



3D RANS Simulation of NREL Phase-VI and MEXICO Wind Turbines

Irfan Ahmed, Matthias Teich, Martin Lawerenz

► To cite this version:

Irfan Ahmed, Matthias Teich, Martin Lawerenz. 3D RANS Simulation of NREL Phase-VI and MEXICO Wind Turbines. 17th International Symposium on Transport Phenomena and Dynamics of Rotating Machinery (ISROMAC2017), Dec 2017, Maui, United States. hal-02953239

HAL Id: hal-02953239

<https://hal.science/hal-02953239>

Submitted on 29 Sep 2020

HAL is a multi-disciplinary open access archive for the deposit and dissemination of scientific research documents, whether they are published or not. The documents may come from teaching and research institutions in France or abroad, or from public or private research centers.

L'archive ouverte pluridisciplinaire **HAL**, est destinée au dépôt et à la diffusion de documents scientifiques de niveau recherche, publiés ou non, émanant des établissements d'enseignement et de recherche français ou étrangers, des laboratoires publics ou privés.



Distributed under a Creative Commons Attribution 4.0 International License

3D RANS Simulation of NREL Phase-VI and MEXICO Wind Turbines

Dr.-Ing. Irfan Ahmed^{1*}, Dipl.-Ing. Matthias Teich², Univ.-Prof. Dr.-Ing. Martin Lawerenz³



Abstract

In the recent years, efforts have been given to perform experimental investigations on wind turbine models to facilitate a database for a better understanding of the aerodynamic effects of the different design aspects, and benchmark testing of the numerical model of wind turbine flow simulations. The authors present the result of 3D RANS simulations of two test cases, namely the Phase-VI [1] of NREL's measurement campaign conducted under the (Unsteady Aerodynamic Experiments) UAE, and the EU project (Modelled Experiments in Controlled Conditions) MEXICO [2] by the International Energy Association, under the IEA Wind Task 29, Mexnext Phase III. The commercial software FINETM/Turbo is used for this purpose. The results from the simulations are compared to the measurement data available from the experimental investigations. The results will serve as a benchmark test for the numerical code, and will allow a better understanding of the development of flow field structures along the radial positions, specially near the tip region of the blade, thus facilitating improved design philosophies.

Keywords

Horizontal Axis Wind Turbine — Computational Fluid Dynamics — Experimental Investigation

^{1,2,3} Department of Thermal Power Engineering, Section of Turbomachinery, University of Kassel, Germany

*Corresponding author: irfan.ahmed@uni-kassel.de

NOMENCLATURE

ϵ	turbulent dissipation rate
ϑ	opening angle of conical section
λ_2	second eigenvalue of $(S^2 + \Omega^2)$
$\vec{c}$	absolute flow velocity vector
c_z, c_y, c_x	absolute flow velocity components
$\vec{w}$	relative flow velocity vector
k	turbulence kinetic energy
l	length
l_c	chord length
l_{turb}	turbulence length scale
p	static pressure
r	radial distance
x, y, z	Cartesian coordinate points
C_f	skin friction coefficient
C_p	coefficient of pressure
C_N	normal force coefficient
C_T	tangential force coefficient
I	turbulence intensity
M_T	turbine torque
N	rotational speed
R_{tip}	blade radius
S	rate of strain tensor
Ω	rate of rotation tensor

INDICES

0	inlet station
1	upstream station
2	downstream station

INTRODUCTION

With the growing demand on renewable energy conversion, wind energy has undergone a recent boom. This is translated into the increase in the installed power. The modern wind turbines are upscaled in capacity which has resulted in an increase of the dimension of such turbines. Incentives are taken worldwide to increase the yield from the wind energy sections, and national energy policy guidelines, along with market research have attracted increasing involvement of scientific community members in the development the sector.

Such attempts require an understanding of the science of wind energy conversion. The flow around the wind turbine can be predicted with the help of aerodynamic simulations, which is very demanding in computational effort. The grid resolution required to resolve the flow structures for the wind turbine flow using 3D Navier Stokes equations is too high for the currently available computational hardware. The less computationally demanding Reynolds Averaged Navier Stokes solvers are

SUPERSCRIPTS

$\vec{A}$	vector
-----------	--------

to be put to benchmark testing before they can be used in the development of the future wind turbine designs. Recently, efforts have been given to acquire a better understanding of the flow field around the wind turbine. One such extensive measurement program is carried out by the National Renewable Energy Laboratory (NREL) under the Unsteady Aerodynamic Experiments. The Phase-VI [1] of NREL's measurement campaign conducted detailed aerodynamic field measurement for a 2-bladed wind turbine with rotor diameter 10.058 m in the $24.4\text{ m} \times 36.6\text{ m}$ Wind Tunnel of the NASA Ames research center. A more recent experiment is carried out under the EU project MEXICO [2] for a 3-bladed horizontal axis wind turbine with a rotor diameter of 4.5 m in the $9.5\text{ m} \times 9.5\text{ m}$ wind tunnel of the Large Low-speed Facility (LLF) in the German Dutch Wind Tunnel (DNW). Detailed measurements of the aerodynamic flow field around the wind turbine along with the blade surface pressure data are available from the measurement campaign.

It is to be mentioned that, apart from the available field-measurement data, both the test cases mimic different operating conditions of the respective wind turbines. The boundary conditions for the NREL Phase-VI wind turbine are taken for a near-stall operating condition of the wind turbine, whereas, the Mexnext Phase III measurement data are taken for a normal operating condition of the MEXICO wind turbine. Furthermore, field-measurement data at the inflow and the wake regions are available for the MEXICO wind turbine, allowing the option for benchmark testing numerical codes in terms of their ability to resolve secondary flow structures in the flow field.

In the following sections, an effort is made to simulate the aerodynamic flow field around both the aforementioned turbines, and the results are compared to perform a benchmark testing. The simulations are carried out using the commercial software FINETM/Turbo from the company NumecaTM.

1. NUMERICAL METHODS

The aerodynamic flow field is simulated using the finite volume solver Euranus from the commercial software FINETM/Turbo. The field variables are simulated using the Reynolds-Favre averaged Navier Stokes equations. The turbulence properties are modeled using 2-equation model for the turbulence kinetic energy k and turbulent dissipation rate ϵ .

An effort has been given here to test the performance of the high Reynolds number (high-Re) turbulence models in resolving the flow structures of the wind turbine flow field. The high-Re turbulence model [3] employs wall functions to model the near wall region of the wall boundary layer. This allows one to employ a grid discretization allowing lower computational effort.

The solution algorithm employs a time-marching scheme

for resolving the flow field. The numerical scheme is equipped with a number of convergence acceleration methods [4, 5]. A full multigrid approach is used to solve the coarse grid, and successively uses the results to initiate and update the fine grid results. Furthermore, implicit residual smoothing is employed for the Runge-Kutta schemes. In addition to that, local time stepping is used in the time marching scheme. Furthermore, as the flow field Mach number lies within a low subsonic regime, suitable preconditioning [6] is used to accelerate the time marching scheme.

The boundary conditions are selected as to mimic the conditions prevailing during the measurement campaigns documented in [1, 2]. For the numerical simulation of the turbulence conditions, the turbulent kinetic energy k , and turbulent dissipation rate ϵ are prescribed at the inflow. The values are predicted using the turbulence intensity values I , the wind tunnel flow velocity $|\vec{c}_0|$, and turbulence length scale l_{turb} values exerted from the measurement data. The computational domain is defined as periodic. This allows one to model one blade pitch to represent the flow domain. The far-field boundaries are selected as external boundary conditions. At the far-field, static pressure p , and flow velocity components are prescribed. The values are taken from the measured quantities prevailing at the wind tunnel during the measurement campaigns.

2. COMPUTATIONAL DOMAINS

In the following sections, the preparation of the computational domains for both test cases will be registered in brief.

2.1 Test Case I: NREL Phase 6

In a first step, blade geometry from the Phase-VI wind turbine [1] of the Unsteady Aerodynamic Experiments carried out by NREL is used. The Phase-VI measurement campaign dealt with a two-bladed upwind horizontal axis wind turbine. During the experiments, blade surface pressures were measured along with the inflow measurement with the help of blade-mounted 5-hole probe. Additionally, structural loads were measured along with the generator power. The twisted and tapered blade is constructed of *S* 809 airfoil profile developed by the National Renewable Energy Lab [7]. The blade pitch angle was set at 3° , and the cone angle was set as 0° . The inflow velocity of the chosen working point is 10.051 m/s with a pure axial flow. The blade rotational speed was taken as $N = 72.14\text{ rpm}$. Detailed description of the campaign along with the following benchmark testing of different numerical codes can be found in [1, 8, 9, 10].

2.1.1 Computational Setup

The tower geometry was not considered during the simulation. Furthermore, the hub profile was modified. The original Phase-VI turbine had instrumentation on its

hub, which would have disrupted the inflow. Furthermore, these parts are not rotation symmetric, and are thus hard to model in the available grid generation scheme. Instead, a hub profile is taken with cylindrical cross section. The nose section of the hub is taken to be of elliptical shape. The tail section is modeled as a cone with an opening angle of $\vartheta \approx 4^\circ$. The resulting geometry is rotation symmetric. For the computations, one blade pitch is taken to represent the blade passage. The computational domain is stretched around five times the rotor radius up- and downstream of the rotor, and about 1.43 times rotor radius along the spanwise direction.

2.2 Test Case II: MEXICO

Under the EU project MEXICO [2], a 3-bladed horizontal axis wind turbine with a rotor diameter of 4.5 m is investigated in the $9.5\text{ m} \times 9.5\text{ m}$ test section of the LLF in DNW. In this case, the experimental investigations corresponded to a more detailed measurement campaign. Blade surface pressure distributions were measured, along with inflow and wake-field measurement with the help of particle image velocimetry. The blade structural loadings were also recorded. In addition, the generator torque was measured. Furthermore, far-field acoustic measurements were carried out for different combinations of flow conditions and blade configurations. A comprehensive description of the measurement campaign can be found in [11, 12].

2.2.1 Computational Setup

For the numerical simulations, the tower geometry was excluded. The hub profile was modified at the aft section to facilitate a rotation symmetric geometry. The computational domain was stretched by 3.3 times the rotor radius along the inflow, and 5.8 times the rotor radius along the wake region. The far-field is extended by 2.98 times. The dimensions are so selected as to cover the open section of the LLF facility.

2.3 Spatial Discretization

The spatial discretization is carried out with the help of the automatic structured grid generator AutogridTM. Figure 1 depicts the blade skin meshes for the two test cases. Hence, the blades are seen from the top which gives an idea of the blades geometrical twist. The NREL Phase-VI blade is constructed of *S* 809 airfoil profile except in the blade-root region (figure 1a). The MEXICO wind turbine, has a tip section gradually converting from the airfoil section of NACA 6-series airfoil to elliptical profile at the tip (figure 1b). The flow direction is along the z axis for both cases. The NREL turbine's rotation direction is counterclockwise, whereas the MEXICO turbine rotates along the clockwise directions, when observed from the inlet. For both cases, the blade blocks are meshed using an O4H topology [13], with the blade skin resolved through an O block, with 4 H blocks placed surrounding the O grid. The 3D grid is obtained by

stacking such blade-to-blade meshes along the radial direction.

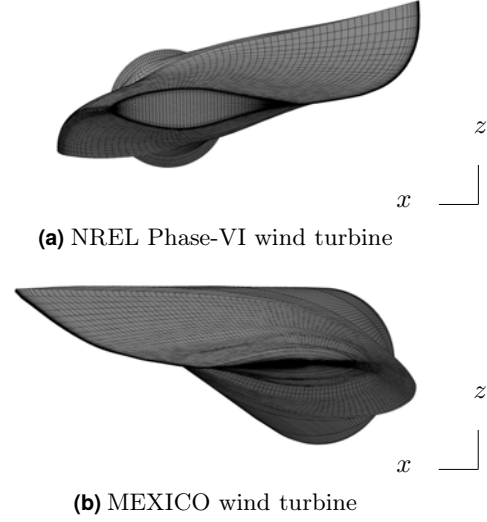


Figure 1. Blade skin mesh for NREL Phase-VI and MEXICO wind turbines

Figure 2 depicts the results of the grid convergence studies for both test cases. Turbine torques are plotted against the grid size. In both cases, the turbine torques M_T are normalized with the values obtained from the finest grid $M_{T,finest}$. Judging the rate of the changes in M_T with grid refinements, the grid sizes of NREL Phase-VI was taken as of $\approx 7.29 \times 10^6$ nodes, and that of the MEXICO wind turbine was taken as of $\approx 6.38 \times 10^6$ nodes. A further refinement in the grid size would have been computationally cost-inefficient for the respective cases. Using the values of M_T , a grid convergence index [14] of 0.004% has been achieved for the NREL Phase-VI turbine, whereas for the MEXICO wind turbine the value grid convergence index achieved is 2.46%.

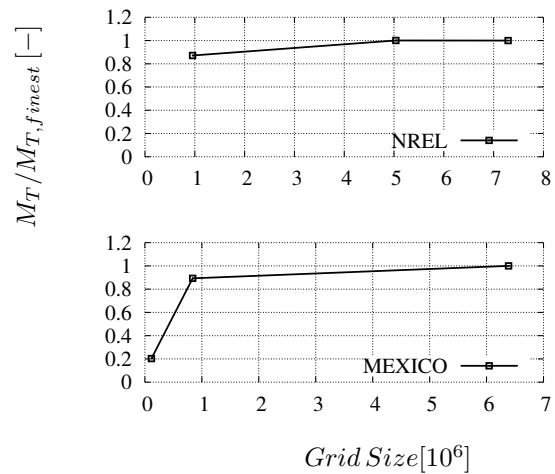


Figure 2. Grid Convergence

3. RESULTS

In the following sections, results obtained from the numerical simulations for both test cases are presented. An effort is made to compare the results from the numerical simulations with those obtained from the measurement campaigns.

3.1 Test Case I: NREL Phase 6

The following section documents the performance of the numerical solver in predicting the flow field around the NREL Phase-VI wind turbine. The results obtained from the numerical simulation are compared to those obtained from the experimental investigations. The 3D flow field around the blade-tip is illustrated with help of streamlines in figure 3. The local flow induction owing to bound circulation can be detected from the shape of the streamlines. This phenomenon changes the local inflow angle, which is generally not predicted in the 2D solution algorithms [15]. Additionally, secondary flow structures can be detected near the blade tip. The presence of the tip vortices disrupts the inflow streamlines, thus creating high aerodynamic losses.

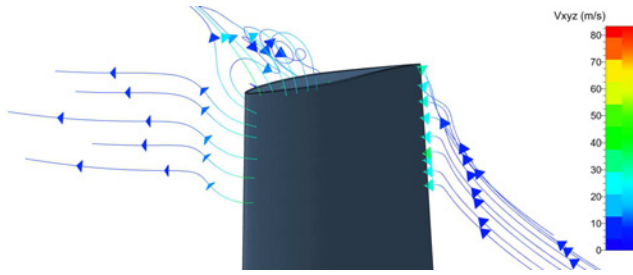


Figure 3. 3D flow near the blade tip

In a further effort, blade pressure contours are plotted along with the isobars in figure 4. Hence, both the pressure (left), and the suction (right) sides are illustrated. The isobars show the rotational effect on the flow. In addition to that, the flow structures around the blade root geometry are also observed to have secondary flow structures. The radial upwash might be originally initiated from the near wall flow at the hub, but the fact that the operating conditions modeled in scope of this study is that of a turbine at the onset of stall, makes it hard to quantify the effect of the radial upwash on the spanwise flow conditions.

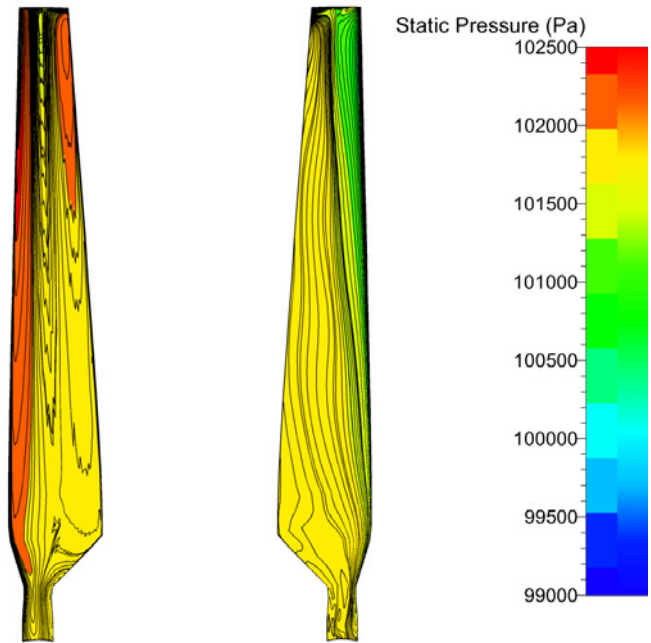
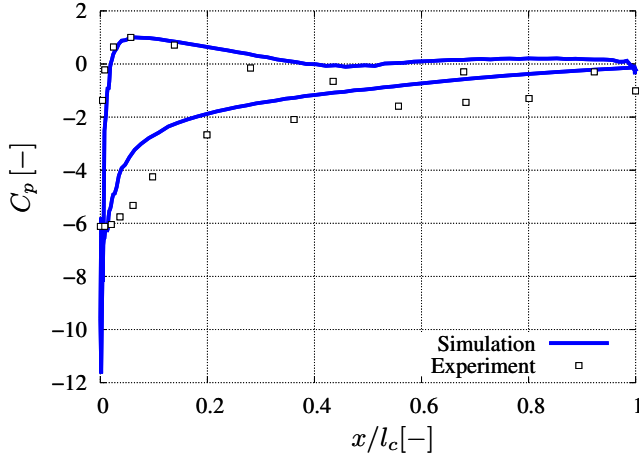
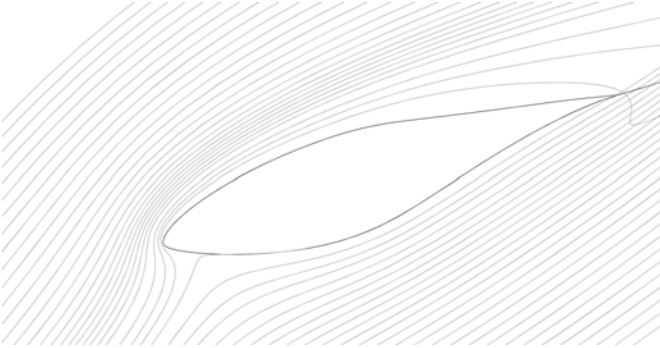


Figure 4. Blade surface pressure distributions

The distributions of the coefficient of pressure along the airfoil suction and pressure sides are depicted in the following sections. The results are extracted for the five radial stations $r/R_{tip} = 0.30, 0.47, 0.63, 0.80$, and 0.95 . For these radial stations surface pressure measurement data are available from the experiments [1]. At the inboard section, ($r/R_{tip} = 0.30$), the coefficient of pressure is over-predicted by the numerical solver compared to the experimental data, as seen in figure 5a. An analysis on the streamlines of the relative velocity figure 5b shows that in the numerical model, flow separation occurs past the mid-chord position along the airfoil suction side. The simulation result indicates towards a different inflow induction, which is projected in the C_p distribution near the leading edge.



(a) Distribution of coefficient of pressure at $r/R_{tip} = 0.30$



(b) Airfoil streamlines at $r/R_{tip} = 0.30$

Figure 5. Flow characteristics at $r/R_{tip} = 0.30$

In a further step, the near wall flow for the next inboard radial station ($r/R_{tip} = 0.47$) is analyzed. The comparison of the distribution of coefficient of pressure is presented in figure 6a, and the streamlines of the relative velocity past the airfoil section are illustrated in figure 6b. During the experiment, the flow near the leading edge of the blade is separated at this radial station [16, 8]. In the simulation, the separation is delayed up to the mid-chord section.

Figure 7 depicts the skin friction coefficient distribution for the suction side along the investigated radial stations. The zoomed view of the distributions near the mid-chord regions are presented in the inset. For the inboard section, transition occurs around half chord length. The trends suggest that, the flow transition point wanders towards the leading edge for the outboard section.

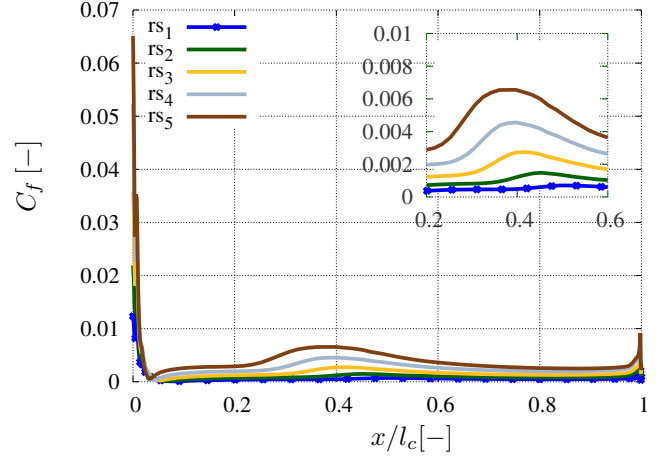


Figure 7. Distributions of blade suction side skin friction coefficient C_f . ($rs_1 : r/R_{tip} = 0.30$, $rs_2 : r/R_{tip} = 0.47$, $rs_3 : r/R_{tip} = 0.63$, $rs_4 : r/R_{tip} = 0.80$, $rs_5 : r/R_{tip} = 0.95$)

The comparison of the C_p distributions for the remaining three radial stations $r/R_{tip} = 0.63$, 0.80 , 0.95 are depicted in figure 8 and in figures 9a and 9b. For these radial stations, the predictions from the numerical simulations are fairly in coherence to the results obtained from the experimental investigations. Also to be noted is that, the numerical model shows a good performance in predicting the distribution of C_p along the both the airfoil suction and pressure sides despite the highly loaded condition the blade is operating under.

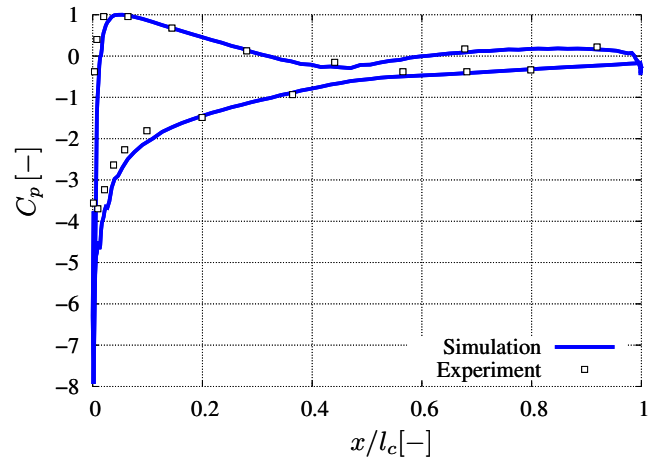
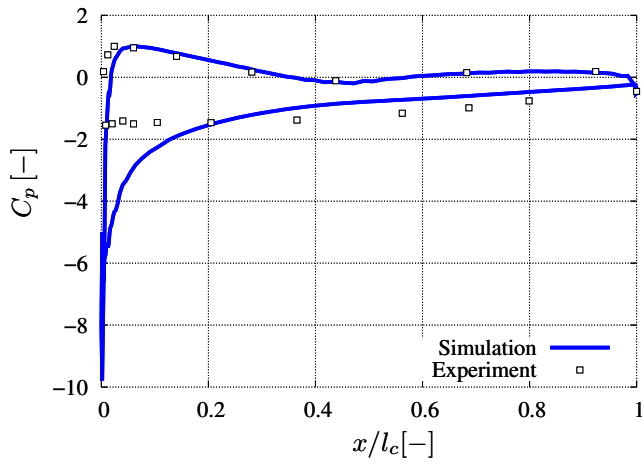
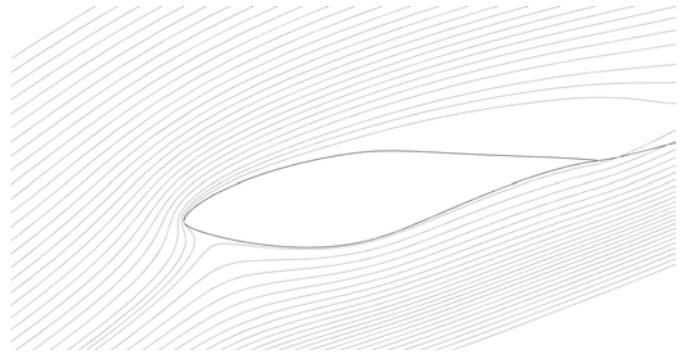
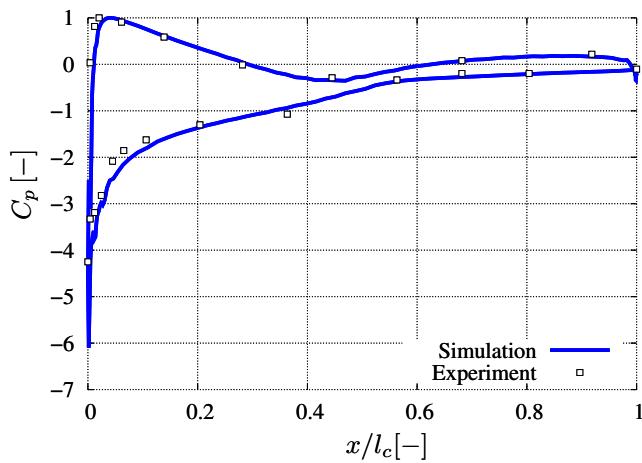
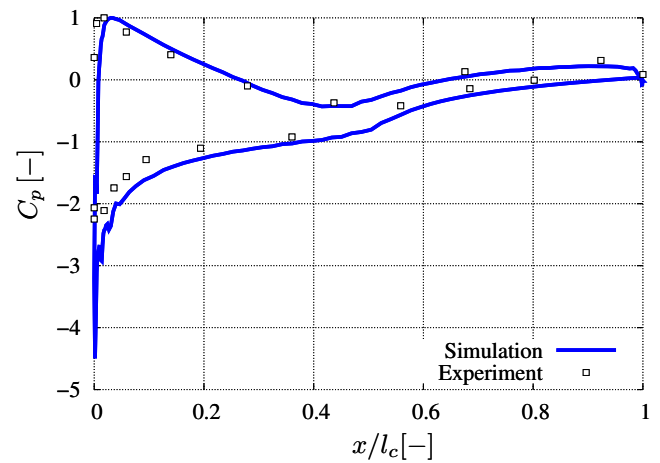


Figure 8. Distribution of coefficient of pressure at $r/R_{tip} = 0.63$

(a) Distribution of coefficient of pressure at $r/R_{tip} = 0.47$ (b) Airfoil streamlines at $r/R_{tip} = 0.47$ **Figure 6.** Flow characteristics at $r/R_{tip} = 0.47$ (a) $r/R_{tip} = 0.80$ (b) $r/R_{tip} = 0.95$ **Figure 9.** Distribution of coefficient of pressure at $r/R_{tip} = 0.80, 0.95$

The distribution of the normal force coefficient is depicted in figure 10. The values are plotted for the five radial positions investigated. At the innermost blade station, large difference is observed between the simulation and the experiment. This can be traced back to the difference in the resolution of airfoil flow-field observed in figures 5a and 5b. The simulation result reproduces the C_N value derived from the experimental data for the blade middle and upper section. At the upper most section, however, simulation result suggests a higher value of C_N than the experimental investigation.

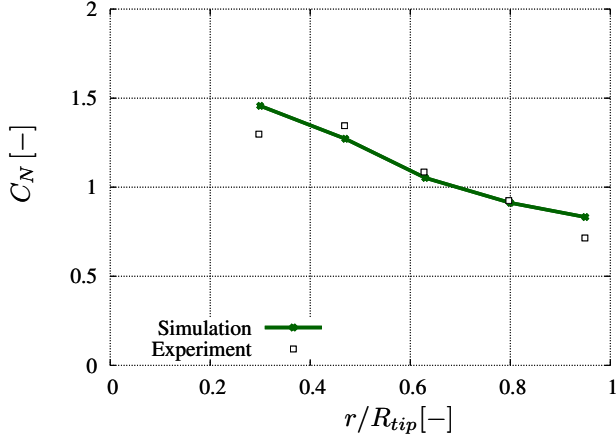


Figure 10. Radial distribution of normal force coefficient C_N

The distribution of the tangential force coefficient for the five radial stations investigated is depicted in figure 11. The values match for the outboard section, whereas for the inboard sections, the values of C_T differs, which can be traced back to the flow characteristics observed in figures 5 and 6.

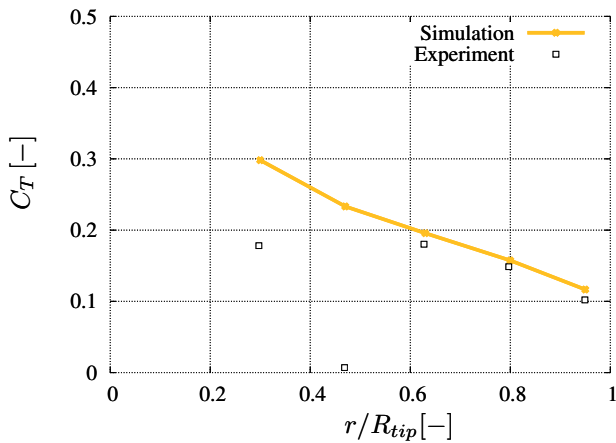


Figure 11. Radial distribution of tangential force coefficient C_T

3.2 Test Case II: MEXICO

In the following section, results from the investigations on the flow field around the MEXICO wind turbine are presented. The test case taken in scope of this study is from the New MEXICO experiment [11]. For the simulation, the inflow velocity was taken as 10 m/s . The blade rotational speed was taken as $N = 425.1 \text{ rpm}$.

Figure 12 depicts the blade surface pressure contours. Additionally, isobars of the blade surface pressure values are plotted. The blade suction side is depicted on the left, and the pressure side is shown on the right hand side. The operating condition simulated herewith was meant for an attached flow condition. The inboard section shows a clean flow pattern.

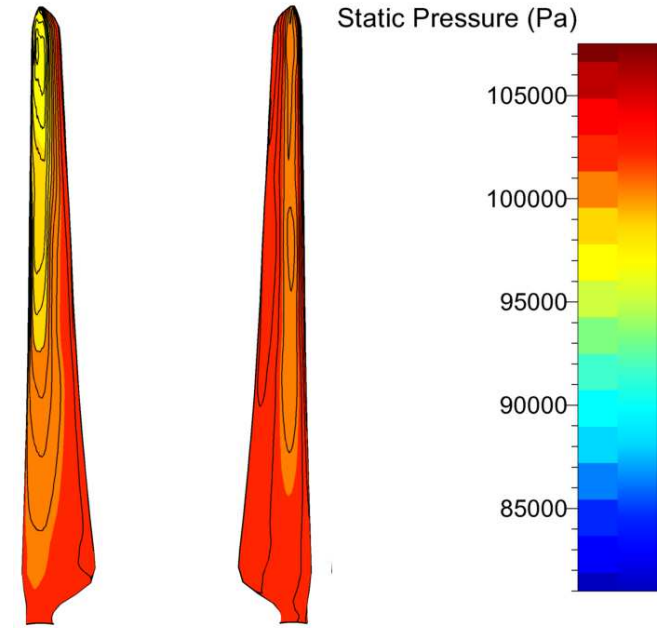


Figure 12. Blade surface pressure distributions

The distributions of the coefficient of pressure along the airfoil suction and pressure sides are depicted in figure 13. During the experiments, blade surface pressure measurements were carried out at five radial stations $r/R_{tip} = 0.25, 0.35, 0.60, 0.82$, and 0.92 . These are compared with the results obtained from the numerical simulations. The inboard section values deviate the most, but it was reported that owing to the uncertainties of the pressure sensors used, the values taken from the experimental investigations are most probably of low quality, and the values from the numerical simulations are rather to be trusted [11, 17].

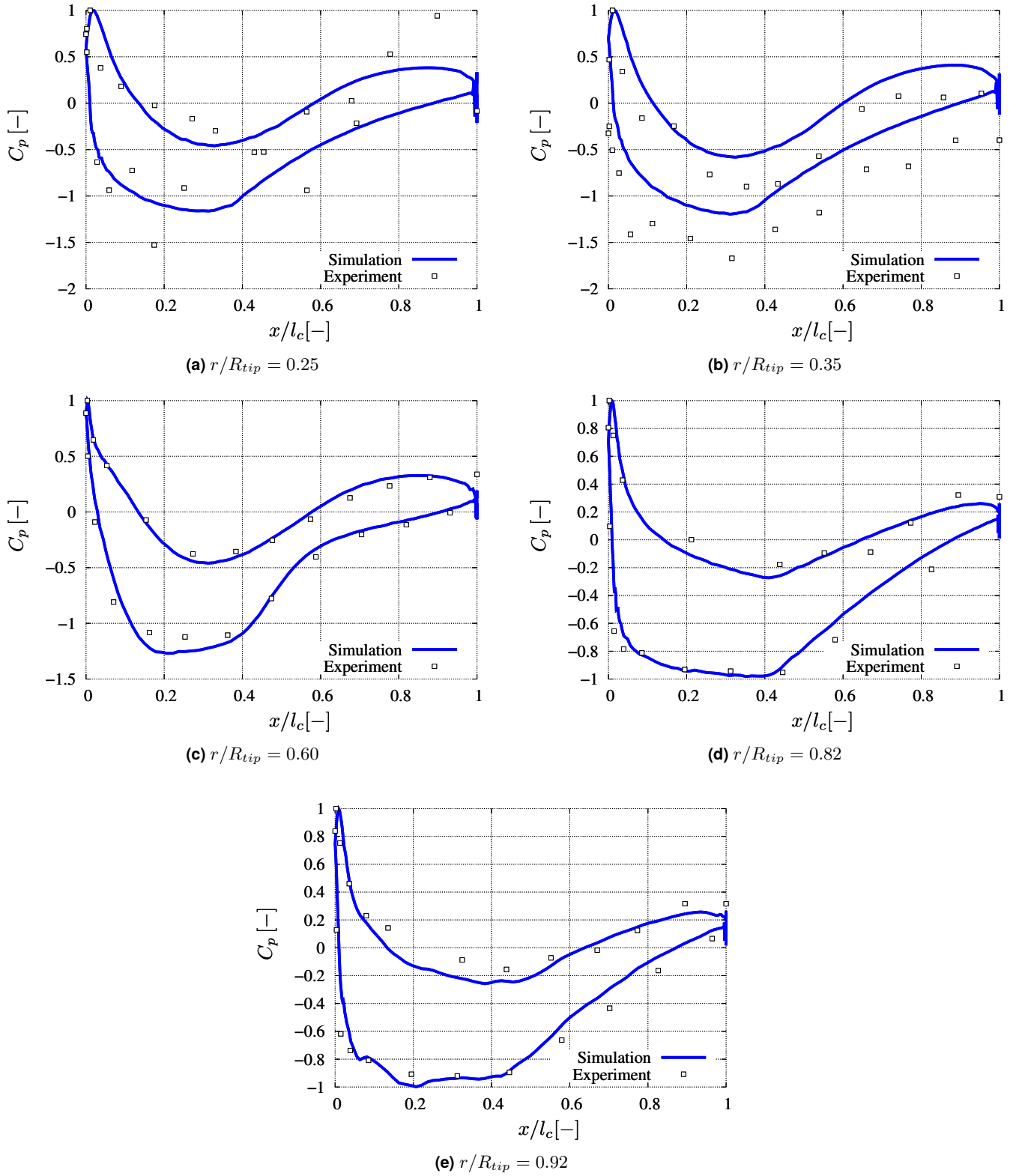


Figure 13. Comparison of coefficient of pressure distributions

In the following sections, an effort is made to compare radial distribution of the flow velocity components at the inflow and the wake of the wind turbine. During the New MEXICO measurement campaign, particle image velocimetry measurements were carried out along planes situated at $0.133 \times R_{tip}$ distance up- and downstream of the rotor plane. The particle image velocimetry measurement planes were traversed along the radial direction. Measurements were carried out for the blades at thirteen azimuthal positions covering 27% of the circumference [11]. The values of these thirteen azimuthal positions are taken and averaged to represent the circumferential averaged radial distributions of the flow velocities. As mentioned before, the simulations were carried out for one blade pitch. The flow field variables are averaged along the pitch direction and the results obtained are depicted in figures 14 to 16.

The comparison of the axial velocity components are shown in figure 14. The simulation predicts a different flow distribution near the tip region. Obviously, numerical performance is affected by employing steady state modeling of the flow field.

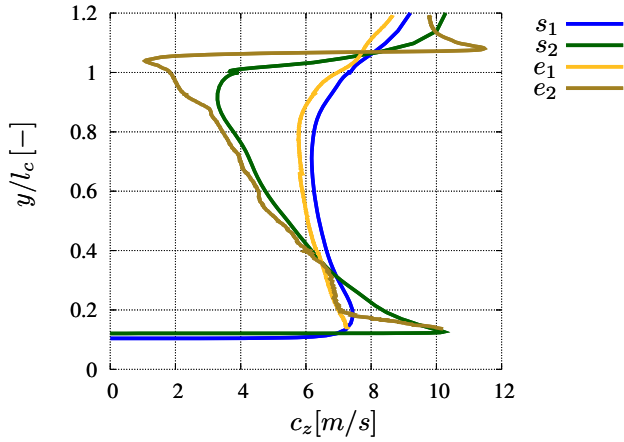


Figure 14. Comparison of axial flow velocity c_z up- and downstream of the blade. (s_k : simulation, e_k : experiment. $k = 1$: inflow, $k = 2$: wake)

Figure 15 illustrates the comparison of the radial flow velocity distributions obtained from the simulation with those from the particle image velocimetry measurements. The values are fairly in agreement between the simulation and the measurement. Although the difference at the wake is still there, it is not as pronounced as for the axial velocity near the tip region.

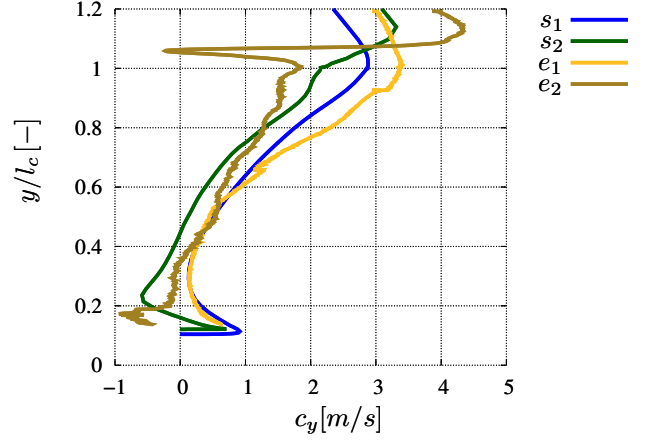


Figure 15. Comparison of radial flow velocity c_y up- and downstream of the blade. (s_k : simulation, e_k : experiment. $k = 1$: inflow, $k = 2$: wake)

The comparison of the circumferential velocity components are shown in figure 16. The flow induction near the leading edge is captured by both the simulation and the experimental investigations. At the wake, however, the values differ over the spanwise extent. The experimental result registered higher flow turning than that found from the simulation. The difference is about 0.4 [m/s] for mid-span.

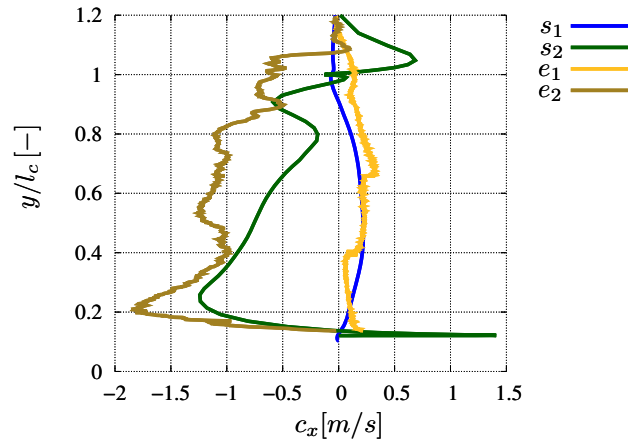


Figure 16. Comparison of circumferential flow velocity c_x up- and downstream of the blade. (s_k : simulation, e_k : experiment. $k = 1$: inflow, $k = 2$: wake)

The findings from the comparisons of the different flow velocities seen in figures 14 to 16 raise a critical question regarding the performance of the numerical modeling, particularly, considering the fact that the simulation results substantially reproduce the measured blade surface pressure distributions (figure 13).

To further understand the flow field predicted in the numerical simulation, the λ_2 criterion [18] is plotted in figure 17 for one blade pitch at about $0.266 \times R_{tip}$ distance aft of the rotor plane. To facilitate the investigation on the flow structures past the blade tip, the domain is extended up to the far field. Hence the λ_2 values of the relative flow velocity $\vec{w}$ is illustrated with the help of contour plot. Additionally, iso-contour lines are plotted to detect the vortical flow structures. The blade is situated in the 90° position. Secondary flow structures are observed along the circumference near the hub. At the inboard section, traces of vortical structures are detected. Presence of trailing vortices are seen originating from the blade inboard section under transition from circular to airfoil profile. Secondary flow structures are also detected near the blade leading edge. Traces of vortical flow structures are also detected in the outboard section. The blade tip shear layer is clearly visible, whose effect extends past the tip radius. The blade tip vortices are dispersed in the 3D domain, giving a region of highly mixed secondary flow structures whose extend is almost 1.5 times the blade tip radius.

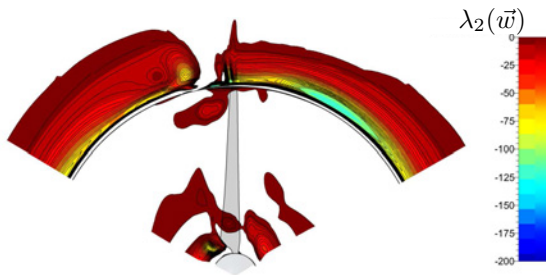


Figure 17. λ_2 criterion of the the relative flow velocity $\vec{w}$ at the wake

4. SUMMARY

The present contribution presents an effort to benchmark the commercial flow solver FINETM/Turbo for the simulation of horizontal axis wind turbine flow using RANS simulation. For this purpose, two state of the art research wind turbines are selected. The flow field turbulence is resolved using high Reynolds number turbulence model. The results give a valuable insight on the performance of the numerical model. For both test cases, the simulation results imitate the results obtained from the experimental investigations with fair accordance. The numerical code predicts the blade surface pressure distributions quite accurately when compared to the results

obtained from the measurement campaigns. Investigations on the NREL Phase-VI turbine shows difference in the blade surface pressure distributions obtained from the simulation and those from the experiment for the inboard sections. The MEXICO test case, being the most recent research project in this field, offers a substantial source of data for testing the numerical code. An effort has been given to compare flow-field velocity components at the inlet and the near wake region of the blade. The comparison between the circumferentially averaged values obtained from the simulation and particle image velocimetry measurements presented a good match for the inflow region. However, notable differences were observed for the values at the wake region. In an effort to better understand the underlying cause of the discrepancies, the near wake plane was investigated further. The investigation shows that, in the numerical simulation, traces of secondary flow structures are detected near the blade at the inboard and outboard sections, which are spread out along the blade pitch. The results suggest that these secondary flow structures were not fully captured by the particle image velocimetry measurements. As such intensive measurement campaign requires huge computational effort, and are cost-intensive, the employment of high fidelity computational method is thus justified. The results obtained from these investigations will play a major role in understanding the physics of the development of the aerodynamic flow field around the blade. Resolution of the near wall flow and the secondary flow structures around the blade would facilitate a better understanding of the aerodynamic loss mechanisms, thus allowing an efficient design of future wind turbines. Further investigations have to be made in this line to deeply investigate the flow structures leading up to and in the wake of the turbine.

5. ACKNOWLEDGMENTS

The authors would like to express their gratitude to the IEA Wind Taskforce 29 for Mexnext Phase III, specially Dr. J. G. Schepers and Dr.ir. K. Boorsma from the Energy Research Center of Netherlands for providing us with the geometry and the measurement data available for the MEXICO test case. A special thanks goes to Mr. Bastian Dose from the University of Oldenburg, Germany, for helping with the issues regarding the MEXICO wind turbine geometry.

REFERENCES

- [1] M. M. Hand, D. A. Simms, L. J. Fingersh, D. W. Jager, J. R. Cotrell, S. Schreck, and S. M. Larwood. Unsteady Aerodynamics Experiment Phase VI: Wind Tunnel Test Configurations and Available Data Campaigns. Technical report, National Renewable Energy Laboratory, 2001.

- [2] J. G. Schepers, K. Boorsma, S. Gomez-Iradi, P. Schaffarczyk, A. Jeromin, T. Lutz, K. Meister, B. Stoevesandt, S. Schreck, D. Micallef, R. Pereira, T. Sant, H. A. Madsen, and N. Sørensen. Final report of IEA Task 29, Mexnext (Phase 1). Analysis of Mexico wind tunnel measurements. (ECN-E-12-004), February 2012.
- [3] B.E. Launders and D.B. Spalding. The Numerical Computation of Turbulent Flow. *Comp. Methods Appl. Mech. Eng.*, 3:269–281–, 1974.
- [4] Charles Hirsch. Numerical Computation of Internal and External Flow, volume 1: Fundamentals of Numerical Discretization. John Wiley & Sons, Oxford, Burlington, USA, 1995.
- [5] NUMECA International. Theoretical Manual FINE™/Turbo v9.1 Flow Integrated Environment. September 2014.
- [6] N. Hakimi. Preconditioning methods for time dependent Navier-Stokes equations. PhD thesis, 1997.
- [7] Design and Experimental Results for the S809 Airfoil. NREL/SR-440-6918, January 1997.
- [8] R.P.J.O.M. van Rooij and E.A.Arens. Analysis of the experimental and computational flow characteristics with respect to the augmented lift phenomenon caused by blade rotation. *Journal of Physics, Conference Series* 75 (2007) 012021, 2007.
- [9] N. N. Sørensen, J. A. Michelsen, and S. Schreck. Navier–Stokes Predictions of the NREL Phase VI Rotor in the NASA Ames 80 ft x120 ft Wind Tunnel. *Wind Energy*, 5:151–169, 2002.
- [10] C. Masson, J. Johansen, N. N. Sørensen, F. Zahle, C. Bak, H.A. Madsen, E. Politis, G. Schepers, K. Lindenburg, H. Snel, R.P.J.O.M. van Rooij, E.A.Arens, G.J.W. van Busse, G.A.M. van Kuik, F. Meng, T. Sant, A. Knauer, G. Moe, X. Munduate, A. Gonzalez, E. Ferrer, S. Gomez, G. Barakos, S. Ivanell, S. Schreck, et al. IEA Wind Annex XX: HAWT Aerodynamics and Models from Wind Tunnel Measurements. NREL/TP-500-43508. December 2008.
- [11] K. Boorsma and J. G. Schepers. New MEXICO experiment. Preliminary overview with initial validation. (ECN-E-14-048), September 2014.
- [12] J. G. Schepers, K. Boorsma, P. Schaffarczyk, H. A. Madsen, T. Lutz, I. Herraez, and S. Schreck. Final report of IEA Wind Task 29: Mexnext (Phase 2). (ECN-E-14-060), December 2014.
- [13] Numeca International. User Manual Auto-Grid™/Turbo v9. September 2014.
- [14] P. J. Roache. Perspective: A Method for Uniform Reporting of Grid Refinement Studies. *Journal of Fluids Engineering*, 106:405–413, September 1994.
- [15] M.O.L. Hansen, J.N. Sørensen, S. Voutsinas, N. Sørensen, and H. A. Madsen. State of the Art in Wind Turbine Aerodynamics and Aeroelasticity. *Progress in Aerospace Sciences*, 42(4):285–330, 2006.
- [16] J. Iwan D. Alexander Ali A. Ameri, Kirsten Ranft and Edmane Envia. Acoustic Analysis of the NREL Phase VI Wind Turbine. In *Proceedings of ASME Turbo Expo 2010: Power for Land, Sea and Air GT2010 June 14-18, 2010, Glasgow, UK, 2010*.
- [17] Abdelhamid Bouhelal, Christian Masson, Arezki Smaili, and Ouahiba Guerri. Evaluation of Various Turbulence Models accuracy for prediction of HAWT Aerodynamics Performance Using New MEXICO Measurements. 1st International Symposium on Wind and Tidal Power, ISWTP2017(008), May 2017.
- [18] Jinhee Jeong and Fazle Hussain. On the identification of a vortex. *Journal of Fluid Mechanics*, 285:69–94, 1 1995.