Proceedings of the 7th International and 45th National Conference on Fluid Mechanics and Fluid Power (FMFP)
December 10-12, 2018, IIT Bombay, Mumbai, India

FMFP2018–PAPER NO. 321

Design of an Optimized Inlet Shroud for a Flanged Diffuser

Dhruv Suri
Department of Aeronautical and Automobile Engineering
Manipal Institute of Technology
Karnataka, India

Sridhar Idapalapati
School of Mechanical and Aerospace Engineering
Nanyang Technological University
Singapore

Makam Raama
School of Mechanical and Aerospace Engineering
Nanyang Technological University
Singapore

Abstract
Numerical simulation using commercial CFD package ANSYS Fluent ® is carried out for a horizontal axis wind turbine with a flanged diffuser. An optimized inlet shape that further accelerates the flow through the diffuser has been proposed and evaluated. The principle behind the increase in mass flow rate due to the shape of the inlet shroud has been discussed, with emphasis on the modelling techniques presented. The low static pressure aft of the flange at the exit periphery induces a greater mass flow through the diffuser, thereby resulting in a higher capacity factor of the enclosed wind turbine. A comparison between different inlet shroud configurations has also been presented.

Keywords: *Diffuser augmented wind turbine, airborne wind energy; numerical simulation; CFD*

I.INTRODUCTION

Research and development of wind energy has been ongoing for over a century. The objective of extracting the maximum amount of kinetic energy from the prevailing wind while reducing the size of the rotor system has been a complex task. An array of systems and devices [1]–[5] have been designed to increase the capacity factor of a conventional wind turbine. The fundamental principle which a wind turbine operates is the extraction of kinetic energy of the prevailing wind which is further utilized to drive a mechanical shaft. The mechanical energy of rotation of this shaft is converted to electrical power using a generator. The power extracted from a wind turbine is given relation 1.

$$P = \frac{1}{2}\rho A V^3 \quad (1)$$

P is the extracted power, ρ is the freestream density, A represents the rotor area and V is the freestream velocity. According to Betz [6], the maximum kinetic energy that can be extracted from the incumbent air is 59.3 percent of the total energy of the freestream. Conventional rotor systems operate within this limit. The purpose of this study is to provide an optimized design of an inlet flange which leads to an intrinsic increase in the mass flow rate of the air incident on the rotor plane.

Figure 1: Schematic of a conventional horizontal axis wind turbine with a flanged diffuser [7]

Previous studies have been conducted to increase the extracted kinetic energy over the Betz limit. Igra [8] investigated the use of vortex augmented wind turbines, tip vane wind turbines as well as shrouded wind turbines, wherein the rotor system is encompassed in an aerodynamically optimized shroud. The generation of vortices for a vortex augmented wind turbine leads to an increase in the local wind velocity due to a reduction in pressure aft of the flange. This configuration leads to a significant decrease in the static pressure immediately downstream of the system and would lead to a deprecation of the augmented velocity values. Similarly, tip vanes lead to an increase in the induced drag of the rotor blades, thereby nullifying the added increase in local velocity. A wide array of studies such as Gilbert and Foreman [9], Abe et al [10], [11], Foreman et al [12] and Igra have tested for the viability of a shrouded wind turbine. Furthermore, a multitude of numerical approaches have also been presented by Aranake et al [13], Aranake and Duraisamy [14], Ohya et al [15]. These studies rely on relevant modifications to the Blade Element Momentum theory using results obtained by Vaz et al [16], Khamlaj and Rumpfkeil [17], and Hashem et al [18]. This present study employs the results of previous studies and further proposes an optimized shape and geometry of an inlet shroud at the inlet of the tapering diffuser fore body.

The geometry utilized for the present study has been shown in Figure 1. While numerous computational as well as experimental investigations have studies the effect of flange height, diffuser angle, hub ratio and centerbody length, there has been no emphasis placed on the size and shape of the inlet shroud. As determined by Ohya et al [15], the shape of the shroud largely determines the streamlined velocity through the diffuser body. The purpose of this study is to ascertain an optimal geometrical configuration of the inlet shroud.

II. DESIGN PRINCIPLE

Five parameters as investigated by Abe and Ohya [11] with maximal impact on the aerodynamics of the diffuser include diffuser open angle, flange height, hub ratio, centerbody length, and inlet shroud shape primarily affect the performance of the diffuser. The incorporation of a flange by Ohya et al [15] increases the mass flow rate through the diffuser by creating a region of low pressure behind the flange. The aerodynamic theory for diffuser augmented wind turbines has been developed by Igra [8]. The numerical simulation carried out in the present study is based on analytical models validated by previous computational studies and experimental trials.

In the present study, numerical simulation using commercial CFD package has been carried out for a variety of inlet shroud configurations, using baseline geometrical parameters validated by Abe and Ohya [11].

III. METHODOLOGY

The nonlinear eddy viscosity model developed by Abe et al was used to characterize the turbulence intensity. Commercial CFD software FLUENT 18.2 was used to characterize and solve the partial differential equations based on the SIMPLE algorithm.

A. Governing Equations

The governing equations that describe the system are the continuity, momentum and energy equations coupled with a nonlinear eddy viscosity form of the K – epsilon turbulence model expressed by full elliptic Navier Stokes partial differential equations. The flow is assumed to be steady, turbulent, incompressible and two dimensional. Low Mach preconditioning is applied to reach convergence in a faster manner. No slip conditions are accounted for at both the shroud surface as well as the domain boundaries. In equations (2), (3), (4) and (5), u and v represent the components of flow velocity along the x and y axes respectively, P is the freestream pressure, μ_e is the dynamic viscosity of the fluid.

Continuity Equation

$$\frac{\partial(\rho u)}{\partial x} + \frac{\partial(\rho v)}{\partial y} = 0 \quad (2)$$

Momentum Equation

x – Direction

$$\frac{\partial(\rho u^2)}{\partial x} + \frac{\partial(\rho uv)}{\partial y} = -\frac{\partial P}{\partial x} + \frac{\partial}{\partial x}\left[\mu_e \frac{\partial u}{\partial x}\right] + \frac{\partial}{\partial y}\left[\mu_e \frac{\partial u}{\partial y}\right] - \frac{2}{3}\frac{\partial}{\partial x}(\rho K) + \rho g \quad (3)$$

y – Direction

$$\frac{\partial(\rho uv)}{\partial x} + \frac{\partial(\rho v^2)}{\partial y} = -\frac{\partial P}{\partial y} + \frac{\partial}{\partial x}\left[\mu_e \frac{\partial v}{\partial x}\right] + \frac{\partial}{\partial y}\left[\mu_e \frac{\partial v}{\partial y}\right] - \frac{2}{3}\frac{\partial}{\partial y}(\rho K) \quad (4)$$

Energy Equation

$$\frac{\partial(\rho uT)}{\partial x} + \frac{\partial(\rho vT)}{\partial y} = \frac{\partial}{\partial x}\left[\zeta_e \frac{\partial T}{\partial x}\right] + \frac{\partial}{\partial y}\left[\zeta_e \frac{\partial T}{\partial y}\right] + S_T \quad (5)$$

B. Boundary Conditions

The boundary conditions applicable to the present system are shown in Figure 2.

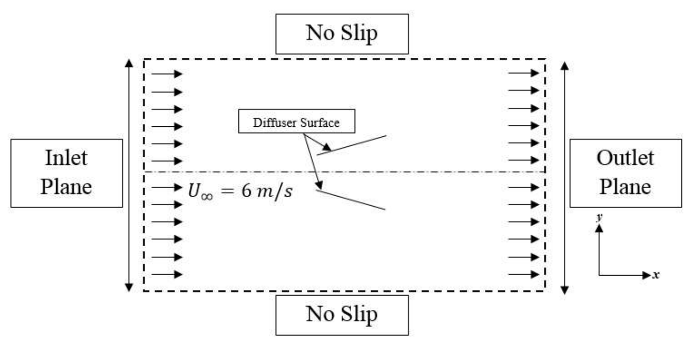

Figure 2: A schematic of the boundary conditions employed

C. Meshing

The domain is discretized by dividing the relevant area into a number of nodes and elements. A structured mesh generation algorithm is utilized with inflation around the proximity and curvature of the diffuser geometry. The

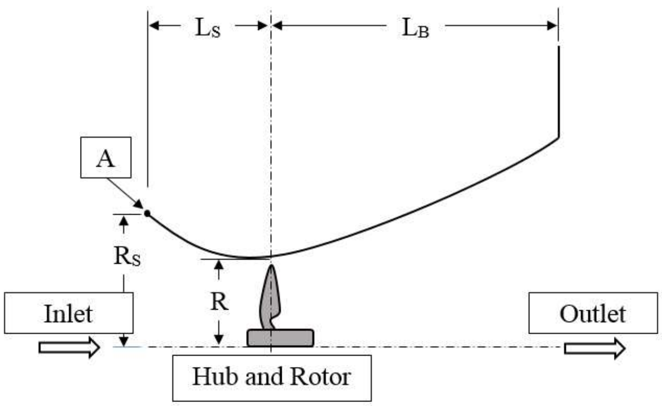

Figure 3: Design variables optimized by the genetic algorithm

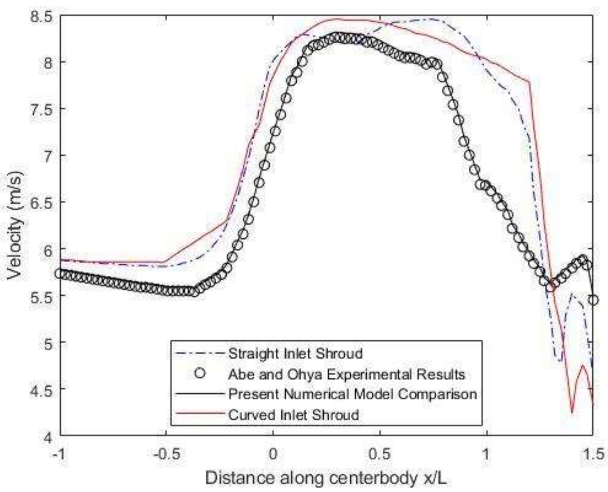

Figure 4: Velocity as a function of non-dimensional distance; comparison of present numerical framework with experimental model of [7]

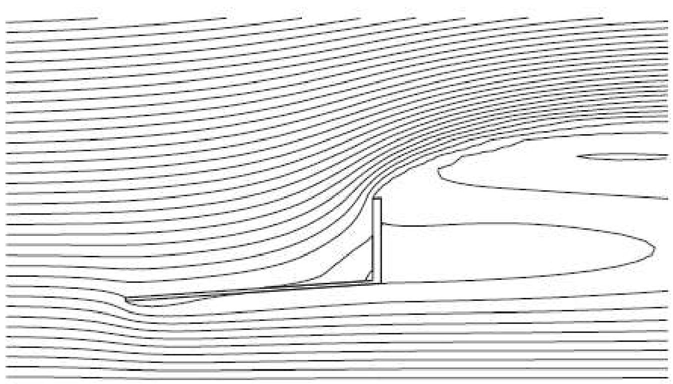

Figure 5: Streamlines of the flow field of a diffuser without any inlet shroud configuration

stream wise direction is sub divided into 320 cells while the radial direction is divided into 132 cells. The number of nodes generated is 490000. The maximum face size of the quadrilateral elements used is 1 mm by 1mm. Grid independence studies were conducted by reducing the size of the quadrilateral elements to 0.90 mm by 0.90 mm, however the element quality showed an increase of 1e-06, and hence grid independence was assumed to be achieved with the aforementioned element size.

D. Fluid Domain

For the numerical simulation, the shroud was enclosed in a two dimensional rectangular domain with two hundred tolerance on either side, measured from the flange. To test for the accuracy of the employed numerical model, a comparison between the experimental results obtained by Abe and Ohya [11] was conducted. The geometrical parameters were consistent with that of the study. The Reynolds number of the flow was set to 120000, diffuser open angle 15 degrees, L/D = 1.25, h/D = 0.35 and diameter of inlet D = 12 cm. A comparison between the non dimensionalised velocity at the centerbody obtained in the present study and the experimental results obtained by Abe and Ohya [11] is shown in Figure 4.

The initial choice of inlet shroud shape was obtained by employing a shape optimization methodology similar to Khamlaj and Rumpfkeil [17], wherein an evolutionary genetic algorithm was used to define an inlet shroud geometry and vary the shroud parameters. Inlet shroud parameters included the curvature defined by a quadratic polynomial, with a moving point A as shown in Figure 3. A reduction in the number of design variables was achieved by using one fourth order Bezier curve to describe the shape of the inlet shroud.

IV. NUMERICAL RESULTS

Following the numerical simulation of the two dimensional models obtained from a semi stochastic, semi deterministic genetic algorithm, it was seen that the maximum augmentation ratio as well as mass flow rate was seen for two geometries: an inlet representing a perfect quarter circle with $R_S/D = 0.37$, and a straight shroud with $R_S/D = 0.53$. These two geometries may be compared to a standard diffuser configuration without any inlet shroud represented in Figure 1.

The numerical simulation provides detailed information about the pressure and velocity distributions at the exit periphery as well as through the centerbody of the diffuser augmented wind turbine. These results help show the effect of the diffuser shape as well as the inlet shroud. Figure 5 shows the velocity contours of the flow around a baseline geometric configuration which was used as a comparison with Abe and Ohya's [7] experimental investigation. The model was found to be in good agreement with experimental values. A detailed view of the optimized geometry for a curved inlet shroud configuration is shown in Figure 3. Figure 4 indicates that an increase in the velocity is seen to be 1.4 times the freestream velocity for an optimized curved inlet shroud configuration and 1.2

times for an optimized straight inlet shroud configuration, as obtained by performing numerical solutions on the models obtained from the genetic algorithm. From the velocity distribution shown in Figure 4, it can be noted that a curved shroud improves performance by further streamlining the flow incident on the rotor plane and further increasing the static pressure at the entry periphery. The increase in pressure is due to a reduction in velocity imposed by an obstruction in the form of a flange. Also, it is noticed that a similar increase in static pressure occurs for a straight inlet shroud configuration, however this configuration does not aid in streamlining the flow towards the enclosed rotor system. To validate the effectiveness of a curved inlet shroud configuration, the velocity contours of an elliptical configuration produced by the genetic algorithm were compared with that of the optimized curved shroud configuration. On further examination of the velocity contours, it was seen that the elliptical configuration helped streamline the flow to the rotor plane but did not increase the static pressure in the periphery of the inlet.

Using the numerical results described in this section, it is possible to draw a conclusion on an optimized shroud geometry for a flanged diffuser.

V. CONCLUSIONS

On the basis of the present numerical study, an optimized inlet shroud configuration has been modelled and the following conclusions are obtained:

1. There is good agreement between the numerical results obtained through the present study and previous experimental trials
2. A curved inlet shroud configuration was seen to be most effective in increasing the augmentation ratio of the structure
3. The optimum inlet shroud shape was determined to be a quarter circle with $R_S/D = 0.37$
4. The genetic algorithm utilized in this study can be further modified to optimize other relevant aerodynamic parameters of the shroud, however the computational complexity of implementation would be high
5. Examination of the velocity contours show that the wake in the case of a curved inlet expands more rapidly as compared to a conventional diffuser augmented wind turbine configuration. This effect can be used to improve the performance of conventional small wind turbines
6. The present configuration also serves to improve the cut in speed characteristics of conventional systems

REFERENCES

[1] T. Kannan, … S. M. of engineering, and undefined 2013, "Design and flow velocity simulation of diffuser augmented wind turbine using CFD," *wind-energy.ucoz.com*.

[2] S. Hjort, H. L.- Energies, and undefined 2014, "A multi-element diffuser augmented wind turbine," *mdpi.com*.

[3] H. Elbakry, A. Attia, O. A.-… (WCST), 2016 World, and undefined 2016, "Simulation of Diffuser Augmented Wind Turbine performance,"

[4] F. B. Juangsa, B. A. Budiman, M. Aziz, and T. A. F. Soelaiman, "Design of an airborne vertical axis wind turbine for low electrical power demands," *Int. J. Energy Environ. Eng.*, vol. 8, no. 4, pp. 293–301, Dec. 2017.

[5] J. Radhakrishnan and D. Suri, "Design and Optimisation of a Low Reynolds Number Airfoil for Small Horizontal Axis Wind Turbines," in *IOP Conference Series: Materials Science and Engineering*, 2018, vol. 377, no. 1.

[6] W. Albring, "A. Betz, Introduction to the Theory of Flow Machines. XVI + 281 S. m. 168 Fig. Oxford/London/Edinburgh/New York/Toronto/Paris/Braunschweig 1966. Pergamon Pres. Preis geb. 75 s. net," *ZAMM - Zeitschrift für Angew. Math. und Mech.*, vol. 47, no. 2, pp. 140–141, 1967.

[7] K. Abe, M. Nishida, A. Sakurai, Y. Ohya, … H. K.-J. of W., and undefined 2005, "Experimental and numerical investigations of flow fields behind a small wind turbine with a flanged diffuser,"

[8] O. I.-E. C. and Management and undefined 1981, "Research and development for shrouded wind turbines,"

[9] B. Gilbert, K. F.-J. of, and undefined 1983, "Experiments with a diffuser-augmented model wind turbine," *… .asmedigitalcollection.asme.org*.

[10]K. Abe, Y. Jang, M. L.-I. J. of H. and Fluid, and undefined 2003, "An investigation of wall-anisotropy expressions and length-scale equations for non-linear eddy-viscosity models,"

[11]K. Abe, Y. O.-J. of wind engineering and industrial, and undefined 2004, "An investigation of flow fields around flanged diffusers using CFD,"

[12]K. Foreman, B. Gilbert, R. O.-S. Energy, and undefined 1978, "Diffuser augmentation of wind turbines,"

[13]A. Aranake, V. Lakshminarayan, K. D.-R. Energy, and undefined 2015, "Computational analysis of shrouded wind turbine configurations using a 3-dimensional RANS solver,"

[14]A. Aranake and K. Duraisamy, "Aerodynamic optimization of shrouded wind turbines," *Wind Energy*, vol. 20, no. 5, pp. 877–889, May 2017.

[15]Y. Ohya, T. Karasudani, A. Sakurai, … K. A.-J. of wind, and undefined 2008, "Development of a shrouded wind turbine with a flanged diffuser,"

[16]D. do Rio, A. Mesquita, … J. V.-E. conversion and, and undefined 2014, "An extension of the blade element momentum method applied to diffuser augmented wind turbines,"

[17]T. Khamlaj, M. R.- Energy, and undefined 2018, "Analysis and optimization of ducted wind turbines,"

[18]I. Hashem, M. Mohamed, A. H.-T. I. C. F. Dyn, and undefined 2016, "Numerical Investigation of Small-Scale Shrouded Wind Turbine with a Brimmed Diffuser,"