled for simplicity and lower capital expense. It features high solidity (16 blades) and is power-rated at 15kW[1]. The turbine diameter considered in the current study is approximately 7.5 *m*. In the future, results of the current study will be used to investigate whether RDWT generates wakes less destructive than those of the conventional three-bladed HAWTs and thus, can be implemented in larger numbers in wind farms. They also will be used to optimize the number of blades to minimize the downstream wakes and the flow disturbance, and to improve the RDWT efficiency.

In this study, fluid forces on the blades of RDWT and the flow structure in its near wake were examined analytically and computationally. The analytical model (Gluart's momentum theory[2]) is based on empirical data[3] and provides estimates for the turbine thrust and power. It does not account for the flow turbulence though. Computations are conducted using commercial computational fluid dynamic (CFD) software, STAR CCM+ by CD Adapco[4]. Results of simulations are compared with the experimental data[1] collected during six months from a prototype RDWT installation at the test site at the Wind Science and Engineering center, Texas Tech University located at the Reese Technology Center in Lubbock, Texas. Data for other turbines are used for comparison as well.

II. Analytical Model

Following the classical momentum theory, the wind turbine power coefficient C_P can be calculated analytically[3] in terms of the axial induction factor as follows:

$$C_P = 4a(1-a)^2.$$

The axial induction factor a is the variable which corresponds to the degree to which the turbine slows the mean velocity of the flow over the turbine. Each root of the power coefficient equation represents a different flow state and axial induction factor.

Similarly, two axial induction factors may be calculated for a given a single thrust coefficient. The first corresponds to a windmill state and the second one to a turbulent wake state. Both states are relevant to the analysis of a flow around and behind RDWT as low angles of attack across the blades, broad chord length, and high solidity (σ = 0.42) (Fig. 1) promote separation in the wake behind RDWT.

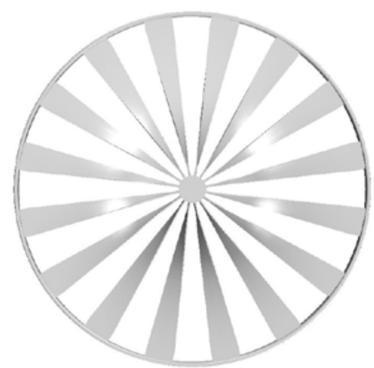

Figure 1. The Keuka Wind Turbine.

Buhl[4] derived an equation for calculating the turbulent wake state axial induction factor that fits the empirical data curves and potentially accounts for the tip and hub losses:

$$C_T = \begin{cases} 4a|1-a| & a \le 0.4 \\ \frac{8}{9} + \left(4f - \frac{40}{9}\right)a + \left(\frac{50}{9} - 4f\right)a^2 & a > 0.4 \end{cases}$$

The RDWT blades are essentially twisted plates shown in Fig. 1. The shape allows for relatively easy analytical derivation of fluid forces acting on the turbine. The RDWT thrust force and the thrust force coefficient are calculated analytically by considering the momentum conservation in an incompressible flow and neglecting viscous forces.

III. Computational Model

A complex structure of a flow around RDWT presents many challenges for conducting accurate and reliable numerical simulations. The presence of dynamic stall, turbine operation over a broad range of Reynolds numbers, and interaction with atmospheric turbulence are just a few of such challenges. Requirements for the appropriate computational grid resolution render Direct Numerical Simulations (DNS) and Large Eddy Simulations (LES) to be computationally unfeasible for industry. Reynolds-Averaged Navier-Stokes (RANS) turbulence models are a potential alternative without the computational cost of DNS and LES. RANS models are typically used by industry for simulating the flow of the far wake of wind turbines[5] and readily available in many CFD software packages. In this study, the ability of RANS models to accurately simulate a near-wake flow of RDWT and quantitatively make power and thrust predictions was examined.

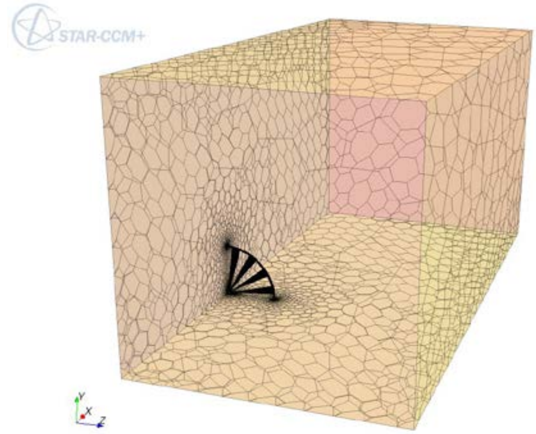

Figure 2. The quarter-turbine computational domain.

Our previous sensitivity studies of flow simulations over a rotating disk[6] and around a single RDWT blade[7] informed the choice of RANS models to be used for this study. It was shown, that the standard *k*-ε turbulence model provides an accurate description of the flow physics over the disk surface if paired with a suitable grid free of highly skewed cells[6]. In the current study, the realizable *k*-ε turbulence model[8] was chosen for computations. It generates solutions as accurate as the standard *k*-ε model does, but maintains the solution accuracy for lower quality grid cells[4]. Additionally, the realizable *k*-ε model in STAR CCM+ is accompanied by the two-layer all y^+ wall treatment option which uses the initial cell y^+ value as a criteria whether to resolve explicitly the viscous sublayer *or* to apply wall functions[4,9].

Computations were also conducted with the shear stress transport (SST) turbulence model[10] because of the known model's ability to accurately represent the structure of the adverse pressure-gradient flows including flows with separation, and because the SST model is accompanied by a similar all y^+ wall treatment as the realizable *k*-ε model.

The STAR CCM+ second order upwind advection scheme was chosen for the computation of all derivatives[4].

Hybrid computational grids were generated and utilized in our study as they offer a suitable compromise between accuracy and ease of use[11]. A computational domain for the quarter-of-RDWT simulations is shown in Fig. 2. The grid is composed of cells of two different types (Fig. 3). The domain size is 4*R*x4*R*x10*R*, where *R* is the RDWT radius. The thickness of 5 prism layers of the growth rate of 1.07 is 0.06 *m*. Polyhedral cells fill the remaining volume and were varied in density. The grid parameters were adopted from our previous work[7] where the sensitivity analysis of simulations of a flow around a single blade was conducted. Polyhedral-only meshes for the same domain were also created and varied by the growth rate from the turbine surface. The quarter-turbine grids varied in size from 780,000 to 1,800,000 elements. Periodic boundary conditions were assigned to two planes that cut across the turbine. Other planes that limit the computational domain were treated as pressure outlets.

The grid parameters used in simulations of a flow around the whole RDWT were identical to the quarter-turbine grids of the polyhedral cells only. The domain size was 10*D*x10*D*x10*D*. The mesh sizes varied from 815,000 to 3,500,000 cells.

In simulations of the quarter of RDWT, the turbine was located at $x = 4R$ from the inlet plane. The whole turbine was located at $x = 5D$. The uniform inlet velocity equal to the free-stream wind speed was assigned as the inlet condition in all simulations. Its value varied in the range of inlet velocities from 1 m/s to 12 m/s corresponding to the range of Reynolds numbers (based on the turbine diameter) from 480,000 to 5,700,000. No symmetry planes or periodic conditions were necessary in the whole turbine simulations. Therefore, the remaining surfaces for this geometry were specified as pressure outlets. Angular velocities of the turbine were specified to correspond to the values obtained from the RDWT prototype at the aforementioned free-stream wind speeds. The turbine tower was not represented.

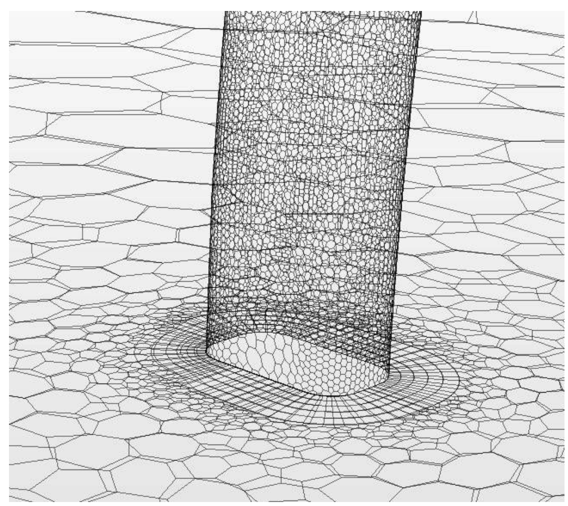

Figure 3. The quarter-turbine hybrid grid enlarged in the area around the turbine rim.

IV. Results

In the previous study[7], the mean axial induction factor was found to be a = 0.0685 without including the tip and hub losses (f = 1) for the flow around the quarter of RDWT. The mean thrust and power coefficients corresponding to that mean axial induction factor were calculated using the classical momentum theory and the modifications[2,3] for the turbulent wake states discussed above in the *Analytical Model* section. They were found to be C_T = 0.2552 and C_P = 0.2377.

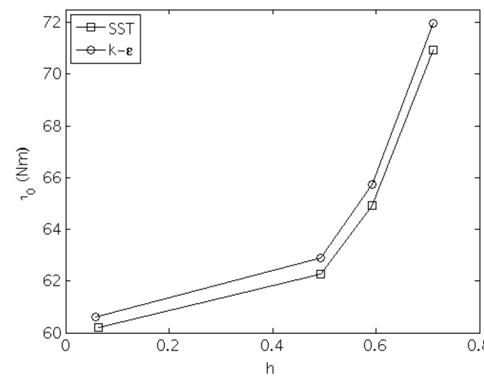

Figure 4. Grid convergence study.

In the present study, grid-independent values of the torque induced on the whole RDWT were computed for each set of the inlet and angular velocities (as explained in the *Computational Model* section) by using Richardson Extrapolation technique[11]. The data for the quarter turbine were recalculated to include cases when hybrid grids with prism layers were used. Computations were conducted with both turbulence models. Grid convergence for the torque is shown in Fig. 4 along with the extrapolated value and three computed torque values in the asymptotic range (Baker[11]) for the complete turbine.

Figure 5 shows the distribution of the y^+-values at the center of the wall cells in the quarter-turbine grid with and without prism layers at the free stream wind velocity of 1 m/s and prescribed rotation. The complex geometry causes a large variety of the wall cell y^+-values. and justifies the implementation of the all y^+ wall treatments for the selected turbulence models.

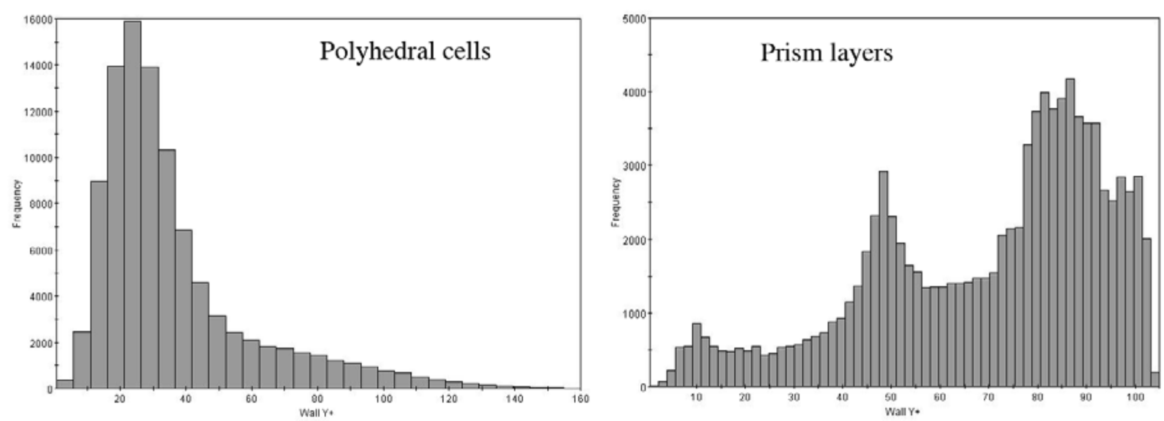

Figure 5. Distribution of the y^+-values at the center of the wall cells in the quarter-turbine grids at the free stream wind velocity of 1 m/s.

The extrapolated torque τ_0, the turbine's mass moment of inertia I, and the specified angular velocity ω_0 from the experimental data were used to calculate the ideal torque τ and ideal rotation rate ω, using the expressions below:

$$\tau - \frac{1}{2} I \omega_0^2 = \tau_0, \qquad \tau - \frac{1}{2} I \omega^2 = 0.$$

The product of the extrapolated torque (the torque induced by the flow) and the ideal rotation rate is the ideal mechanical power of the turbine. The power coefficient values obtained with the two turbulence models for the whole turbine at the free-stream wind velocity of 1, 3, 6, 9, and 12 m/s are shown in Fig. 6 along with the data for the quarter turbine obtained using hybrid grids at the free-stream wind velocity of 1, 6, and 12 m/s. The computed power coefficient curve suggests a peak power coefficient of 0.3 at the free-stream wind velocity of 4 to 5 m/s. The calculated data appears to underpredict the turbine power performance to compare with the experimental data. The experimental data also showed that the turbine cannot extract power at the wind speed less than 3 m/s.

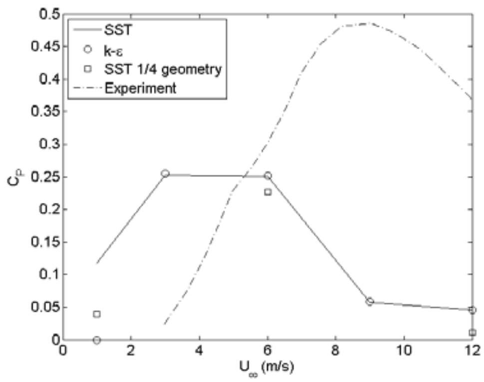

Figure 6. Power coefficient.

The thrust coefficient was computed at the same free-stream wind velocities. Its values were also compared with the data for three industrial turbines[12] (Fig. 7). The mean thrust coefficient calculated from the experimental data in our previous study[7] using momentum theory (C_T = 0.1954) disagrees with the computed thrust coefficients in Fig. 7. Comparison of the computational results and thrust coefficients from industrial turbines suggests that the computed thrust coefficients may be underpredicted.

Experimental data for the flow structure in a wake of the RDWT prototype is currently unavailable. Therefore, the simulation results are qualitatively compared with the experimental data for other turbines and with the other simulations data. Computations were conducted on the finest polyhedral-only grids for the whole turbine and its quarter using the two turbulence models.

Figure 8 shows that the mean axial velocity profiles at the distances of one and three radius behind the turbine obtained with the two turbulence models are in close agreement with one another in a case of the whole turbine simulations. Slight discrepancy of the results is observed in simulations of the quarter turbine. In the figure, the LES results for a conventional turbine from Sørenson et al[13] are also given for the comparison. The LES results were obtained using a grid of 6,000,000 cells. The extent to which the turbine geometry affects the near wake is difficult to predict[5]. The RDWT design may be a reason for the different maximum mean axial velocities obtained in the current simulations and in LES[13] at the distance of one radius behind RDWT. However, underprediction of the velocity deficits and turbulence intensity peaks as well as overprediction of the turbulent kinetic energy dissipation when RANS models[14] are used in simulations of the turbine wakes[5,15] is well documented[5,15] and appears to be the primary cause of the difference between the RANS and LES solutions shown at the distance of three radius behind RDWT.

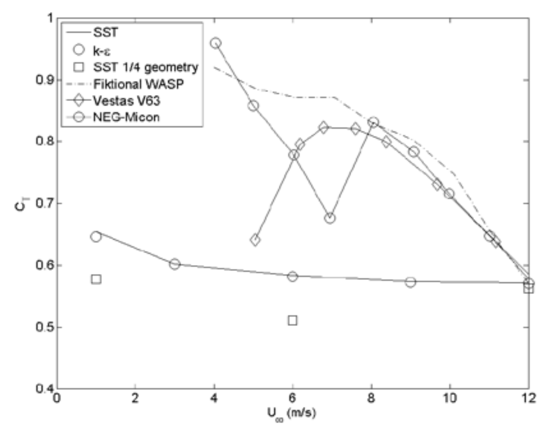

Figure 7. Thrust coefficient.

Figure 9 shows the velocity contour plots for the whole turbine and its quarter obtained using polyhedral grids at U_∞ = 1 m/s.

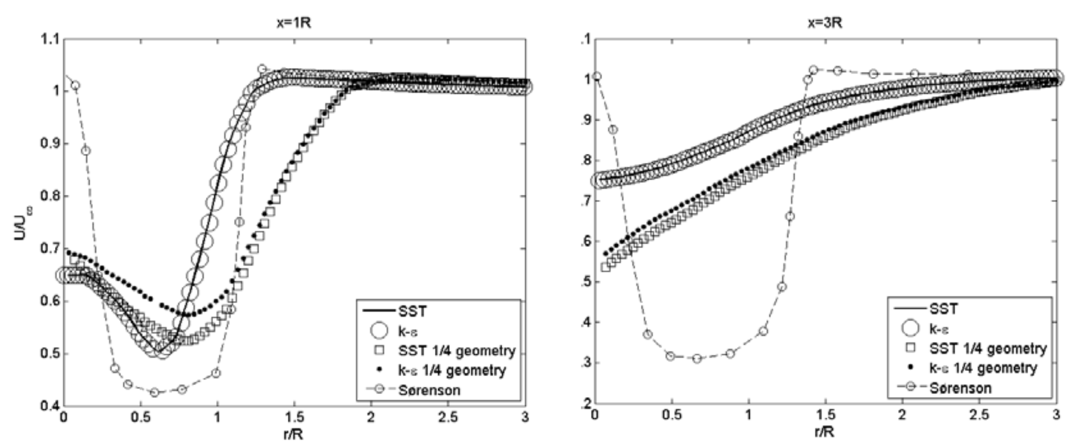

Figure 8. Mean wake velocity.

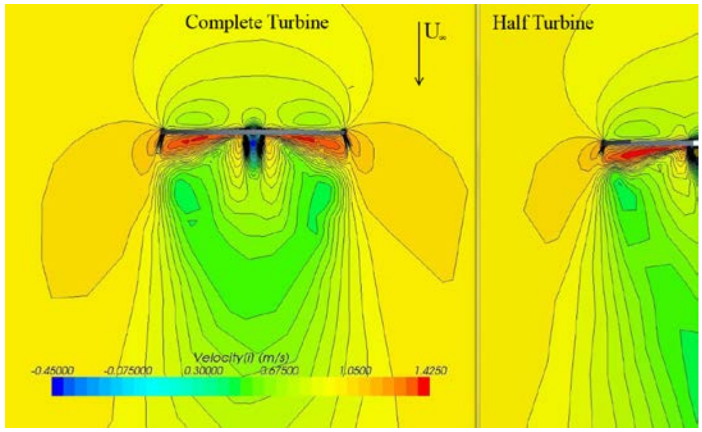

Figure 9. Mean wake velocity contours for U_{∞}= 1m/s.

Figure 10 shows the simulation results for the turbulence intensity. Results obtained with the two turbulence models are in agreement with one another for the whole turbine and its quarter. The reduced turbulence intensity values obtained in the quarter-of-turbine simulations are suspected to be the result of using the symmetry plane boundary conditions. Maeda et al.[16] performed a wind tunnel test with the similar flow parameters and found the turbulence intensity peaks to exist as far behind the turbine as five diameters. At x = 10D, Maeda et al. observed the single broad, shallow peak. In the current simulations, a similar peak is observed at x = 1.5D. Therefore, the data of Maeda et al.[16] and that of Rados et al.[15] may be an indication that the turbulence intensity is underpredicted in our simulations.

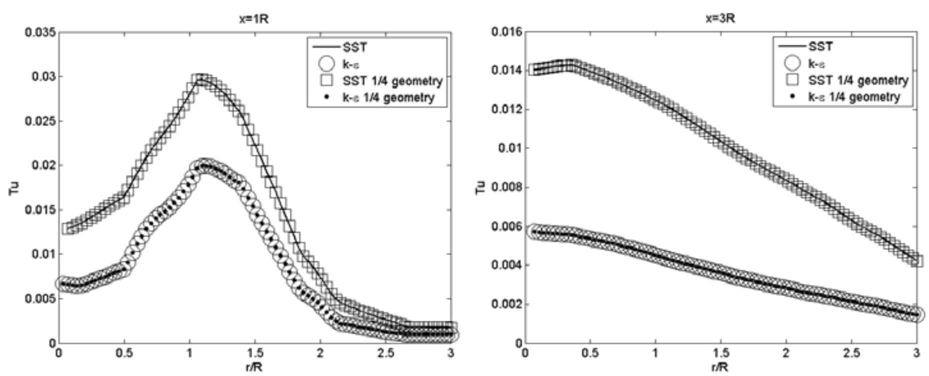

Figure 10. Wake turbulence intensity profiles.

Conclusions

The realizable k-ε turbulence model with the two-layer all y^+ wall treatment option produced remarkably similar results to the SST turbulence model with the all y^+ wall treatment option in all simulations. The realizable k-ε model and SST model agreed better than the standard k-ε model and the SST model in our previous work.

The estimates of the power coefficient obtained using the mechanical torque and rotation method underpredicted the turbine performance. The power coefficients obtained in the whole-turbine simulations were marginally more accurate than those obtained in the quarter-turbine simulations. Similar results were observed for the thrust coefficients, although validation was not performed for this coefficient.

Velocity and turbulence intensity profiles in the near-wake behind RDWT demonstrated that the selected RANS models are capable of capturing the general shape of the flow structure. As no validation data exists for the RDWT wake yet, comparison was made for other wind turbine designs. The results of such comparison suggest that the velocity deficit, turbulence intensity, and turbulent kinetic energy in the RDWT wake may be underpredicted. This conclusion has been well documented for the other turbine designs [15,17]. Still, validation data specific for the RDWT design are necessary to draw solid conclusions, because the turbine geometry may play a significant role.

The results also indicate that the use of the symmetry plane boundary conditions with periodic inflow/outflow to reduce the cost of simulations has a negative impact on the accuracy of computations of the velocity and turbulence intensity profiles in the given flow geometry.

Acknowledgments

The research was supported in part by the Junior Faculty UNM-LANL Collaborative Research Grant and by the Center for Advanced Power Systems (CAPS) at the Florida State University. The authors would also like to acknowledge the Center of Advanced Research Computing of the University of New Mexico for providing the access to high-performance computing facilities and consulting support for this research, and CD-Adapco for providing STAR CCM+ to the University of New Mexico for academic purposes.

References

[1] Rankin, A. J., Poroseva, S. V., and Hovsapian, R. O., "Power Curve Data Analysis for Rim Driven Wind Turbine," *ASME Early Career Technical Journal*, Vol. 10, 2011, pp. 27-32.

[2] Spera, D. A., *Wind Turbine Technology: Fundamental Concepts in Wind Turbine Engineeering*, 2nd ed., ASME Press, 2009.

[3] Buhl, M. L. Jr., "A New Empirical Relationship between Thrust Coefficient and Induction Factor for the Turbulent Windmill State," *Technical Report*, National Renewable Energy Laboratory, Golden, CO, 2005.

[4] STAR-CCM+. Ver. 6.02.007, CD-Adapco.

[5] Sanderse, B., van der Pijl, S. P., and Koren, B., "Review of Computational Fluid Dynamics for Wind Turbine Wake Aerodynamics," Wind Energy, Vol. 14, 2011, pp. 799-819.

[6] Snider, M. A.,Poroseva, S. V., "Sensitivity Study of Turbulent Flow Simulations Over a Rotating Disk," AIAA-2012-3146, *42nd AIAA Fluid Dynamics Conference and Exhibit Proceedings*, 2012.

[7] Kaiser, B. E., Poroseva, S. V., Snider, M. A., Hovsapian, R. O., Johnson, E., " Flow Simulation Around a Rim-Driven Wind Turbine And In Its Wake," *58th ASME Turbo Exposition Proceeding*, 2013.

[8] Shih, T.-H., Liou, W.W., Shabbir, A., Yang, Z. and Zhu, J. 1994. "A New k-epsilon Eddy Viscosity Model for High Reynolds Number Turbulent Flows - Model Development and Validation", NASA TM 106721.

[9] Rodi, W., Mansour, N. N., "Low Reynolds Number k-epsilon Modelling with the Aid of Direct Simulation Data." *J. Fluid Mech*, Vol. 250, 1993, pp. 509-529.

[10] Menter, F. R., Rumsey, C. L., "Assesment of Two-Equation Turbulence Models for Transonic Flows," AIAA-94-2343, 25th *AIAA Fluid Dynamics Conference Proceedings*, 1994.

[11] Baker, T. J., "Mesh generation: Art or Science?", *Progress in Aerospace Sciences*, Vol. 41, 2005, pp. 29-63.

[12] Moskalenko, N., Krzysztof R., and Antje, O. "Study of wake effects for offshore wind farm planning." *Modern Electric Power Systems (MEPS), IEEE 2010 Proceedings of the International Symposium*, 2010.

[13] Sørensen, J. N., Mikkelsen, R., and Troldborg, N., "Simulation and modelling of turbulence in wind farms," *Proceedings EWEC 2007*, 2007.

[14] Wilcox, D. C., *Turbulence Modeling for CFD*. 3rd ed., D C W Industries, 2010, pp. 303-306.

[15] Rados, K. G., Prospathopoulos, J. M., Stefanatos, N. C., Politis, E. S., Chaviaropoulos, P. K., and Zervos, A., " CFD Modeling Issues of Wind Turbine Wakes Under Stable Atmospheric Conditions," EU UPWIND # SES6 019945.

[16] Maeda, T., et al. "Wind tunnel study on wind and turbulence intensity profiles in wind turbine wake," *Thermal Science*, Vol. 20, No. 2, 2011, pp. 127-132.

[17] Tachos, N. S., Filios, A. E., and Margarls, D. P., "A Comparative Numerical Study of Four Turbulence Models for the Prediction of Horizontal Axis Wind Turbine Flow," *Mechanical Engineering Science*, Vol. 224, 2010, pp. 1973-1981.