

Computational Analysis of High Lift Generating Airfoils for Diffuser Augmented Wind Turbines

Anhad Singh Bajaj¹, Aniruddha Paranjape¹, Shaheen Thimmiah¹, Raahil Nayak¹, and Jaykrishnan Radhakrishnan¹

1. Department of Aeronautical and Automobile Engineering, Manipal Institute of Technology, Manipal 576104, Karnataka, India.

Correspondence: Aniruddha Paranjape (aniruddha.dp97@gmail.com)

Abstract.

The present study aims to assess the aerodynamic performance of Diffuser Augmented Wind Turbine (DAWT) using high lift generating airfoils in the construction of the shroud/diffuser. The study is a Computational Fluid Dynamics (CFD) analysis which is carried out using Reynolds Averaged Navier-Stokes (RANS) simulations. The flow across the duct and rotor blades, which are modeled as an actuator disk (AD), is analyzed. Various High-Lift generating airfoils and their geometries were taken into consideration and analyzed with additional geometric modifications, such as a flange, to improve flow through the AD and increase the augmentation factor.

1 Introduction

Wind energy is establishing itself as one of the largest sources of renewable energy in the world. In 2018, wind energy production reached approximately 600 GW and wind power accounted for almost 6% of the global electricity demand. Wind energy is the most promising, effective and dependable source of renewable energy. Recent predictions of global warming and reaching the point of no return calls for the entire global community to shift focus into developing effective renewable energy systems. The abundance of wind all across the globe has caused researchers to focus on a system can increase power output without a penalty for the size of the wind turbine. While large wind turbines are placed where the wind topology is optimum, smaller wind turbines are locally built to supply power to meet the demands.

A bare wind turbine modeled after an idealized actuator disk can extract only 59.3% of the kinetic energy of the flow. This is known as the Betz-Joukowski limit as defined by the Betz law of wind turbine performance. This limit however, can be overcome by some additional 'augmentation' devices. The idea of a Shrouded or Diffuser Augmented Wind Turbine (DAWT) was first explored by Lilley and Rainbird [7], where they described power augmentation of small shrouded wind turbines as far back as 1956. Since then, numerous studies have been conducted to investigate and optimize the power augmentation factor for wind turbines through various means. This paper aims to investigate the effects of high-lift generating airfoils on a DAWT at moderate Reynolds numbers. A high-lift airfoil improves the aerodynamic efficiency (C_L/C_D) at low speeds by having a high lift coefficient with minimum drag penalties. The thickness has very little effect in determining the maximum lift coefficient of

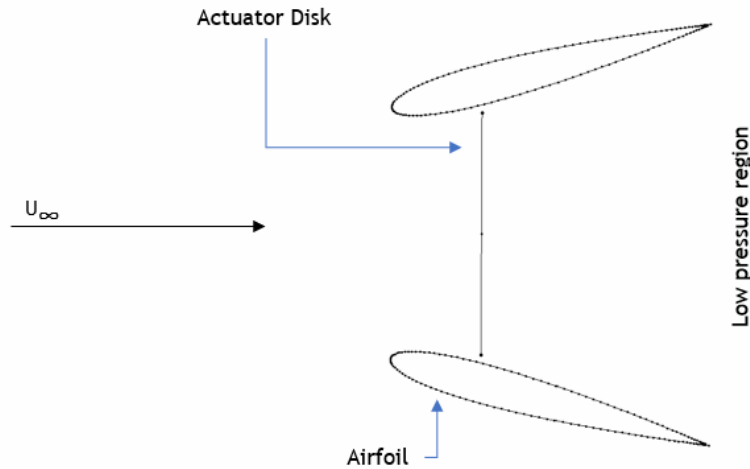


Figure 1. Schematic of the geometry and its orientation to relative velocity U_∞

the airfoils. By adding a diffuser the wake of the turbine blades is allowed to rapidly expand causing a drop in pressure, which in turn leads to an increase in the mass flow rate of the incoming flow. This allows for the power augmentation ratio beyond the Betz-Joukowski limit. Through wind tunnel testing Igra found that power coefficient could be improved by 80% of that of a conventional wind turbine just by placing a diffuser over it [8]. Abe and Ohya, varied with Diffuser Open Angle by adding a flange around the diffuser exit. Flanged Diffusers show that the additional geometric modifications to the shroud can cause a larger wake expansion due to vortices generated at the tips of the flange, as a result of which an enhanced low pressure region is generated in the vicinity [4], which in turn increases the mass flow rate. The loading coefficient of such Flanged Diffusers is much less, compared to that of a bare wind turbine. In a DAWT, maximum energy can be drawn from a high thrust coefficient actuator disk. The thrust can be increased by increasing the expansion ratio. However, this cannot be increased indefinitely as it would cause boundary layer flow separation. However, the boundary layer is re-energized as by adding a flange. Thus there is an optimum configuration for which the velocity through the actuator disk can be maximized. [4,5]

The turbine blades of the Diffuser Augmented Wind Turbine in this present study are modeled as an actuator disk. A pressure drop is induced across this disk which simulates the wake flow for the wind turbine for a specific coefficient of thrust. The entire flow field can be identified as a one dimensional flow field. The separation effects and flow losses from the tips are assumed to be negligible. In this documentation, a basic form of a genetic algorithm is used on a pool of various high lift generating airfoils. The airfoils are categorized as per their family and tests against different parameters such as area ratio, Flange Open

Angle, and position of the flange opening along the airfoil chord length are successively performed to find the best performing airfoils in each class of airfoils. This process was inspired from the work of Dighe et al, who employed a similar technique. [2]

ICEM CFD[®], a module of ANSYS[®] and a commercial Computational Fluid Dynamics (CFD) tool, has been used for meshing as it offers more control over the meshing process. ANSYS Fluent[®], a well known commercial RANS solver, has been used for the computational study of the various shape parameters of the models.

2 Actuator Disk Modeling

The purpose of the rotor is to extract kinetic energy from the velocity that impinges on the blades of the rotor. This causes a reduction in velocity of the across the blades. This change in kinetic energy invariably causes a thrust force in the direction of the incoming flow. There is also a frictional drag component that is created as result of the flow over the blades. The resultant torque created due to the rotational motion of the blades also contributes to the change in the kinetic energy of the flow. Thus wind turbine forces are operated balancing the thrust force, torque of the blades and kinetic energy of the flow. The Actuator disk (AD) method is a tool used to model the forces and moments of the rotor system of the turbine blades. The basic idea of an actuator disk is to replace the rotor blades with a permeable disc of equivalent area. The actuator disk uses the mass and momentum conservation principles to balance the applied forces as compared to the axial and tangential momentum equations that balance the applied forces on the real rotor blades. Thus the actuator disk maybe used to conveniently model and assist the calculation of rotor aerodynamics. [10]

The Actuator Disk (AD) exerts a constant thrust T_A per unit surface. The coefficient is given by [3]:

$$C_{TAD} = \frac{T_A}{\frac{1}{2}\rho U^2 S_a}$$

where, ρ is the fluid density and U is the freestream velocity and S_a is the surface area of the actuator disk. The pressure drop across the AD (Δp) is input as a constant during the simulations and the force $T_A = S_a \times \Delta p$. Thus a constant coefficient of thrust can be maintained across the the AD.

The average AD velocity can be found by integrating the freestream velocity in the X direction over the defined surface area of the AD:

$$U_{AVG} = \frac{1}{S_a} \int \frac{\partial U}{\partial x} dS$$

Using the above results we can define a power coefficient for the duct geometry with an AD of surface area S_a :

$$C_P = \frac{P}{\frac{1}{2}\rho U^2 S_a} = \frac{U_{AVG}}{U} C_{TAD}$$



Figure 2. The genepool of high-lift generating airfoils

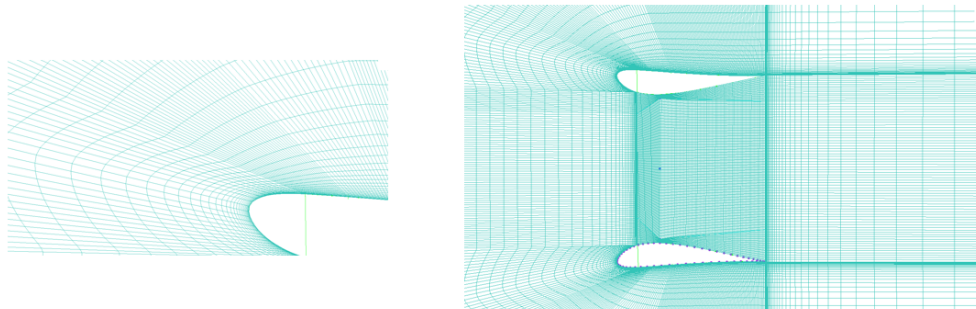


Figure 3. Meshes at the leading edge of the airfoil and at the AD

3 Numerical Approach

3.1 Governing equations

- To properly model the viscous flows over the various diffuser configurations at moderate Reynolds numbers, the Navier–Stokes equations are selected in Cartesian coordinate system. The turbulence model used is $k-\omega$ SST which is expressed as partial differential equations. The $k-\omega$ SST, which was developed by Menter [9] is a robust model for turbulence growth. The $k-\omega$ SST blends the use of $k-\omega$ for near wall flow while using $k-\epsilon$ for the free-stream flow. This makes the model more accurate and reliable yet computationally more feasible as compared to Large Eddy Simulations (LES) or Direct Numerical Simulation (DNS) which are computationally heavy. Although the models are known to often report higher values of turbulence than normal, the discrepancies are known and documented and these errors are small in order.
- 10 The governing equations are as follows[9]:

Kinematic Eddy Viscosity:

$$\nu_T = \frac{a_1 k}{\max(a_1 \omega, SF_2)}$$

Turbulence Kinetic Energy:

$$\frac{\partial k}{\partial t} + U_j \frac{\partial k}{\partial x_j} = P_k - \beta^* k \omega + \frac{\partial}{\partial x_j} \left[(\nu + \sigma_k \nu_T) + \frac{\partial k}{\partial x_j} \right]$$

Specific Dissipation Rate:

$$\frac{\partial \omega}{\partial t} + U_j \frac{\partial \omega}{\partial x_j} = \alpha S^2 - \beta \omega^2 + \frac{\partial}{\partial x_j} \left[(\nu + \sigma_\omega \nu_T) + \frac{\partial \omega}{\partial x_j} \right]$$

3.2 Grid Studies

- A grid validation and a grid analysis study was conducted to verify the accuracy of mesh and results. Three meshes were used with different number of nodes and elements. All the meshes had Selig 1210 as the airfoil and were simulated under similar conditions with an inlet velocity of 6m/s.
- 15

- The first mesh was a very coarse mesh with roughly 4627 nodes and 4776 elements in total and took 129s to converge. As expected, the mesh gave a very poor results with a velocity of only 7.826434m/s at the AD. The second mesh was a fine mesh with a total of roughly 174246 nodes and 175291 elements. This mesh took about ten minutes for the solution to converge.
- 20 This mesh gave a better and more accurate result with a velocity of 8.678735m/s at the AD. The third mesh was an even finer mesh which had a total of 456031 nodes and 457512 mesh elements. This mesh took about 22 minutes for the solution to converge. This mesh gave a velocity of 8.7638m/s at the AD. The finest mesh differed by a 0.98% from the medium quality mesh. Thus the medium quality mesh with 174246 nodes and 175291 elements was chosen as it was accurate with an added advantage of reduced computational time and power.

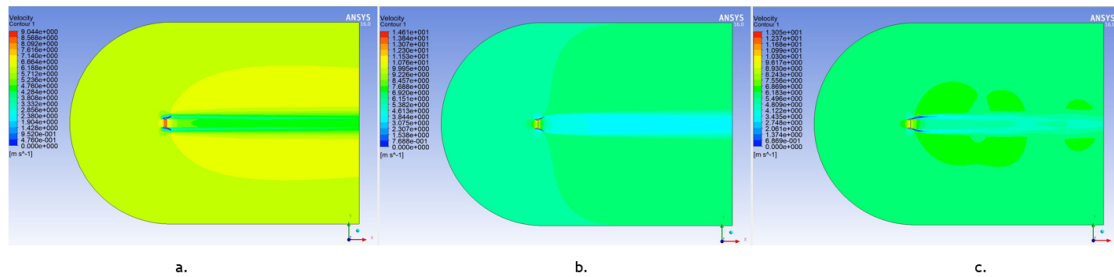


Figure 4. Comparison of the results: a. Coarse b. Medium c. Fine

Grid	Number of Elements	Velocity output (m/s)
Coarse	4776	7.826434
Medium	175291	8.678735
Fine	457512	8.7638

Figure 5. Tabulated Results

4 References

- [1] C. Ghenai, T. Salameh, I. Janajreh: Modeling and Simulation of Shrouded Horizontal Axis Wind Turbine Using RANS Method, Jordan Journal of Mechanical and Industrial Engineering
- [2] V.V. Dighe, G.de Oliveira, F. Avallone, G.J.W. van Bussel: On the effects of the shape of the duct for ducted wind turbines, 2018 Wind Energy Symposium
- [3] V.V. Dighe et al: Towards improving the aerodynamic performance of a ducted wind turbine: A numerical study, 2018 J. Phys.: Conf. Ser. 1037 022016
- [4] Yuji Ohyaa, Takashi Karasudania, Akira Sakuraib, Ken-ichi Abeb, Masahiro Inouec: Development of a shrouded wind turbine with a flanged diffuser, Journal of Wind Engineering and Industrial Aerodynamics 96 (2008) 524–539
- [5] A. Aranake and K. Duraisamy: Aerodynamic optimization of shrouded wind turbines, Wind Energ. (2016) DOI: 10.1002/we.2068
- [6] Aly M. El-Zahaby, A.E. Kabeel, S.S. Elsayed, M.F. Obiaa: CFD analysis of flow fields for shrouded wind turbine’s diffuser model with different flange angles, Alexandria Engineering Journal (2016)
- [7] G.M Lilley, W.J Rainbird: A preliminary report on the design and performance of ducted windmills
- [8] Ozer Igra: Research and Development for Shrouded Wind Turbines,Energy Cont. Momt Vol. 21. pp. 13 to 48. 1981

- [9] F. R. Menter: Two-Equation Eddy-Viscosity Turbulence Models for Engineering Applications, AIAA JOURNAL Vol. 32, No. 8, August 1994
- [10] Mikkelsen, R. F. (2004). Actuator Disc Methods Applied to Wind Turbines. Kgs. Lyngby, Denmark: Technical University of Denmark (DTU). MEK-FM-PHD, No. 2003-02
- 5 [11] Suri, Dhruv. "Design of an Optimized Inlet Shroud for a Flanged Diffuser." (2019)
- [12] Radhakrishnan, Jayakrishnan, and Dhruv Suri. "Design and Optimisation of a Low Reynolds Number Airfoil for Small Horizontal Axis Wind Turbines." IOP Conference Series: Materials Science and Engineering. Vol. 377. No. 1. IOP Publishing, 2018
- [13] Suri, D., Radhakrishnan, J., Nayak, R. (2019). Lattice Boltzmann Method to Analyse Fluid Flow Around a Circular
- 10 Cylinder.
- [14] Dighe, V., Suri, D., Avallone, F., van Bussel, G. Ducted wind turbines in yawed flow: A numerical study.
- [15] Suri, D. (2019). The Potential for Ducted Wind Turbines in an Urban Environment.