Comparison of CFD simulations to non-rotating MEXICO blades experiment in the LTT wind tunnel of TUDelft

Ye Zhang, Alexander van Zuijlen, Gerard van Bussel

DUWIND, Faculty of Aerospace Engineering, TUDelft, Kluyverweg 1, 2629HS, Delft, The Netherlands
E-mail: ye.zhang@tudelft.nl

Abstract. In this paper, three dimensional flow over non-rotating MEXICO blades is simulated by CFD methods. The numerical results are compared with the latest MEXICO wind turbine blades measurements obtained in the low speed low turbulence (LTT) wind tunnel of Delft University of Technology. This study aims to validate CFD codes by using these experimental data measured in well controlled conditions. In order to avoid use of wind tunnel corrections, both the blades and the wind tunnel test section are modelled in the simulations. The ability of Menter’s $k - \omega$ shear stress transport (SST) turbulence model is investigated at both attached flow and massively separated flow cases. Steady state Reynolds averaged Navier Stokes (RANS) equations are solved in these computations. The pressure distribution at three measured sections are compared under the conditions of different inflow velocities and a range of angles of attack. The comparison shows that at attached flow condition, good agreement can be obtained for all three airfoil sections. Even with massively separated flow, still fairly good pressure distribution comparison can be found for the DU and NACA airfoil sections, although the RISO section shows poor comparison. At the near stall case, considerable deviations exists on the forward half part of the upper surface for all three sections.

1. Introduction
With the growing size and cost of wind turbines, more reliable and accurate aerodynamic models are needed for the loads and performance prediction. To achieve this goal, in recent years, many high quality experiments have been carried out with the aim of providing a reliable experimental database for validation and improving aerodynamic wind turbine models. One of such the experiments is MEXICO in which a three bladed wind turbine rotor model with a diameter of 4.5 meters is measured in the Large Scale Low Speed (LLF) facility of the German Dutch Wind tunnel Organization (DNW) in 2006 [1]. In this experimental campaign, the loads were measured by instrumented Kulite pressure transducers on the rotating blades. Moreover, a detailed flow field of the wind turbine wake was also obtained by using PIV technology, which makes this experimental measurement as a unique database for validation of numerical models.

However, due to the complexity of the aerodynamics around the rotating wind turbine rotor, many uncertainties have been introduced by rotational effects, blade/nacelle interaction, pressure transducers, blade geometry, etc. These uncertainties lead to some differences between most CFD calculations and experimental measurements in the pressure distribution on the
measured sections, see [2][3][4]. These deviations still cannot be clearly explained. To reduce the uncertainties and get more reliable experimental data, as an intermediate step, each MEXICO blade is measured under non-rotating conditions. This experiment is performed in low speed low turbulence (LTT) wind tunnel of Delft University of Technology. Two test configurations are performed based on the five instrumented sections. One test set-up measures the pressure distribution at 35% spanwise position of blade 1 where the pressure transducers are installed at the inboard section. This set-up exposes the outboard part of the blade outside of the wind tunnel. The other test set-up for blade 2 and blade 3 measures the outboard section in the wind tunnel. All the blades are measured first at clean and then at tripped conditions. Detailed sectional pressure aerodynamic characteristics have been obtained for various inflow wind speeds and a range of angles of attack.

For non-rotating wind turbine blade loads calculation, similar research was done by other authors. Sørensen, et al [5] compared CFD results to experimental pressure distribution and integrated forces for two rotors’ blades during standstill conditions. The results show that in terms of turbulence modeling, the Menter’s $k - \omega$ SST turbulence model gives better loads prediction capability than the original Wilcox’s $k - \omega$ turbulence model. In addition, fully turbulent assumption makes it difficult to predict tangential forces correctly when the flow is fully attached. CENER also investigated MEXICO wind turbine loads calculation by CFD simulations at standstill. The conclusion is the non rotational cases are harder to predict than expected, and the standstill cases are not fully predicted yet due to the uncertainties of the experimental data [2]. As a recommendation, the author states that more CFD computations and turbulence models are needed for better understanding the loads calculation at standstill.

In this paper, first, the MEXICO blade geometry and experimental procedure will be briefly introduced. Then the grid generation and numerical set up are shown in section 2. After that, the comparison between CFD results and experimental data will be shown in section 3.

2. Methodology

2.1. Experimental procedure

The non-rotating MEXICO blades experiment campaign is carried out in the low speed low turbulence (LTT) wind tunnel of Delft University of Technology. This closed return wind tunnel has a test section of 1.80m wide, 1.25m high, and 2.60m long. The free-stream turbulence level in the test section varies from only 0.015% at 20m/s to 0.07% at 75m/s. The maximum velocity of the test section can reach about 120m/s [6]. During this experiment, the inflow velocity varies from 20m/s to 90m/s.

In the experiment, the pressure distribution is measured by 148 Kulite pressure transducers, distributed over five sections on the three blades: on blade 1 at 25% and 35% spanwise sections, on blade 2 of 60% spanwise section and on blade 3 of 82% and 92% spanwise sections. For each tapered blade, there are three different airfoils from the root to the tip. They are the DU91-W2-250 airfoil from 20% to 45.6% span, the RISØ-A1-21 airfoil from 54.4% to 65.6% span and the NACA064-418 from 74.4% span to the tip. The remaining parts of the blade are the transition region connects two adjacent airfoil profiles. Figure 1 shows the airfoil distributions over the span. The chord and twist distributions of the MEXICO blades are also given in figure 2.

Two experimental configurations are tested based on the locations of five instrumented sections. As shown in figure 3, the test set-up of blade 1 measures the pressure at 35% spanwise location with pressure transducers installed in the inboard section. The other test set-up of the blade 2 and blade 3 includes the outboard blade in the whole wind tunnel, see figure 4. The sectional pressure distribution can be obtained from the calibrated Kulite pressure transducers over each section. What is more, the sectional total drag is also measured by a wake rake, which

---

1 National Renewable Energy Center of Spain, one collaboration partner in the MEXICO project
Figure 1. Airfoil distributions over the span of the MEXICO blade.

Figure 2. Chord and twist distributions of the MEXICO blade.

Table 1. Parameters of some cases to be compared

| Blade No. | Measured section | Angle of attack $\alpha[^\circ]$ | Inflow velocity [m/s] | $Re_c$         |
|-----------|------------------|----------------------------------|----------------------|----------------|
| 1         | 35% span         | 8, 15, 19                        | 35                   | $4.6 \times 10^5$ |
| 2         | 60% span         | 8, 15, 19                        | 35                   | $3.4 \times 10^5$ |
| 3         | 92% span         | 8, 15, 19                        | 60                   | $4.0 \times 10^5$ |

measures the static and total pressure in the wake behind the model. To avoid the transition flow from laminar to turbulence over the blade surface, fixed transition was ensured by zigzag stripe placed at 10% chord both on the upper and lower surfaces. The MEXICO blades are measured at different sectional geometric angles of attack $\alpha$ and at different chord based Reynolds numbers $Re_c$ based on the radial section where the pressure transducers are located. Part of obtained experimental data are selected to validate the CFD models, as indicated in table 1.
2.2. Numerical approach

Two computational domains are identified to model the whole CFD geometry, both the blade and the wind tunnel test section. One is a cylindrical domain which includes the blade, and the other is the wind tunnel test section. The blade geometry of the computational model is identical to the one tested in the experiment. Apart from that, to include the blockage effect present in this experiment, the wind tunnel is also exactly modelled in the simulations. In order to set the corresponding geometric angle of attack \( \alpha \) to the measurements, the inner cylindrical domain including the blade is oriented to the desired pitch angles in every computation. The detailed dimensions of the wind tunnel and blades are shown in figure 5 and 6. The distance from the bottom of the wind tunnel test section wall to the measured sections are \( h_1 = 0.7625 \text{m} \), \( h_2 = 1.1 \text{m} \) and \( h_3 = 0.2 \text{m} \), respectively. Different mesh generation strategies are applied for the MEXICO blades. For the first set-up, a multi-block structured mesh with O-grid topology is used for the whole model. In order to solve the boundary layer flow around the blade, the non-dimensional wall distance value \( y^+ \) is below 2 and doing so approximate 20 cell layers exist in the boundary layer. The expansion ratio of these cells is 1.2 in the normal direction towards the outer region. On the other hand, a hybrid mesh is applied for the models of blade 2 and 3. A structured hexahedral grid is generated in the viscous region for resolving the boundary layer flow, and an unstructured tetrahedral grid fills the remaining part of the domain. Boundary layer development on the wind tunnel wall is not taken into account in all these models. Three grid levels are generated for these computations to investigate the grid independency. The schematic of the computational domain and mesh for both two numerical set-ups can be seen in figure 7 and 8, respectively.

![Figure 5. Numerical set-up of the model: MEXICO blade 1](image1)

![Figure 6. Numerical set-up of the models: MEXICO blade 2 and 3](image2)
The CFD solver used in this simulation is OpenFOAM. This code is based on the finite volume method and solves the incompressible RANS. The two equations linear eddy viscosity turbulence model, Mentor's $k-\omega$ SST \cite{8}\cite{9}, is applied for modeling the Reynolds stress $-p_{ij}u_i u_j$. The pressure and velocity are decoupled by the SIMPLE algorithm \cite{10}. A second order linear upwind discretization scheme is used for the convection term, and the diffusion term is discretized by second order central differencing scheme. The iterative tolerance for all solved variables is set below $10^{-7}$. No-slip boundary condition is enforced for the blade surface and slip boundary condition is chosen for the wind tunnel wall. The Dirichlet boundary condition with fixed wind speed is specified at inlet boundary and the outlet boundary is set as a Neumann boundary condition. An arbitrary mesh interface \cite{7} is used between the two adjacent mesh domains.

3. Results and discussion

In this section, a grid refinement study for CFD calculations is presented. Next, the calculated pressure distribution at particular blade sections is compared with experimental measurement for different wind speeds and different angles of attack. Finally, the detailed flow pattern over the blades of CFD results is discussed.

3.1. Grid refinement study

To perform a spatial convergence study, the calculations are carried out for three different grid levels. The detailed grid configurations are listed in table 2. Figure 9 shows the pressure coefficient distribution for the DU section at 35% span of blade 1 with the wind speed $V_\infty = 35m/s$ and angle of attack $\alpha = 8^{\circ}$. As the grid refines, the difference of the pressure distribution for the coarse, medium and fine grid becomes smaller. Therefore, the fine grid is considered to achieve grid independency, and the numerical results shown below are all based on this fine grid.

3.2. Pressure distribution

DU section at 35m/s, 35% span

The comparison of the calculated and measured pressure distribution at 35% spanwise section of blade 1 for three different angles of attack are shown in figure 10. The airfoil profile on this section is the DU airfoil with 25% thickness. In the attached flow case $\alpha = 8^{\circ}$, there is good match between calculated results and measured data except for some locations on the suction side. For instance, the sensor at location around $x/c = 0.05$, this difference can also be seen in some other researchers’ work \cite{2} and \cite{4}. This difference maybe caused by the instability of
Figure 9. Grid convergence study

the pressure transducers. It has to be pointed out that in the old MEXICO experiment, the measured data did not show correct pressures at the inboard sections 25% and 35% span [1]. For $\alpha = 15^\circ$ and $\alpha = 19^\circ$, considerable discrepancy can be observed between $x/c = 0$ and $x/c = 0.4$ on the suction side. The pressure is underpredicted by CFD such that the separation point is delayed a bit compared to the experimental result.

Figure 10. Pressure coefficient comparison at 35% spanwise section of blade 1: $\alpha = 8^\circ$, $15^\circ$ and $19^\circ$ for wind speed $V_{\infty} = 35m/s$

RISØ section at 35m/s, 60% span

In figure 11, the numerical and experimental results are compared at the 60% spanwise section of blade 2. At $\alpha = 8^\circ$, the numerical result, in general, shows a good agreement with experimental data, except the pressures at the locations $x/c = 0.1 \sim 0.5$ which are not adequately captured. For the case $\alpha = 15^\circ$ and $\alpha = 19^\circ$ with massive separation flow, as expected the numerical results show a poor pressure prediction on the upper surface, at the locations $x/c = 0.1 \sim 0.7$ and $x/c = 0.1 \sim 0.6$. In addition, the experimental results show that the separation occurs at around $x/c = 0.4$ and $x/c = 0.3$, while both CFD calculations severely delay the separation point prediction.

NACA section at 60m/s, 92% span

Figure 12 shows the pressure distribution of blade 3 at 92% span for wind speed $V_{\infty} = 60m/s$. For the NACA airfoil profile on this section, at $\alpha = 8^\circ$ with attached flow. There is an excellent

| Grid Characteristic   | Coarse | Medium | Fine  |
|-----------------------|--------|--------|-------|
| Chordwise nodes       | 90     | 120    | 150   |
| Spanwise nodes        | 80     | 100    | 125   |
| First grid spacing(mm)| 0.015  | 0.009  | 0.005 |
| $y^+$                 | 1.5    | 1.0    | 0.5   |
| Maximum skewness      | 1.1    | 0.97   | 1.05  |
| Maximum orthogonality | 65.8   | 65.6   | 66.1  |
| Maximum aspect ratio  | 1675   | 3543   | 4191  |
| Total cells           | 526K   | 874K   | 1452K |

Table 2. Grid configurations
Figure 11. Pressure coefficient comparison at 60% spanwise section of blade 2: $\alpha = 8^\circ$, 15$^\circ$ and 19$^\circ$ for wind speed $V_\infty = 35m/s$

Figure 12. Pressure coefficient comparison at 92% spanwise section of blade 3: $\alpha = 8^\circ$, 15$^\circ$ and 19$^\circ$ for wind speed $V_\infty = 60m/s$

match between CFD result and measured data. For $\alpha = 19^\circ$ post stall case with massively separated flow, $k - \omega$ SST turbulence model provides comparatively good results compared with experimental data, even in the post stall region. As expected, at $\alpha = 15^\circ$, unsteady flow become dominant and considerable difference between numerical result and experimental measurement can be seen on the upper surface at forward half part, which also exists for the DU section and RISØ section at this angle of attack.

3.3. Flow analysis

DU section at 35m/s

Figure 13 shows the time averaged pressure contours and streamlines over the DU section at 35 m/s, for three different angles of attack. As the angle of attack increases, the attached turbulent boundary layer separates due to an adverse pressure gradient, and the region of reversed flow moves forward on the upper surface. At $\alpha = 19^\circ$, one more small vortex appears and sheds backwards compared with $\alpha = 15^\circ$ case. In order to investigate in more detail, the flow behavior at the location $r/R = 0.35$, the limiting streamline on the surface of blade 1 is presented in figure 14. This is the part of the blade near the 35% span. Even though the flow is attached over the whole measured section at $\alpha = 8^\circ$, a clear radial flow near the trailing edge of the airfoil section can be seen at $r/R = 0.35$. On the right side of this blade section, one significant stall
cell can be observed. As the angle of attack increases, the stall cell moves towards to the root part, and the separated flow of the turbulent boundary layer covers most part of the inboard blade.

**RISØ section at 35 m/s**

For the RISO section on blade 2, it should be noted that considerable separated flow region appears on the suction side when the angle of attack is 8°, as can be seen in figure 16. The turbulent boundary layer begins to separate after the chordwise location around \( x/c = 0.6 \). When the angle of attack increases, classical owl’s eyes structured stall cells begin to develop and grow up, from the trailing edge to the leading edge. In figure 15, at \( \alpha = 15^\circ \) and \( \alpha = 19^\circ \), no obvious reversed flow pattern can be observed by the streamline visualization around the airfoil trailing edge as it is observed for the DU section in figure 13. This is due to the measured section \( r/R = 0.35 \) locates near the center of the stall cells, which bring streamwise velocity component along the chordwise direction. Therefore, not much reversed flow pattern can be seen in that region, even though the turbulent boundary layer is still separated.

**NACA section at 60 m/s**

For the NACA section at \( r/R = 0.92 \) span, from figure 17 and 18, the attached flow can be seen on the suction side in the outboard part at \( \alpha = 8^\circ \). At the tip region, the turbulent boundary layer is pushed by the tip effect to the root direction. Strong radial inboard flow can be observed at the trailing edge of the NACA airfoil section. At \( \alpha = 15^\circ \), the mushroom shape of the stall cell begins to develop at the left side of the measured section, and the stall cell grows and moves to the root direction along the blade with the increased angle of attack. At \( \alpha = 19^\circ \), the stall cells are clearly visible.
Figure 15. Pressure contour and streamline over the RISØ airfoil at $\alpha = 8^\circ$, $\alpha = 15^\circ$ and $\alpha = 19^\circ$ of wind speed $35m/s$.

Figure 16. Pressure contour and limiting streamline on the suction side of blade 2 at $\alpha = 8^\circ$, $\alpha = 15^\circ$ and $\alpha = 19^\circ$ of wind speed $35m/s$. The red vertical line indicates the measured section $r/R = 0.60$.

4. Conclusion

The current paper presents the numerical results of CFD simulations for the non-rotating MEXICO blades. Two numerical simulation set-ups, identical to the experiment in the LTT wind tunnel have been carried out. RANS-based simulations combined with Menter’s $k-\omega$ SST turbulence model have been applied for comparing the calculated pressure distribution with experimental data for three different sections of the MEXICO blades, at different inflow velocities and geometric angles of attack.

For attached flow condition, good agreement between CFD calculations and experimental measurement is obtained in the pressure distribution of the DU and NACA sections at $\alpha = 8^\circ$. When there is massive separated flow at $\alpha = 15^\circ$ and $\alpha = 19^\circ$, the pressure at forward half part on the upper surface is under estimated with different levels at all three different sections. Moreover, the prediction at $\alpha = 19^\circ$ shows a better agreement compared with the case $\alpha = 15^\circ$. Overall speaking, at the tip and root region, corresponding to the NACA and DU sections, respectively, the numerical solution is more in agreement with experiments compared to the mid span region which corresponds to the RISØ section.

The limiting streamline shows the detailed flow behavior on the blades. As the angle of attack increases, the turbulent boundary layer begins to separate and stall cells appear. For the DU and NACA sections, these stall cells grow up and moves to the root along the blade, while for the RISØ section, the stall cells seem to just grow up, but without movement. In addition, the streamwise velocity component induced by the stall cells make less reversed flow pattern at the trailing edge of RISØ airfoil.

Future work will focus on the improvement of wind turbine airfoil simulation at high angle...
Figure 17. Pressure contour and streamline over the NACA airfoil at $\alpha = 8^\circ$, $\alpha = 15^\circ$ and $\alpha = 19^\circ$ of wind speed 35m/s

Figure 18. Pressure contour and limiting streamline on the suction side of blade 3 at $\alpha = 8^\circ$, $\alpha = 15^\circ$ and $\alpha = 19^\circ$ of wind speed 60m/s. The red vertical line indicates the measured section $r/R = 0.92$ of attack (beyond stall). Unsteady RANS simulation as well as Large Eddy Simulation (LES) will be performed in order to get better comparison with the experimental data.

Acknowledgments

The author would like to acknowledge Koen Boorsma and Daniel Baldacchino for the experiment’s support. Meanwhile, the financial support of China Scholarship Council is also gratefully acknowledged.

References

[1] Schepers J and Snel H 2007 Model experiments in controlled conditions ECN report
[2] Schepers J, Boorsma K, Cho T, Gomez-Iradi S, Schaffarczyk P, Jeromin A, Shen W, Lutz T, Meister K, Stoevesandt B and others 2013 Final report of IEA Task 29, Mexnext (Phase 1): Analysis of Mexico wind tunnel measurements Wind Energy
[3] Tsalicoglou C, Jafari S, Chokani N and Abhari R S 2013 RANS Computations of MEXICO Rotor in Uniform and Yawed Inflow J. Eng. Gas Turbines Power 136 011202
[4] Bechmann A, Stensen N N and Zahle F 2011 CFD simulations of the MEXICO rotor Wind Energy. 14 677-89
[5] Stensen N N, Johansen J, Conway S, Stensen N N, Johansen J and Conway S 2004 CFD computations of wind turbine blade loads during standstill operation KNOW-BLADE, Task 3.1 report RISO National Laboratory Report
[6] Ferreira C S, Kuik G van, Bussel G van and Scarano F 2009 Visualization by PIV of dynamic stall on a vertical axis wind turbine Exp Fluids 46 97-108
[7] Farrell P E and Maddison J R 2011 Conservative interpolation between volume meshes by local Galerkin projection Computer Methods in Applied Mechanics and Engineering 200 89-100
[8] Menter, F and Esch, T 2001 Elements of Industrial Heat Transfer Prediction 16th Brazilian Congress of Mechanical Engineering (COBEM)

[9] Hellsten A 1998 Some improvements in Menters k-omega SST turbulence model AIAA Paper 98-2554

[10] Patankar S V and Spalding D B 1972 A calculation procedure for heat, mass and momentum transfer in three-dimensional parabolic flows International Journal of Heat and Mass Transfer 15 1787-806