

A Review on Using CFD Technique on Aerodynamic Design of Wind Turbine Rotor Blades

Mohammad Nowfel Mahiuddin

Department of Mechanical Engineering, OPJS University Churu, Rajasthan
420nowfel@gmail.com

Abstract

CFD technique on aerodynamic design of the wind turbine rotor blades is a popular technique which helps in converting the wind speed into mechanical energy. It is based on Navier-Stokes equations. In computational fluid dynamics, we use computers to execute the calculation which is required while simulating the contact of either liquid or gases with surface demarcated by boundary condition. Among the other important aspect in wind turbine are the blades. The blades that are commercially viable in such wind turbines are airfoil shaped cross sections and straight span-wise profile at the same time these blades also functions smoothly when the wind speed is low. In the present paper, we have tried to review the research conducted on wind turbines rotor blades with the help of CFD technique.

Keywords:CFD, Wind turbine blade, Renewable energy.

1. Introduction

When any machine that converts the wind speed into mechanical energy and subsequently getting converted into electricity, it is widely known as Wind turbine. Wind turbine is also known as wind generator, wind power unit, aero generator etc. There are two types of wind turbines. The first one is vertical axis turbine and second type is horizontal axis turbine. Horizontal axis turbine are widely in use these days. The most important element in any wind turbine is its blade. This is wind turbine blade that actually converts the wind energy into kinetic energy. The research on the design of blade of wind turbine also has a significant importance. In the absence of computer, the research on computer was very limited but when computer started getting used in designing of blades, the research has progressed quickly in the recent years. One of the area that get prominence is the aerodynamic performance of wind turbine which can be understood by using computational fluid dynamics.



Fig.1: Horizontal Turbines



Fig.2: Vertical Turbines

Source: <https://govschoolagriculture.com/tag/wind-turbines/>

2. COMPUTATIONAL FLUID DYNAMICS (CFD)

The method that we use to envisage the flow of any gas or liquid with the help of applied mathematics, computational software and physics is known as computational fluid dynamics. It is based on Navier-Stokes equations. In computational fluid dynamics, we use computers to execute the calculation which is required while simulating the contact of either liquid or gases with surface demarcated by boundary condition. Computational Fluid Dynamics (CFD) has developed from a mathematical interest to wind up as a basic instrument in relatively everywhere in Fluid dynamics. The multiple areas where computational dynamics is getting used are Aerospace, Automotive, Chemical Processing, Biomedical, Hydraulics, Power Generation, HVAC, Marine, Oil & Gas etc. From the time when digital computer became popular among the masses, Computational Fluid Dynamics as a developing science became the buzzword among the researchers. Geometry is usually getting used to start any analysis with the help of CFD.

Understanding the geometry is essential for CFD analysis. Geometry in case of CFD is where any gas or liquid will flow. Hence, it is important to outline inlet boundary, outlet boundary and the conditions of far field. In comparison to experimental dynamics, CFD is cheaper, faster and has multiple use whereas is generally used for single purpose. One other advantage is that it can be taken to different places. It is environment friendly also as the computer consumes lesser energy.

3. REVIEW of LITERATURE:

The main objective of the paper is to understand the different types of work undertaken in the field of wind turbine with help of CFD analysis. For this we mainly focused on the research papers available on internet and has been widely read by the people. The time frame for collecting the paper was for 2011-2017.

Malik, et al [1] in their research deliberates on the performance characteristic of twisted turbine blade. CFD modelling of twisted blades are used for analysis. They have done CFD simulation for 2-D airfoils to examine the aerodynamic characteristics. Then they have examined a complete 3-D wind turbine 3 blades rotor with nacelle. They used GAMBIT for geometry and ANSYS FLUENT for performing simulations.

Alaimo et al. [2] believes that Computational Fluid Dynamics is best way to understand the complex aerodynamic flow in the functioning of wind turbine. They studied the impact of diverse geometric features on the precision of simulating a rotating wind turbine. They compared the performance of straight blade vertical wind turbine with a helical blade. They used ANSSYS Fluent software and RANS equations.

According to Zhong [3], the framework used in present research is a unique approach. He used the three-dimensional viscous-inviscid interaction code. This code is MIRAS code which has a very high accuracy. MIRAS was being developed at the Technical University of Denmark. MIRAS is computationally expensive too. The proposed surrogate- based approach however has tried to resolve it. Results show that almost indistinguishable aerodynamic execution can be accomplished utilizing the new plan strategy and that the approach is successful for the aerodynamic outline of wind-turbine rotors.

Alaimo [4] contemplated the impact of various numerical viewpoints on the precision of recreating a turning wind turbine to comprehend the complex aerodynamic stream related to wind turbine working. To simulate the different geometry of wind turbine, the author has seen the effects of a number of elements like rotational velocity structure and size of mesh etc. He further compared two turbines one with vertical axis and another with helical blade.

Mohamed [5] brought into a different aspect in to the study of wind turbines. He states that noise pollution resulting due to wind turbines has a significant importance and hence can not be ignored. The research introduced a new design of the lift VAWTs (vertical axis wind turbines) to reduce the noise emissions. Each blade in the turbine is built by two airfoils. The aerodynamics field of the new plan has been examined numerically to acquire the noise emission from the blades. Unsteady Reynolds-arrived at the midpoint of Navier– Stokes (URANS) conditions are utilized to get the time-precise arrangements. The dispersing between the airfoils in each edge at various tip speed proportion has been considered in this work. The outcomes showed that the 60% spacing is the best arrangement of the double airfoil from the commotion decrease perspective. This new outline lessens the created noise by 56.55%.

O'Brien et al [6] a review of wind turbine aerodynamics research is presented. The review is limited to Horizontal Axis Wind Turbine (HAWT) investigations. The focus is on recent near wake experiments, wake predictions by commercial CFD codes and current FSI and structural modelling attempts. For the near wake, the review considers experiments carried out in controlled conditions whereby the incoming freestream is perpendicular to the rotor plane. Additional anomalies such as wind shear, gusts and yaw transition are not considered. The survey of 3D commercial codes is also focused on HAWT models in parallel flow conditions. Finally, the structural models reviewed are divided into two separate categories: 1) blade deflection and performance under aerodynamic loads, and 2) the vibrational response of blades under aerodynamic loading. The aim is to highlight common trends within near wake experiments and investigate both CFD and FE modelling strategies – to identify current limitations and future opportunities within the sector.

The present work aims to study the aerodynamic characteristics of the NREL phase II (generated only with S809 profile along the span for an untwisted case) rotor that is a horizontal axis downwind wind turbine rotor and which is assumed to stand isolated in the space. The two dimensional steady-incompressible flow Reynolds average NavierStokes equations, are solved by using the commercial CFD package Ansys Fluent. The 2D computations are first performed on S809 airfoil in order to define the most suitable model to be used; the turbulence closure model has been chosen among four possible candidates (standard $k-\epsilon$, Spalart-Allmaras, $k-\omega$ and $k-\omega$ sst) based on comparison of pressure coefficient for the different configurations with experimental results. Secondly, through a three dimensional study we tried to simulate the experiment for wind speed velocities of 7.2, 10.56, 12.85, 16.3, and 9.18 m/s. Results of pressure and torque for considered wind turbine rotor have been directly compared to the available experimental data. The comparisons show that CFD results along with the turbulence model used can predict the span-wise loading of the wind turbine rotor with reasonable agreement. The work presented here is the first stage of project that aims at giving a better understanding of the main influence of the rotational effect on boundary layer separation, and identify the stalled configuration in order to control this latter in future work [7].

Yu, Dong Ok, and Oh Joon Kwon [8] say that the aeroelastic reaction and the airloads of the horizontal-axis wind turbine rotor blades were numerically researched utilizing a coupled CFD– CSD technique. The blade aerodynamic loads were acquired from a Navier– Stokes CFD stream solver in light of unstructured cross sections. The blade elastic disfigurement was computed utilizing a FEM-based CSD solver which utilizes a nonlinear coupled flap-lag-torsion beam hypothesis. The coupling of the CFD and CSD solvers was refined in an approximately coupled way by trading the data between the two solvers at rare interims. At to start with, the present coupled CFD– CSD technique was connected to the NREL 5MW reference wind turbine rotor under steady axial flow conditions, and the mean rotor loads and the static blade deformation were contrasted and other anticipated outcomes. At that point, the shaky blade aerodynamic loads and the dynamic blade reaction because of rotor shaft tilt and tower obstruction were researched, alongside the impact of the gravitational power. It was discovered that due to the aeroelastic blade distortion, the blade aerodynamic loads are

fundamentally decreased, and the precarious dynamic load practices are likewise changed, especially by the torsional deformation. From the observation of the pinnacle impedance, it was likewise discovered that the aerodynamic loads are unexpectedly lessened as the blades go by the tower, bringing about oscillatory blade distortion and vibratory loads, especially in the flap-wise direction.

In another article which presents a wind plant control strategy that optimizes the yaw settings of wind turbines for improved energy production of the whole wind plant by taking into account wake effects. The optimization controller is based on a novel internal parametric model for wake effects called the FLOW Redirection and Induction in Steady-state (FLORIS) model. The FLORIS model predicts the steady-state wake locations and the effective flow velocities at each turbine, and the resulting turbine electrical energy production levels, as a function of the axial induction and the yaw angle of the different rotors. The FLORIS model has a limited number of parameters that are estimated based on turbine electrical power production data. In high-fidelity computational fluid dynamics simulations of a small wind plant, we demonstrate that the optimization control based on the FLORIS model increases the energy production of the wind plant, with a reduction of loads on the turbines as an additional effect [9].

Giahi and Ali [10] in their paper talked about the impacts of wind turbine measure on aerodynamic attributes of a rotor blade utilizing CFD reenactment. It was watched that the consequences of the recreation totally take after the qualities anticipated by Similarity Theory. Both Similarity Theory expectations and reproduction comes about showed that the torque increments with the 3D shape of progress in rotor measurement while the thrust value and aerodynamic powers develop with the square of the adjustment in width.

Ji Yao et al [11] makes the 2D demonstrate for the temperamental numerical simulation of VAWT by FLUENT programming and calculation SIMPLEX joined with the sliding matrix innovation. For numerical simulation, it utilized two turbulence models. They utilize NACA 0018 airfoil arrangement for made the cutting edge 2D show. Area C-H write for CFD examination C is a half crescent shape whose range = 16m and H is a rectangle whose size is = 32m * 30m. In FLUENT they utilized different parameters for examination are Pressure, Velocity, Turbulent Kinetic Energy, Pressure, Velocity Coupling and Grid Generation. They utilize Standard k- ϵ , RNG k- ϵ as Turbulence Model. The outcomes demonstrated that the impact of various turbulence models on the speed field is less, on the weight field is moderately expansive, and on the estimation of the aggregate torque is significantly bigger.

Kaminsky et al., [12] completed the examination of a VAWT utilizing the NACA A0012-34 airfoil. The framework was displayed in Solid Works. The aftereffects of this exploration on the NACA 001234 airfoil demonstrated that it could be an exceptionally reasonable decision for a private VAWT. The 2D investigation gave a slowdown edge of around 8 degrees, in any case, the 3D examination, it is more precise. The consequences of the 3D investigation of vertical axis wind turbine were fragmented.

Rajendran et al. [13] had shown the capability of an incompressible Navier–Stokes CFD technique for the examination of horizontal axis wind turbines. The CFD results are approved against trial information of the NREL control execution testing exercises.

4. Conclusion

From the above audits, we can infer that various analysts are utilizing CFD to examine wind turbine aerodynamic investigation of blade. CFD is broadly utilized for figured the stream examination around the breeze turbine rotor cutting edge (e.g. speed dissemination, weight circulation and so on.) which is influenced by changing breeze speed, approach, tip speed proportion and so on.

CFD has turned into a develop apparatus for anticipating an extensive variety of streams; notwithstanding, one imperative continuous test is the exact portrayal of turbulence.

References

- [1]Malik, Abdul Wahab, et al. "Modeling and Simulation of a Three-Dimensional Adjustable Horizontal Axis Wind Turbine Blade, Using a Commercial Computational Fluid Dynamics (CFD) Code." ASME 2017 International Mechanical Engineering Congress and Exposition. American Society of Mechanical Engineers, (2017).
- [2]Alaimo, Andrea, et al. "3D CFD analysis of a vertical axis wind turbine." *Energies* 8.4 (2015): 3013-3033.
- [3]Zhong, Wen. "Aerodynamic wind-turbine rotor design using surrogate modeling and three-dimensional viscousâ inviscid interaction technique." *Renewable energy* (2016).
- [4]Alaimo, Andrea, et al. "3D CFD analysis of a vertical axis wind turbine." *Energies* 8.4 (2015): 3013-3033.
- [5]Mohamed, M. H. "Reduction of the generated aero-acoustics noise of a vertical axis wind turbine using CFD (Computational Fluid Dynamics) techniques." *Energy* 96 (2016): 531-544.
- [6]O'Brien, J. M., et al. "Horizontal axis wind turbine research: A review of commercial CFD, FE codes and experimental practices." *Progress in Aerospace Sciences* 92 (2017): 1-24.
- [7]Belamadi, R., et al. "CFD study of a horizontal axis wind turbine NREL Phase II." *Revue des Energies Renouvelables* 18.4 (2015): 683-700.
- [8]Yu, Dong Ok, and Oh Joon Kwon. "Predicting wind turbine blade loads and aeroelastic response using a coupled CFD–CSD method." *Renewable Energy* 70 (2014): 184-196
- [9]Gebraad, P. M. O., et al. "Wind plant power optimization through yaw control using a parametric model for wake effects—a CFD simulation study." *Wind Energy* 19.1 (2016): 95-114.
- [10] Giahi, Mohammad Hossein, and Ali Jafarian Dehkordi. "Investigating the influence of dimensional scaling on aerodynamic characteristics of wind turbine using CFD simulation." *Renewable Energy* 97 (2016): 162-168.
- [11] Ji Yao, Jianliang Wang, Weibin-Yuan, Huimin Wang, Liang Cao, "Analysis on the Influence of Turbulence Model Changes to Aerodynamic Performance of Vertical Axis Wind Turbine", ELSEVIER, International Conference on Advances in Computational Modeling and Simulation, Procedia Engineering 31 (2012) 274-281
- [12]Chris Kaminsky, Austin Filush, Paul Kasprzak and Wael Mokhtar, "A CFD Study of Wind Turbine Aerodynamics", Proceedings of the (2012) ASEE North Central Section Conference.
- [13]C. Rajendran, G. Madhu, P.S. Tide, K. Kanthavel, "Aerodynamic Performance Analysis of HAWT Using CFD Technique", European Journal of Scientific Research, ISSN 1450-216X Vol. 65, No. 1 (2011), PP 28-37.