

MODELING A SHROUD FOR THE WIND TURBINE AND CARRYING A FLUID FLOW ANALYSIS USING CFD SOFTWARE PACKAGE

Dr. E. S. Prakash¹, Dr. Madhukeshwara N², Veeresh G. Gunjalli³, Dadapeer D⁴

¹ Professor, Studies in Mechanical Engineering, U.B.D.T.C.E., Davanagere, karnataka, India

² Professor, Mechanical engineering, K.L.E.I.T., Hubballi, karnataka, India

³ Assistant Professor, Mechanical engineering, K.L.E.I.T., Hubballi, karnataka, India

⁴ Assistant Professor, Mechanical engineering, A.G.M.R.C.E.T., Hubballi, karnataka, India

Abstract - It is known that diffuser (Shroud, called in this work) increases the output power in small-scale wind turbine. Therefore flow analysis will be done using FLUENT (a program for CAE) to the diffuser of wind turbine.

A wind turbine blade profile will be modeled and flow analysis will be carried on the model. A shroud will be modeled for the wind turbine and similar analysis will be carried on the same wind turbine blade profile with shroud. A generation CFD result proves that the output power is increased up to 3 times with the shroud for same blade profile.

Key Words: Shroud. CFD Software.

1. INTRODUCTION

Alternative energy sources are an important subject for the application of an effective energy resource in the future since fossil fuels are depleting at faster rate. Furthermore, due to concerns for environmental issues, i.e., global warming, etc., the development and application of renewable and clean new energy are strongly expected. Among others, wind energy technologies have developed rapidly and are about to play a big role in a new energy field. However, in comparison with the overall demand for energy, the scale of wind power usage is still small; especially, the level of development is extremely small in some of the remote places. As for the reasons, various causes are conceivable. For example, the limited local area suitable for wind power plants, the complex terrain compared to that in some of the geographical locations and the turbulent nature of the local wind are pointed out. Therefore, the introduction of a new wind power system that produces higher power output even in areas where lower wind speeds and complex wind patterns are expected is strongly desired.

It is becoming necessary to fully understand how to improve wind turbine efficiency, as energy consumption and to bring down the cost to the economical level. The cost of oil and non-renewable resources is increasing day by day, and the depletion of these

resources will require a sustainable and environmentally friendly energy source. An improvement to wind turbine efficiency will allow the limits of today to be just fulfilled, and someday be able to extract all of the energy from the wind with only a few improvements in technology. A greater number of these high-efficiency turbines would lower the cost of energy and generate the power at lower cost.

Diffuser Augmented Wind Turbine (DAWT) is one of the major technique for improving turbine efficiency of a small scale wind turbine for the conventional horizontal axis wind turbine (HAWT). A Diffuser Augmented Wind Turbine (DAWT) has a duct that surrounds the wind turbine blades and increases in cross-sectional area in the streamwise direction. The resulting sub-atmospheric pressure within the diffuser draws more air through the blade plane, and hence more power can be generated compared to a "bare turbine" of the same rotor blade diameter.

2. LITERATURE REVIEW

Preliminary literature review is made to understand the fundamental concepts of the wind turbine. Some of the important contributions made by pioneers in this field during recent years are discussed below

Grady and Hayder [1] also modeled a diffuser to the bare wind turbine in order to increase the efficiency of the turbine in the CFD software and carried out the fluid flow analysis. The simulation results showed that there is an increase in velocity of wind by 54%.

Toya, et al. [2] modeled the convergent and divergent shroud using Ansys (a program of CAE). A fluid flow analysis is conducted on the turbine with shroud and without shroud. The results showed that the power output with shroud is increased by three times as compared to the bare turbine.

Palapum and Adun [3] developed the diffuser model for the water turbines in the Computational Fluid Dynamics (CFD) Software. The diffuser can amplify the water velocity, which in turn will results in the increases power output of the turbine. The investigation showed

that there is an increase in power coefficient of 2.10 times as compared to the bare turbine

3. PROBLEM DEFINITION AND METHODOLOGY

3.1 Description of Problem

It is known that power of the wind turbine depends on the velocity of the wind. So here an approach is made to increase the velocity of some available wind in atmosphere by constructing a shroud around the turbine. A shroud consisting of inner convergent part followed by a throat and divergent part is constructed in software GAMBIT with the parameters tabulated in table.3.1. From the concept of nozzle and diffuser it is clear that to increase the velocity of the flowing fluid, the cross-section of flow of fluid has to be reduced. By Trial and error method using CFD, the values of table 3.1 are found to be optimum, considering the concept of flow separation

Table.3.1 parameters used for construction of shroud

Parameters	Units	Values
Total length	mm	640
Outer diameter	mm	300
Throat diameter	mm	240
Inner convergent length	mm	200
Throat length	mm	240
Outer divergent length	mm	200

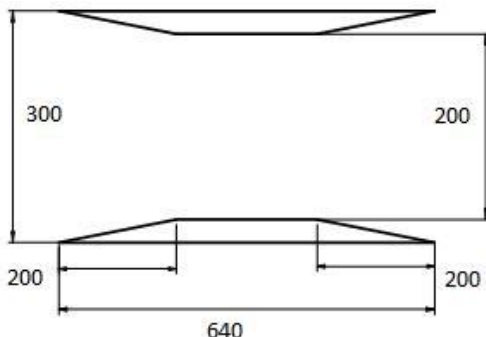


Fig. 3.1: Shroud

3.2 Methodology

Case 1: Without Shroud

Some available wind velocity in the atmosphere (say 10m/s) is made to approach directly on the turbine blade and force acting on the blade can be obtained by carrying CFD analysis over blade. Thus power can be obtained.

Case 2: With Shroud

Same available wind velocity in the atmosphere (say 10m/s) is now made to pass through shroud and approach on the turbine blade which will be kept inside the shroud and force acting on the blade can be obtained by carrying CFD analysis over blade. Thus power can be obtained.

As wind passes through the shroud its velocity will be increased due to which force acting on the blade will also increase and in turn increases the power. With the shroud parameters listed in table.3.1 it is to be determined that how many times the power will be increased. Also a maximum velocity region in the shroud will be determined by contours of velocity magnitude.

Analysis is carried out in two stages where 2D analysis will be carried out and comparison can be made in both with and without shroud cases and conclusion can be made.

In first stage a 2D shroud/diffuser will be modeled according to the geometry shown in fig.3.1 in a modeling software GAMBIT and meshing is also done in the software GAMBIT. The file will be then saved as *.msh (mesh file). The file will be imported to the solver FLUENT software and is read, checked and scaled for the required working unit. A Segregated type of solver is selected and boundary conditions are applied as shown in the fig 4.1. Then solution will be initialized and iterations will be carried out till convergence criteria are observed.

The results will be obtained in contours of pressure and velocity and maximum velocity region can be observed.

In second stage a 2D wind turbine blade will be modeled accordingly as shown in fig.4.7 in a modeling software GAMBIT and meshing is also done in the GAMBIT. The file will be then saved as *.msh (mesh file). The file will be imported to the solver FLUENT software and is read, checked and scaled for the required working unit. A Segregated type of solver is selected. Here the results of 1st stage of analysis are applied. Then solution will be initialized and iterations will be carried out till convergence criteria are observed.

Here results for blade geometry with and without shroud cases will be tabulated and compared and can be concluded which gives the better results.

4. CFD ANALYSIS OVER WIND TURBINE BLADE WITH AND WITHOUT SHROUD

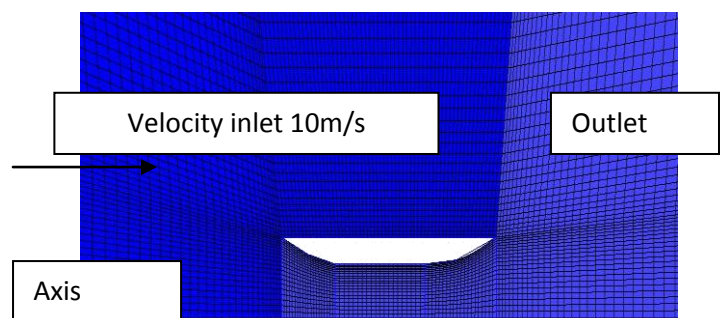


Fig. 4.1: Shroud with control volume

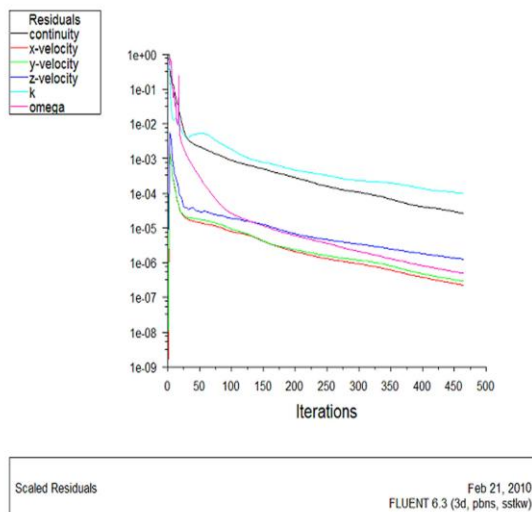


Fig.4.2 Convergence criteria

4.1 Results of CFD analysis on diffuser/shroud and its discussion

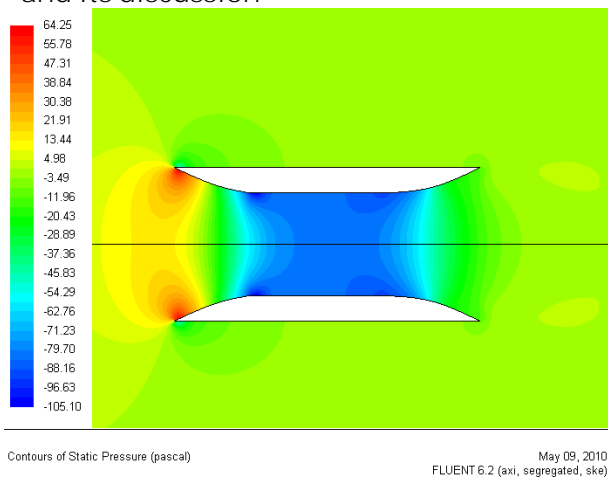


Fig.4.3 Contours of static Pressure

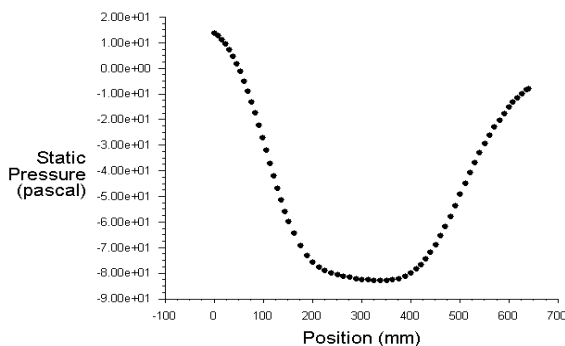


Fig.4.4 Contours of static Pressure-Isometric

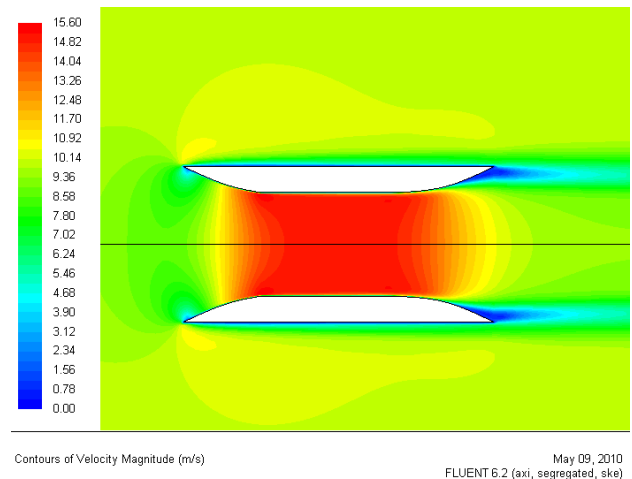


Fig.4.5 Contours of velocity Magnitude

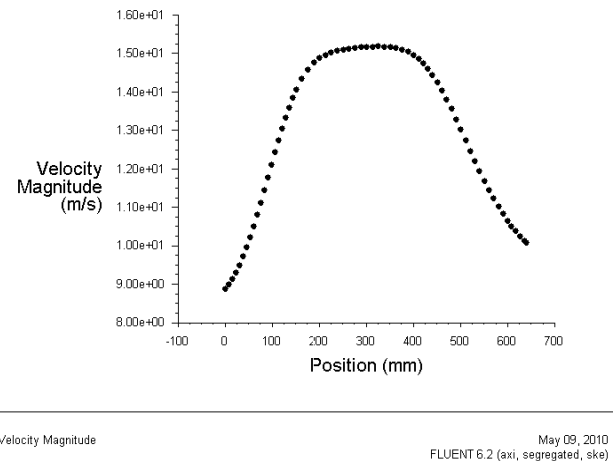


Fig.4.6 Contours of velocity Magnitude-Isometric

Here a shroud has been modeled and a control volume is defined as shown in fig.3.1. An atmospheric air flowing at a velocity of arbitrarily 10m/s has been made to flow through the control volume. Fig.4.3 and Fig.4.5 show the contours of static pressure and velocity. It is observed that the magnitude of velocity of air is increased to about 15.56m/s as it passes through shroud. Theoretical calculations in below also show approximately same results

So, velocity at the 240mm diameter cross section can be obtained as,

$$\begin{aligned}
 A_1 V_1 &= A_2 V_2 \\
 V_2 &= A_1 V_1 / A_2 \\
 V_2 &= d_1^2 V_1 / d_2^2 \\
 V_2 &= 0.3^2 * 10 / 0.24^2 \\
 V_2 &= 15.625 \text{ m/s}
 \end{aligned}$$

4.2 Results of CFD analysis on blade geometry and its discussion

Table.4.1 Nodal data of wind turbine blade geometry

0	0	0
0.0006	0.001687	0
0.009	0.002061	0
0.0015	0.002647	0
0.003	0.003705	0
0.006	0.005235	0
0.009	0.00637	0
0.012	0.007278	0
0.018	0.008672	0
0.024	0.009657	0
0.03	0.010319	0
0.036	0.0107	0
0.042	0.010795	0
0.048	0.010611	0
0.054	0.010174	0
0.06	0.009526	0
0.066	0.008705	0
0.072	0.007748	0
0.078	0.006682	0
0.084	0.005547	0
0.09	0.004381	0
0.096	0.003229	0
0.102	0.002145	0
0.108	0.001184	0
0.114	0.000419	0
0.12	0	0
0.0006	-0.001687	0
0.009	-0.002061	0
0.0015	-0.002647	0
0.003	-0.003705	0
0.006	-0.005235	0
0.009	-0.00637	0
0.012	-0.007278	0
0.018	-0.008672	0
0.024	-0.009657	0
0.03	-0.010319	0
0.036	-0.0107	0
0.042	-0.010795	0
0.048	-0.010611	0
0.054	-0.010174	0
0.06	-0.009526	0
0.066	-0.008705	0
0.072	-0.007748	0
0.078	-0.006682	0
0.084	-0.005547	0
0.09	-0.004381	0
0.096	-0.003229	0
0.102	-0.002145	0
0.108	-0.001184	0
0.114	-0.000419	0



Fig.4.7 Geometric model of Blade

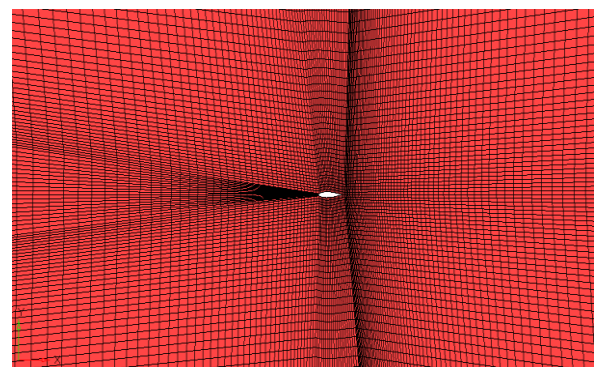
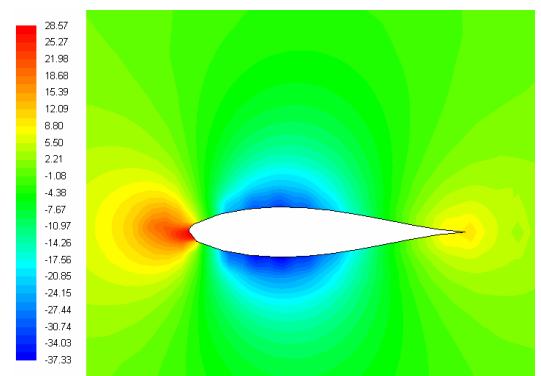
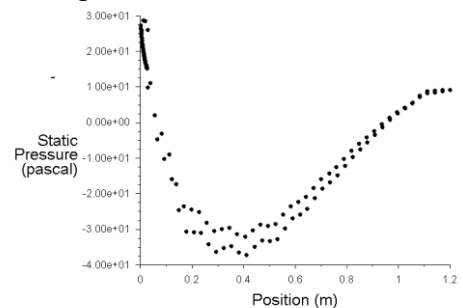


Fig.4.8 Grid Generation around the blade



Contours of Static Pressure (pascal) May 09, 2010
FLUENT 6.2 (2d, segregated, ske)

Fig.4.9 Contours of Static Pressure



Static Pressure May 09, 2010
FLUENT 6.2 (2d, segregated, ske)

Fig.4.10 Contours of Static Pressure

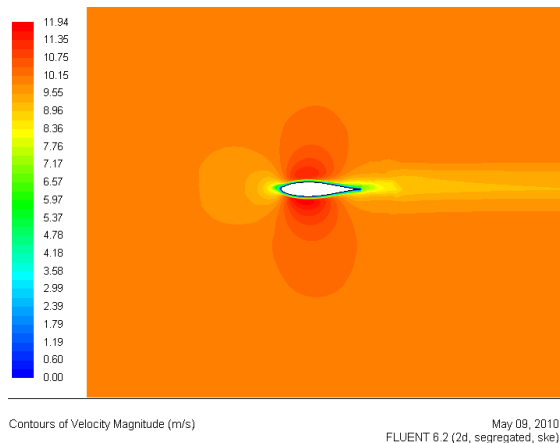


Fig.4.11 Contours of Velocity Magnitude

Table.4.2 Force acting on the blade

For Velocity of air applied directly on blade $V_1= 10$ m/s

Force vector: (1 0 0)			
zone name	pressure force n	viscous force n	total force n
blade	1.4258311	0.52649862	1.9523297
net	1.4258311	0.52649862	1.9523297

For Velocity of air applied on blade through diffuser $V_2= 15.56$ m/s (for $V_1=10$ m/s)

Force vector: (1 0 0)			
zone name	pressure force n	viscous force n	total force n
blade	2.8045716	1.1086727	3.9132444
net	2.8045716	1.1086727	3.9132444

An aerofoil structured blade has been modeled as shown in fig.4.7 and a flow analysis over blade has been carried out. Fig.4.9 and Fig.4.11 show the contours of static pressure and static velocity over blade. If air flowing at a velocity of 10 m/s approaches blade then the forces acting on blade is observed to be 1.95 N and similarly If air flowing at a velocity of 15 m/s approaches blade then the forces acting on blade is observed to be 3.91 N. The values of forces, velocities and power are tabulated in table.4.3.

Table.4.3 Tabulation of Forces and Velocities and Power

Velocity	Forces acting on blade	Power P= FXV
V=10 m/s	F=1.95N	19.5W
V=15 m/s	F=3.91N	58.65W

The air flowing at arbitrarily 10m/s approaches the blade at same velocity of 10m/s. For air approaching the blade at a velocity of 10 m/s, power output P=19.5 W from table 4.2.

The air flowing at arbitrarily 10 m/s as it passes through shroud its velocity increases to approximately 15 m/s and approaches the blade at velocity of 15 m/s. For air approaching the blade at a velocity of 15 m/s, power output P=58.65 W from table 4.2.

5. CONCLUSION

Hence, it's clear that with shroud the velocity of air increases and in turn increases the force acting on blade.

The power output without shroud P=19.5W

The power output with shroud P=58.65W.

The shroud increases the power output for small scale wind turbine by nearly 3 times.

6. REFERENCES

1. Grady M. Isensee, Hayder Abdul-Razzak, "Modeling and Analysis of Diffuser Augmented Wind Turbine", International Journal of Energy Science(IJES), Vol.2 Iss.3 2012 PP.84-88
2. H. Toya, T. Kusakabe, T. Yuji, "Fluid Flow Analysis and Design of a Shroud for Wind Turbine Using ANSYS", Proceeding of International Conference on Electrical Machines and Systems 2007, Oct. 8~11, Seoul, Korea
3. Palapum Khunthongjan and Adun Janyalertadun, "A study of diffuser angle effect on ducted water current turbine performance using CFD", songklanakar journal of science and technology, 34 (1), 61-67, Jan. - Feb. 2012

BIOGRAPHIES



Dr. E. S. Prakash



Dr. Madhukeshwara N.



Veeresh G. Gunjalli



Dadapeer D.