Design and Optimization of a Diffuser Augmented Wind Turbine (Wind Lens Turbine) using CFD

Mithil M. Darpe[1], Sharayu K. Bandekar[2], Prof. S. V. Vanjari[3]

[1,2]Student, Dept. of Mechanical Engineering, SSPM College of Engineering, Maharashtra, India
[3]HOD, Dept. of Mechanical Engineering, SSPM College of Engineering, Maharashtra, India

---

Abstract - *For the last few years a great deal of research work has been done in the field of wind energy to improve the performance of small wind turbines for domestic application. In this paper, a new approach of wind turbine is discussed. In the present study, an attempt is made to increase the efficiency of small wind turbine by adding various diffuser configurations and carrying out optimization through CFD. Design and analysis of the wind turbine rotor blades is conducted using QBlade software under general public license which uses the Blade Element Momentum (BEM) method for the simulation of wind turbines, also it is integrated with XFOIL. Validation of result from QBlade is done through CFD and a good agreement has been seen. Then a long type brimmed diffuser with curved and bumped configuration is studied in order to observe the effect on efficiency. Also, CFD analysis for windspeed obtained from optimized curved diffuser for the rotor blade is done to compare the increase in power and torque. SOLIDWORKS is used for the 3D modelling of the rotor blades and diffuser configurations, while the CFD for the same is carried out in ANSYS Fluent.*

Key Words: Diffuser Augmented Wind Turbine, Wind Lens Turbine, Blade design, CFD

1. INTRODUCTION

In order to develop the energy resources for the present as well as future, so many researches have been under implementation where wind energy is found to be emerging and one of the most potential sources to produce energy. Due to combustion of huge amount of fossil fuels worldwide for power production is causing numerous environmental problems such as acid rain, global warming, smog and climate change. In recent years, to reduce the worldwide dependence on fossil fuels, interest in renewable energy technology has been increased.

The wind lens turbine is a modification of the horizontal wind turbine created by Professor Ohya from the Kyushu University as an attempt to be more efficient in production of electricity and less invasive to both humans and nature. As the normal wind turbine does, the wind lens harnesses the energy of the wind but has a few modifications in order to increase efficiency as well as the impact on the environment. Studies have shown that the Wind Lens can have between two and five times more output of power compared to the wind turbine due to the way it harnesses more wind. The diffuser shape and the brim combined creates more efficiently placed and accurate airflow. This results in a higher amount of energy that is produced.

As the power generated by a wind turbine is proportional to the swept area and the third power of the wind speed. Therefore, for effectively increasing the power output we need to enlarge the swept area or increase the wind speed. This can be achieved by adding flanged diffuser as shown in Fig 1. To the wind turbine, thus, generating separations behind it where low-pressure region draws more wind through the rotor blades as compared to a bare wind turbine. This ultimately increases blade rotation and amount of energy generated.

Fig 1. CAD Assembly of Wind Lens Turbine

DAWT was considered to be an interesting topic at the 1979 wind energy innovative system conference. Yuji Ohya and Takashi Karasudani presented a paper [1] in which they developed a new wind turbine system which consists of a diffuser shroud around the wind turbine with a broad brim at the exit periphery i.e. Wind Lens. They started with the experiments on diffuser shaped structure which showed acceleration of wind speed at the entrance and then they carried out experiments on long type diffuser and brimmed short diffuser, where long type diffuser further showed increase in wind speed. Through output performance test, they achieved about 2 to 3 times increase in output power compared to BWT. Hasim A. Heikel in the paper [2] investigated the effect of diffuser flange inclination which was varied from -15° to +15° with a step of 5° and the diffuser flange depth inside at the exit of the diffuser ranging from 0 to 0.06 with a step of 0.02. It was confirmed that due to formation of low-pressure region behind rotor, there was increase in both wind speed and power produced. The maximum percentage increase of both C_p and C_m of 28% was obtained at TSR = 5.6 and diffuser flange angle of +10°. Whereas, decrease in C_p was observed when the diffuser flange depth goes deeper at the diffuser outlet with a reversed flow occurred at the depth of 0.08. Aly M. El-Zahaby presented a paper [3] in which development and analysis of 2D-axis-symmetric CFD model of B-DAWT was done to increase the generated power. This study mainly focuses on the effect of diffuser flange angles which was varied from -25 to 25 with fixed other diffuser parameters. Validation of presence of vortices behind rotor which creates low pressure region and thus, increase in wind speed near diffuser entrance was observed. While at diffuser flange angle of 15, maximum diffuser entrance wind velocity was reached with 1.953 times increase in power generation compared to BWT viz., 5% more than normal flanged diffuser. T. Saravana Kannan in the paper [4] investigated the effect of wind speed on different shapes of flanged diffuser using software like SOLIDWORKS for 3D modelling and ANSYS Fluent for CFD simulation. Total four concepts were analyzed where diffuser opening angle was varied from 4° to 20°; use of splitter within the diffuser to direct and avoid separation and cause vortex inside the diffuser wall; use of solid cone shaped separator to get optimum air pressure at the wake of diffuser and minimize separation of flow in the diffuser and lastly using a bigger diffuser along with cone shaped splitter to ensure smooth flow inside the diffuser. About 61.25% increase in the wind speed was obtained with diffuser configuration having diffuser opening angle of 16° and 0.5m splitter with splitter opening angle of 4°. Peace-Maker Masukume in the paper [5] studied that the optimized plain conical diffuser reduced the cut-in wind speed of the rotor more than that of BWT, and it gives power output about 2.5 times that of BWT. Raju Govindharajan in the paper [6] observed that the increase in wake formation behind the brimmed diffuser create low pressure region behind the turbine which draws more wind flow to strike on rotor blades results in augmentation of torque. S. A. H. Jafari and B. Kosasih in the paper [7] found that with a high area ratio, greater pressure reduction at the exit of the diffuser is achieved with greater degree of separation which can be overcome by using longer diffuser. Peter J. Schubel and Richard J. Crossley presented a review paper [8] which provides detailed knowledge about modern wind turbine blade design. Following points where highlighted: To reduce efficiency losses, it was suggested to avoid low TSR which increases wake rotation and has to be selected between 6 to 10 for three blade HAWT, selection of an aerofoil shape with high lift to drag ratio and need of specialized tip geometries; designing of rotor blades based on BEM method according to Betz limit; XFOIL free software which is generally used to accurately calculate lift to drag ratio; details about chord length and angle of twist distribution in rotor blades and also introduction to aerodynamic control surface concepts like aileron style flaps, camber control, active twist and boundary layer control. Magdi Ragheb and Adam M. Ragheb in the paper [9] presented the detailed derivation of Betz limit and also described an optimal TSR for a wind turbine. From their research, they noted that turbulent wake is formed when a rotor blade passes through the air stream and if at that instant, next blade is passed then it will not be able to extract power from the wind efficiently and also high vibration stresses may induce. Whereas, it the rotor rotates slow, the air hitting each rotor blade would no longer be turbulent, thus, blocking the wind stream. Therefore, for extracting energy from a wind turbine, it is necessary to allow wind stream from high speed to low speed region. Emre KOC in the paper [10] compared the performance of a HAWT with rotor diameter of 2m from QBlade software and CFD results in ANSYS Fluent. 3D CAD model was created using parameters like optimum chord length and twist angle distribution obtained from QBlade software in CATIA for analysis. Further, performance analysis was performed on QBlade software and ANSYS Fluent for obtaining power and torque curves versus TSR graphs for comparison. It was concluded that BEM method which is used in QBlade software for analyzing WT does not take account of turbulence because of which its result where higher than CFD results.

It is possible to predict aerodynamic performance of wind turbine by using experimentation, Computational Fluid Dynamics (CFD) and Blade Element Momentum (BEM) method. In this paper, we have first used BEM method through QBlade software to predict angle of attack, lift and drag coefficients, chord length and twist angle for rotor blade. Then, the 3D CAD model of rotor blade is generated by using SOLIDWORKS by importing the coordinates of the aerofoil and taking the data from QBlade software. For analyzing and calculating aerodynamic performance, CFD method is used by carrying out analysis in ANSYS Fluent. For validation, power and torque of the turbine obtained from QBlade software and CFD results is taken in consideration. Whereas, various diffuser configurations were analyzed using ANSYS Fluent for selecting optimized diffuser for the WLT.

2. METHODOLOGY

2.1. Rotor Blade

For an ideal wind turbine blade design, the following steps are carried out, by referring to various researches:

1. Calculating the rotor diameter for specific power output. (D = 0.9761 m)
2. Choose a design tip speed ratio (TSR) (λ = 5) and number of blades (no. of blades = 3)
3. Selection of the aerofoil type for wind turbine (NACA 63-415)
4. Determination of the angle of attack where $\frac{C_l}{C_d}$ ratio is maximum (α = 8.75°)
5. Divide the blade in sections (sections = 10)
6. Calculating the inflow angle for each section

7. Calculating the chord length for each section
8. Calculating the twist angle for each section

Steps 4 to 8 are done using QBlade software and for calculating rotor diameter below equation is used. Also, for the selected TSR ratio, rotor blade speed and angular velocity is calculated which will be used for CFD analysis. Reynolds number (Re) and angle of relative wind (θ) is calculated which is used as input for design of rotor blades in QBlade software.

Diameter of Rotor Blades (D) –

$$P = \frac{\pi \rho D^2 v3_{\infty} C_P \eta_m \eta_e}{8}$$

Where

P= Power Output of the Wind Turbine = 50 W

ρ = Density Of air = 1.225kg/m^3

D = Rotor Diameter of the wind mill.

v_∞ = Velocity of Air = 6 m/sec

Taking Maxing Possible Efficiency,

c_P= Betz law coefficient = 0.593

η_m= Mechanical Efficiency = 0.9

η_e= Electrical Efficiency = 0.95

Substituting all the above values into equation (1).

We get, D = 0.9761 m

Rotor Blade Speed (N) and Angular Velocity (Ω) –

$$\lambda = \frac{2\pi RN}{v_\infty}$$

Where,

R = Rotor Radius = 0.048 m

λ = Tip Speed Ratio = 5

Therefore, N = 9.7841 RPS = 587.04 RPM

$$\Omega = \frac{2\pi RN}{60}$$

Therefore, Ω = 2.95 m/s

Reynolds Number (Re) and angle of relative wind (θ) –

$$\mathrm{Re} = \frac{\rho v_\infty D}{u}$$

Where,

u = 1.81 * 10^{-5}

Therefore, Re = 394754.25

$$\theta = \tan^{-1}\frac{1}{\lambda}$$

Therefore, θ = 11.3°

Table 1. is obtained from QBlade software showing position, chord length, twist angle and foil.

Numerical simulations for aerodynamic performance of wind lens turbine blades is carried out using ANSYS Fluent. Since the number of blades of the wind turbine is 3, also they are identical and symmetrically arranged at angular distance of 120°, it is enough to analyse

Table 1. Rotor Blade specifications

Sr. no.	Position (m)	Chord length (m)	Twist angle (°)	Foil
1	0	0.02	0	Circular
2	0.048	0.02	0	Circular
3	0.096	0.061866	6.21249	NACA 63-415
4	0.144	0.0481701	0.0684949	NACA 63-415
5	0.192	0.0392491	-2.81858	NACA 63-415
6	0.24	0.0330444	-5.22119	NACA 63-415
7	0.288	0.0285023	-6.9657	NACA 63-415
8	0.336	0.0250425	-8.28768	NACA 63-415
9	0.384	0.0223234	-9.32303	NACA 63-415
10	0432	0.0201321	-10.1554	NACA 63-415

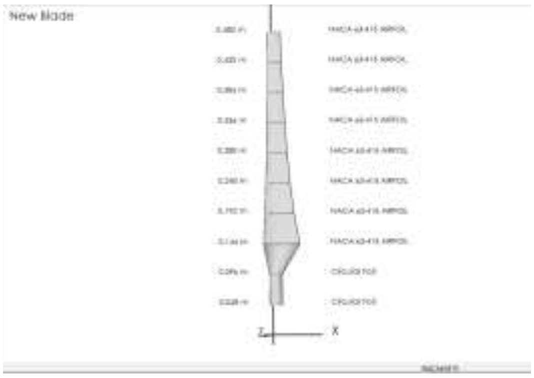


Fig 2. Blade Model from QBlade software

Only one blade with a domain in the shape a sector by creating the interface at 120°. The blade is positioned 0.048m away from the axis of rotation to compensate for the hub. The wind speed considered in the negative Z-direction for analysis is 12 m/s. The relative motion between the blade tip and the flow stream results in a relative velocity which approaches the blade at an angle of 21.1445° which is obtained by trial and error method. A fluid domain is created for the rotor blade with dimensions of inlet and outlet face radius of 3m whereas inlet and outlet face kept at a distance of 1.5m and 3m from rotor blade. A mesh with cell count of 223961 with skewness of 0.97 and orthogonal quality of 0.02216. Bumsuk Kim in a study [11] found that to improve the estimation of wall shear stress in the viscosity sub layer, Wilcox model, BSL (Baseline) model and SST (Shear Stress Transport) model was used where SST model is known for accurately predicting the size of vortex formed behind rotor and also to trace location of separation point due to adverse pressure gradient. So, model is set to viscous SST k-ω was used, while for solution methods, coupled scheme with secondary discretization is set. Also, residual absolute criteria are kept as 1e-06. After initializing the solution, 1000 iterations were calculated.

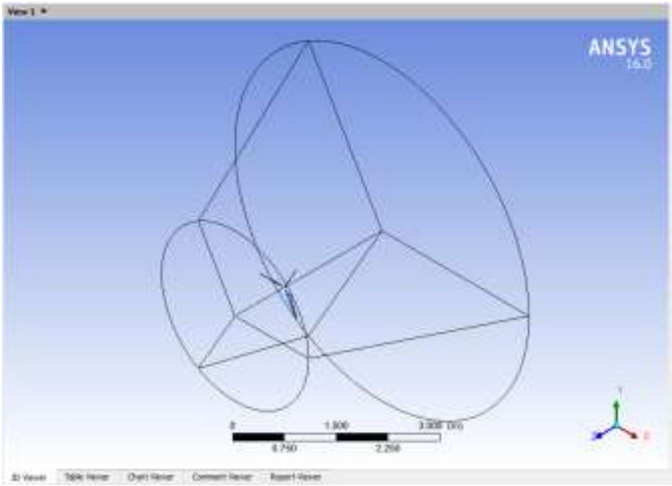


Fig 3. Flow Domain for Blade CFD analysis

2.2. Brimmed Diffuser

Inclusive of conventional straight diffuser there are four types of diffusers – Curved, Stepped and Bumped diffuser configurations. From which, we investigated curved and bumped diffuser with the length of 1.47 times of the diffuser throat diameter and assembled to align with the turbine. Inlet diameter of the diffuser is kept constant for all configurations. Whereas, to investigate the effect om wind speed, brim height is kept as 0.1D and 0.5D at the exit of diffuser and diffuser opening angle varied from 8° to 16° with the step of 4° respectively. Also, throat position is taken as 1/6 and 1/5 relative to wind lens length. Curved and bumped diffuser is shown in fig 4 and fig 5 respectively. And parameters defining the diffuser can be seen in fig 6 whereas its specification are shown in table 2.

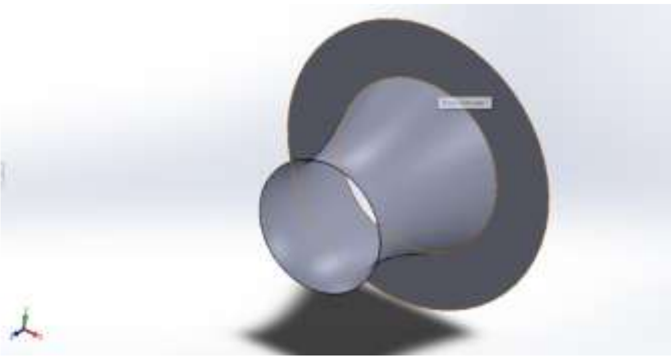


Fig 4. Curved Diffuser

3D CAD model with different configurations were made in SOLIDWORKS whereas analysis is done in ANSYS Fluent. For CFD of diffuser configurations, wind speed is considered as negative 12 m/s in Z-direction with diffuser as a stationary wall. Inlet and outlet face distance from XY-plane is 4m and -2m respectively with enclosure of cylinder with 3 m diameter is made surrounding diffuser. Same as the blade CFD, viscous SST k-ω is selected. In solution methods,

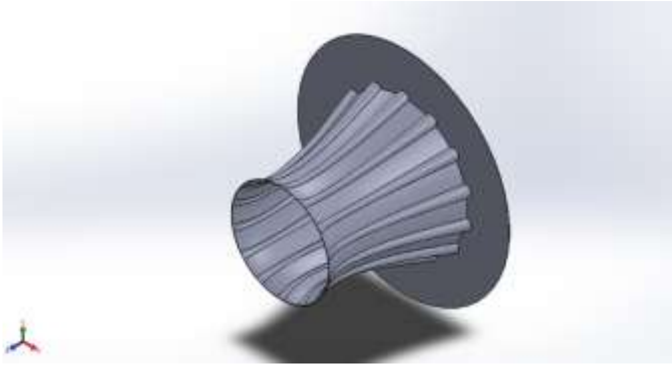


Fig 5. Bumped Diffuser

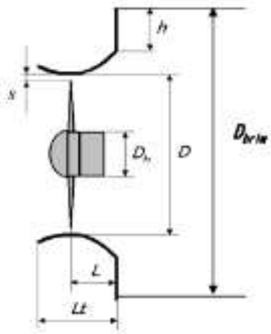


Fig 6. Parameters of Wind Lens Turbine

Table 2. Specification for Diffuser Configurations

Parameters	Dimensions
s	12
Dh	97.6
D_{inlet}	1166.7
D_{rotor}	976
D	1000
L	245 and 294
Lt	1470
h	100 and 500

Scheme is set as SIMPLE with pressure, momentum, turbulent kinetic energy and specific dissipation rate at second order. Then, standard initialization computed from inlet is carried out and result for 600 iterations are calculated.

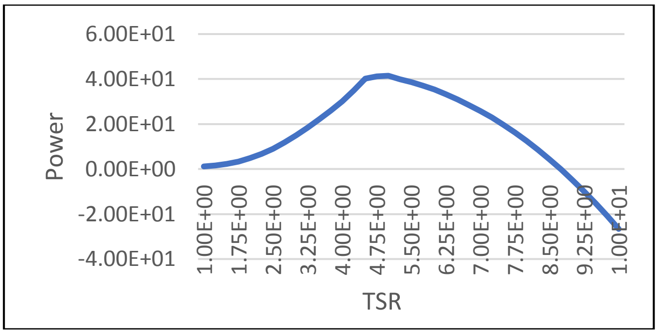
3. RESULTS

3.1. Rotor Blade

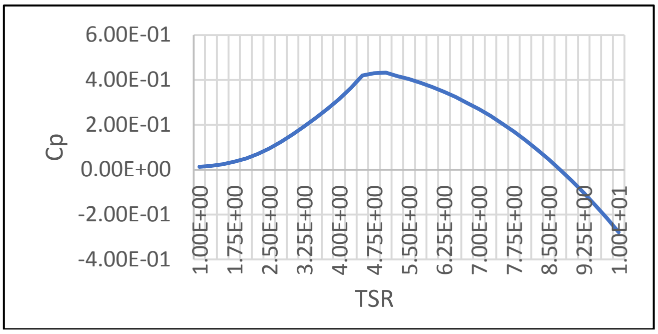
From QBlade and CFD analysis of blade, following results were acquired:
The maximum Cl/Cd ratio of 151.972 is achieved at an angle of attack of 8.75° were the angle was varied from -10° to 20° with airfoil shape taken as NACA 63-415.

Fig 7. NACA 63-415 airfoil shape showing angle of attack

The maximum achievable power coefficient and power output is 0.432 and 41 W respectively, which corresponds to a TSR of 5 and wind speed of 6 m/s which was computed for an ideal rotor by taking into account wake rotation, drag effects and tip losses (Prandtl correction factor). The graph for power and Cp v/s TSR obtained from QBlade software is shown in Graph 1 and Graph 2 respectively.



Graph 1. Power v/s TSR

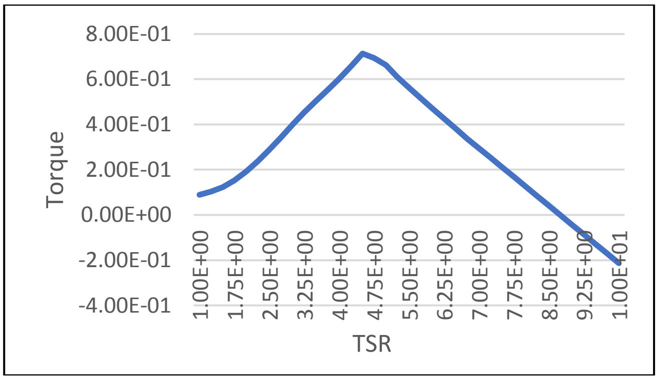


Graph 2. Cp v/s TSR graph

From CFD analysis, calculated Cp of 0.00457 using below equation where T is torque in Nm and power output of 1.835 W is obtained at the wind speed of 12m/s and angle of 20°.

$$C_p = \frac{\text{Power Output}}{\text{Power Input}} = \frac{T \times \Omega}{0.5 \times \rho \times A \times v^3 \times \eta}$$

Whereas, maximum torque obtainable from ideal rotor is about 715Nmm at 6m/s and through CFD analysis it is found to be 607.69Nmm at 12m/s and at an angle of 20°. The graph for torque v/s TSR for an ideal rotor blade obtained from QBlade software is given in Graph 3.



Graph 3. Torque v/s TSR graph

The ϕ with zero θ obtained from QBlade is 21.45°, whereas the same angle obtained from CFD analysis is 20°. Pressure contour for rotor blade which is at 20° angle from the CFD shown in the Fig. 8.

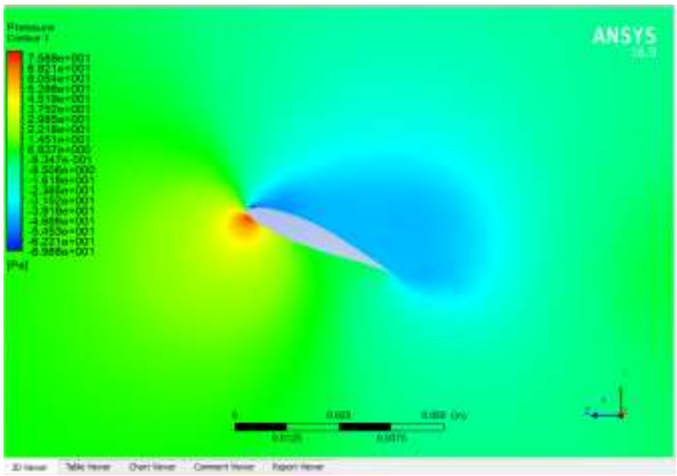


Fig 8. Pressure Contour of Blade Cross-section

3.2. Brimmed Diffuser

From CFD analysis of brimmed diffuser with different configurations, following observations were made:

It is observed that the presence of diffuser induces wake formation which in turn reduces the pressure behind the wind turbine and the strength of it can be augmented by adding a brim at the rear end of the diffuser. The bumped diffuser with 0.5D brim height, pronounced more low-pressure region.

It is found that with a diffuser opening angle of 16°, the wind speed increased most significantly by almost thrice. This is because of the brim combined with diffuser opening angle block 'back-flow' from outer wall of the diffuser while the inlet shroud direct better streamlines of the air flow through the diffuser.

For the throat position of 1/6 relative to the diffuser length, greater mass flow is observed.

Also, curved diffuser with brim performed better when compared to bumped diffuser, whereas the difference between the increase in wind speed for both the diffusers is not much. Pressure contour and velocity vector for optimized curved and bumped diffuser is shown in Fig. 9 and Fig. 10 & Fig. 11 and Fig.12 respectively.

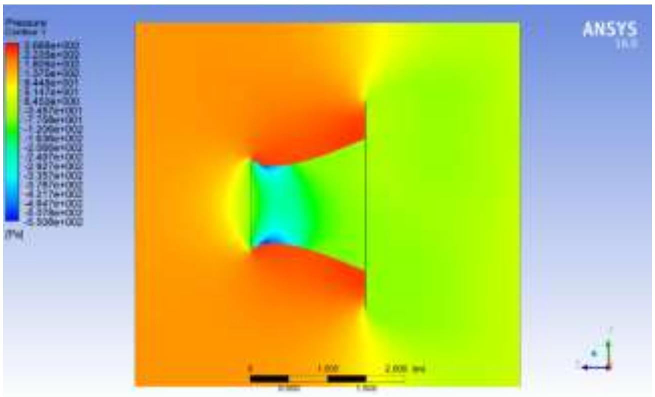


Fig 9. Pressure contour for Brimmed Curved Diffuser

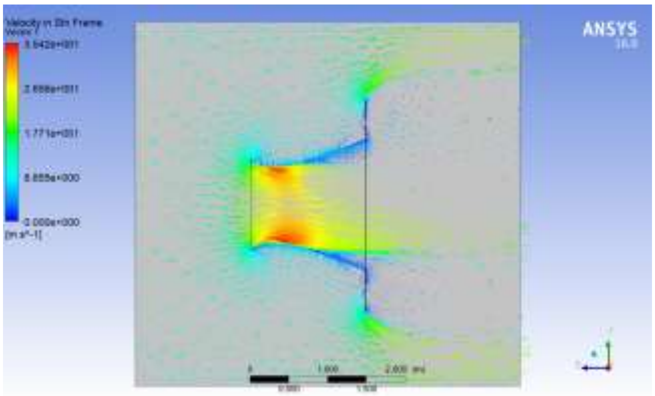


Fig 10. Velocity vector for Brimmed Curved Diffuser

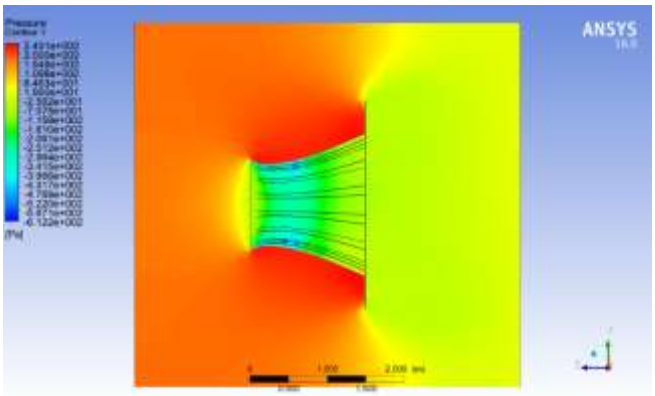


Fig 11. Pressure contour for Brimmed Bumped Diffuser

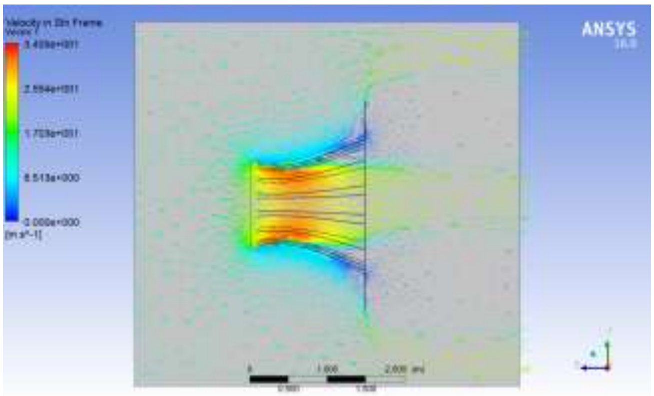
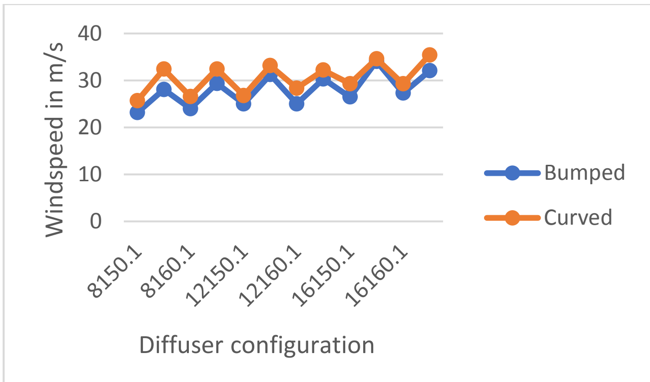


Fig 12. Velocity Vector for Brimmed Bumped Diffuser

From the pressure contour, low pressure regions can be observed behind the brim and significant increase in wind speed near the entrance of the diffuser specifically at the throat where wind turbine blades will be placed can be seen through the velocity vector.

The graph for windspeed at entrance of each diffuser configuration is shown in graph 4 where for instance 8150.1 resembles diffuser opening angle of 8°, at throat distance of 1/5th relative to diffuser length and brim height of 0.1 times to throat diameter.



Graph 4. Windspeed trend in every diffuser configuration

From the CFD analysis of rotor blade done at the average windspeed of 30m/s obtained from optimized brimmed curved diffuser, we get the calculated power output of 12.382W which is about 6.747 times and torque of 1.3667Nm.

4. CONCLUSION

The aim of the paper was to define an optimum rotor blade design and diffuser parameters at a certain design TSR for maximum power output. The numerical investigation focused on the aerodynamic effects contributing to torque as well as power which eventually improved the performance levels due to addition of a diffuser. Corresponding to QBlade data, CFD results were in good agreement with it. Though the power and torque values obtained from QBlade software were higher than the CFD results because BEM theory does not take account of turbulence effects. It was observed that the wind speed in the diffuser was greatly influenced by the opening angle, throat position and brim height with increase of about 2 to 3 times the inlet velocity. Results show that, a remarkable 35.428m/s wind speed which is about thrice the inlet velocity of 12m/s was obtained using curved diffuser with opening angle of 16°, throat position at 1/6th relative to diffuser length and brim height of 0.5 times the throat diameter. From the velocity contour of optimized curved diffuser shown in fig 13, it is evident that velocity contours were occupying the mid-area of the diffuser. Also, the pressure developed at the bottom is significantly higher than

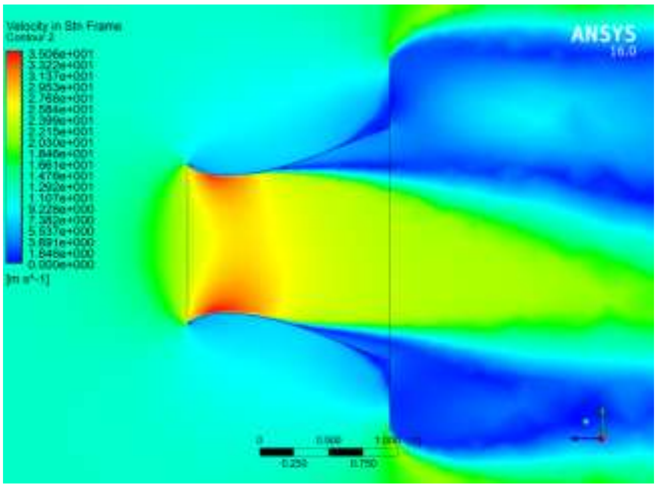


Fig 13. Velocity contour for optimized curved diffuser

the top side of the rotor blade in both CFD results for rotor blade at 12m/s and 30m/s. However, in the case of rotor blade with inlet velocity of 30m/s, the pressure difference is much higher resulting in 6.747 times increase in power output.

REFERENCES

[1] Yuji Ohya, Takashi Karasudani, "A Shrouded Wind Turbine generating high output power with Wind Lens technology", Energies, ISSN 1996-1073, 31March 2010

[2] Hasim A. Heikal, Osayed S. M. Abu-Elyazeed, Mohamed A. A. Nawar and Youssef A. Attai, "On the actual power coefficient by theoretical developing of the diffuser flange of Wind Lens Turbine", Renewable Energy, DOI: 10.10.16/j.renene.2018.02.100

[3] Aly M. El-Zahaby. A. E. Kabeel, S. S. Elsayed and M. F. Obiaa, "CFD analysis of flow fields for Shrouded Wind Turbine's diffuser model with different flange angles", Alexandria Engineering Journal (2017) 56, 171-179

[4] T. Saravanna Kannan, Saad A. Mutasher and Y. H. Kenny Lali, "Design and flow velocity simulation of Diffuser Augmented Wind Turbine using CFD", Journal of Engineering Science and Technology, Vol. 8, No. 4 (2013) 372-384

[5] Peace-Maker Masukame, Golden Makaka and David Tinarwo, "Techno economics analysis of Ducted Wind Turbines and their slow acceptance on the market", Journal of Renewable Energy, Vol.2014, DOI: 10.1155/2014/951379

[6] Raju Govindharajan, Dr. K. M. Parammasiuam, V. Vivek and R. Vishnu Priya, "Numerical investigation and design optimization of Brimmed Diffuser Wind Lens around a Wind Turbine", The Eighth Asia-Pacific Conference on Wind Engineering, December 10-14 2013, Chennai, India

[7] S. A. H. Jafari and B. Kosasih, "Flow analysis of shrouded small Wind Turbine with a simple frustum diffuser with computational fluid dynamics simulations", Journal of Wind Engineering and Industrial Aerodynamics

[8] Peter J. Schubel and Richard J. Crossley, "Wind turbine blade design", Energies ISSN 1996-1073, 6 September 2012, DOI: 10.3390/en5093425

[9] Magdi Ragheb and Adam M. Ragheb, "Wind turbines theory – The Betz equation and optimal rotor tip speed ratio", Fundamental and Advanced topics in Wind Power, Dr. Rupp Carriveau (Ed.), SBN: 978-953-307-508-2

[10] Emre KOC, Onus Gunel and Tahir Yavuz, "Comparison of QBlade and CFD results small scaled HAWT analysis", 5th International Conference on Renwable Energy Research and Applications, 20-23 Nov 2016, Birmingham, UK

[11] Bimsuk Kim, Woojane Kim, Sungyoul Bae, Jaehyung Park and Manneung Kim, "Aerodynamic design and performance analysis of multi-MW class wind turbine blade", Journal of Mechanical Science and Technology 25 (8) (2011) 1995-20

[12] Mithil M. Darpe. Sharayu K. Bandekar and Prof. S. V. Vanjari, "Review paper on Diffuser Augmented Wind Turbine (Wind Lens Turbine)", International Research Journal of Engineering and Technology (IRJET), Vol. 07, Issue 04, April 2020